Introduction to InfoWater

Welcome to InfoWater - The most powerful water modeling software in the world!

Teach Me More About...

Overview

Today's water utilities are realizing the benefits of geographic information system (GIS) technology in managing their water supply and distribution system infrastructures. The effective management of these systems; however, is a multifarious task involving the use of a large amount of digital information with sound engineering analyses to adequately understand the system hydraulic performance, water quality condition, energy consumption, and to efficiently administer the existing resources. This has led to a greater need for a seamless integration of GIS technology with advanced network modeling techniques to ensure that sound, cost-effective engineering solutions can be accomplished in the design, planning, and operation of these systems. InfoWater was developed (by engineers for engineers) specifically to fully address this need by producing a common platform and interface for these synergistic applications.

InfoWater is a fully GIS integrated water distribution modeling and management software application. Built atop ESRI ArcGIS using the latest Microsoft .NET and ESRI ArcObjects component technologies, InfoWater seamlessly integrates advanced water network modeling and optimization functionality with the latest generation of ArcGIS. InfoWater capitalizes on the intelligence and versatility of the geodatabase architecture to deliver unparalleled levels of geospatial analysis, infrastructure management and business planning. Its unique interoperable geospatial framework enables world-record performance, scalability, reliability, functionality and flexibility - all within the powerful ArcGIS environment.

InfoWater offers direct ArcGIS Integration enabling engineers and GIS professionals to work simultaneously on the same integrated platform. It allows you to command powerful GIS analysis and hydraulic modeling in a single environment using a single dataset. You can now create, edit, modify, run, map, analyze, design and optimize your water network models and instantly review, query and display simulation results from within ArcGIS.

InfoWater extends the core features of ArcGIS, providing a comprehensive geospatial environment for complete network model construction, graphical editing, network simulation, results presentation, map generation, and enterprise-wide data sharing and exchange. It also adds rich discipline-specific functionality to ArcGIS designed to streamline and facilitate all aspects of the water distribution modeling workflow. You can now leverage your existing data investments and deliver informed GIS solutions to help you meet and exceed drinking water quality standards, optimize system performance and capital improvements, enhance operations, and achieve customer satisfaction in a timely manner and at a minimum cost.

InfoWater bridges the gap between network modeling and ArcGIS software to support many types of applications in distribution system analyses, including:

- **Master Planning**
- **Fire Flow Assessment**
- **Facility Sizing**
- **Operational Study**
- **Infrastructure Rehabilitation**
- **Emergency Response**
- **Real-time Simulation**
- **Water Quality Evaluation**
- **Multi-Quality Source Blending**
- **Sampling Program Design**

- **Satellite Treatment Identification**
- **Operator Training**
- **Energy Consumption Minimization**
- **Pump Scheduling**
- **Capital Budgeting**
- **Leakage Control**
- **Conservation Studies**
- **System Expansion and Improvement**
- **New System Design**

These comprehensive capabilities will greatly assist you in making informed decisions to ensure the most cost and energy efficient water system – from ongoing operation and maintenance to rehabilitation, enhancement, expansion, and new design.

Our high-level, state-of-the-art research and development effort in GIS-based network analysis is continuing at a rapid pace and we intend to update and refine InfoWater to reflect this progress. We are pleased to be at the forefront of this computer technology and to continue to advance it to an unprecedented level of reliability and performance.

Data Elements

A data element is any representative facility of a water system and, in InfoWater, is either a pipe, pump, valve, junction, tank or reservoir. Click any of the links below to learn more.

Pipes convey water from one point in the network to another. InfoWater assumes that all pipes are flowing full at all times. Flow direction is from the end at higher hydraulic head (internal energy per weight of water) to that at lower head. A pipe has a start and an end node.

Nodes - Nodes represent the endpoints of each pipe and are also locations where two or more pipes join. A stationary facility identified by a single point (X,Y). Nodes can be any of the following:

- Junction Nodes
- Storage Nodes (Tanks and Reservoirs)
- Pumps
- Valves

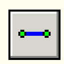
Note: All data elements have a unique 20 character alpha-numeric identification field.

Pipes

A pipe conveys water from one node to another.

- Create a pipe?
- Edit a pipe?
- Edit the data for a group of pipes?
- Delete a pipe?
- Recall a deleted pipe?
- Redraw a pipe?

Create a Pipe

To create a pipe, click on the **Add Pipe** icon  on your **InfoWater Edit Network** tool bar.

Once the command is initiated, select the *From Node* for the new pipe by clicking on the appropriate node and begin digitizing the pipe. Intermediate shape-defining vertices can be placed by clicking the mouse in any location. Configure the pipe accordingly and double-click the mouse to specify the *To Node* to end the digitization process.

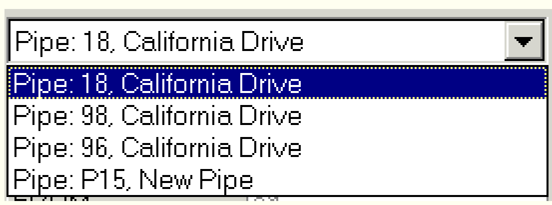
Once the pipe has been added, pipe details can be entered in the **Attribute Browser** window or through the **InfoWater Control Center -> InfoWater** button **-> Edit** menu **-> DB Editor**.

Note: The pipe being added must be snapped to an existing node (junction, tank, reservoir, pump or valve). Either create the node prior to the pipe creation process and/or use the Network Digitization command to create the system "on the fly".

Edit a Pipe

Graphic Selection - To edit the pipe graphically do one of the following:

- Choose the **Select** icon  from the **Attribute Browser** window and click on the specific pipe.
- if a Pipe has been prior selected and is a part of the Browse history, it can be re-selected by choosing it from the **Browse-History** section of the **Attribute Browser** window.



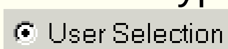
Data Edit - To edit the data related to a pipe, first select the pipe using the tools above. Once selected, edit pipe related data by adding/modifying the **Attribute Browser** window. You may also add/modify through the different pipe data tables.

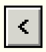
Edit the Data for a Group of Pipes

Group Edits on a group of pipes may be done in one of many ways :

- Using the Domain Manager: Create a domain selecting the specific pipes that you want to edit. Refer to the section on **Domain Manager** for details on the domain creation process. Once the domain has been created, choose the **DB Editor** command from the **InfoWater Control Center -> InfoWater** button **-> Edit** pull-down menu. Depending on the data you want to edit choose the appropriate table type from among the different database tables. Choose 

as the **Data Scope** and click on **OK**.

- Using the User Selection: Select the **DB Editor** option from the **InfoWater Control Center -> InfoWater** button **-> Edit** pull-down menu. Depending on the data you want to edit choose the appropriate table type from among the different database tables. Choose 

as the **Data Scope** and choose the **Select**  command. Now select the desired pipes and right click and choose **Enter** to access the DB Tables for the selected pipes.

Once the specific tables have been opened, enter or Block Edit data to make changes in bulk.

Group Edit for certain specific features may be done by choosing the **Group Editing** (**on Domain** and/or **Selection**) option under the **InfoWater Control Center -> InfoWater** button **-> Edit** pull-down menu. Refer to the section on [Group Editing] to learn more.

Delete a Pipe

To delete a pipe, select the **Delete Pipe** icon  from the **InfoWater Edit Network** tool bar**.** InfoWater prompts the user to confirm deletion if the **Delete Confirmation** command is enabled through the **InfoWater Control Center -> InfoWater** button **-> Tools** menu -> **Project Preferences** command -> **Operation Settings** tab.

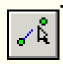
Note: The selected pipes are deleted and saved in the InfoWater recycle bin and can be recalled by using the **Recall** command. However, if **Auto Database Packing** is enabled (**InfoWater Control Center -> InfoWater** button **-> Tools** menu -> **Project Preferences**

command -> **Operation Settings** tab), then any element deleted is permanently deleted from the database and cannot be recalled.

Recall a Deleted Pipe

To recall a deleted pipe, go to **Utilities** menu from the **InfoWater Control Center -> InfoWater** button, point to **Recall**, then select **Pipe**. In order to perform the recall, the user must know the ID of the pipe that was deleted. You may choose the **Show Deleted Pipe** command to obtain a list of pipes deleted from the InfoWater project but saved in the InfoWater recycle bin.

Redraw a Pipe

Redrawing a pipe is the same as modifying its geometry. To redraw a pipe, launch the **Redraw** command  from the **InfoWater Edit Network** toolbar. Then select the pipe that you want to redraw by clicking on it. Now redigitize the new location of the pipe currently highlighted by choosing the new upstream node graphically. Follow the steps underlined above for the Creation of a Pipe to continue with the Pipe redraw.

Other Related Topics - Data Elements**,** Junction Nodes**,** Network**,** Pumps**,** Storage Nodes**,** Valves**.**

Junction Nodes

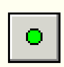
Junction nodes are points placed at the intersection of two or more pipes (intersections or tees), at points of water consumption or inflow, at points where specific analysis values (e.g., pressure, concentration) are desired, and at points where pipe attributes (e.g., diameter, material) change. Junction nodes must have their elevation above datum (e.g., sea level) specified in order for pressure computations to be meaningful.

How do I...

- Create a Junction?
- Edit a Junction?
- Edit the data for a group of Junctions?
- Delete a Junction?
- Recall a deleted Junction?

Create a Junction

To create a junction node, do the following:

- Click on the **Add/Insert Junction** icon  on your **InfoWater Edit Network** toolbar.

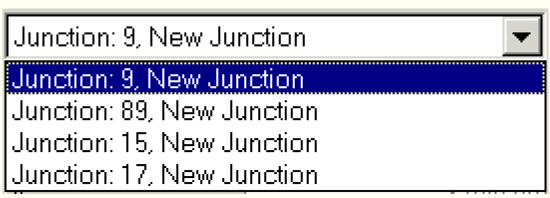
Once the command is initiated, select the location of the new node by left mouse clicking anywhere on the map display. Suggest a new ID and/or a description on the **Junction Identification** dialog box and click on the **OK** button to accept the ID and create the junction. The junction should visually appear at the specified location.

Note: You can also use the [Digitize Network] icon from the **InfoWater Edit Network** to create a pipe and node network "on the fly". Once digitized, the user is able to add relevant modeling data in the **Attribute Browser** window.

Edit a Junction

Graphic Selection - To select a junction node for data edit do one of the following:

- Choose the **Select** icon from the **Attribute Browser** window and click on the specific Junction.
- If a Junction has been prior selected and is a part of the Browse history, it can be re-selected by choosing it from the **Browse-History** section of the **Attribute Browser** window.



Data Edit - To edit the data related to a junction, first select the junction using the tools above. Once selected, edit Junction related data by adding/modifying the **Attribute Browser** window. Additionally modifications may be done by editing the **DB Tables** that may be accessed from the **InfoWater Control Center -> InfoWater** button **->** **Edit** Pull-down menu -> **DB Editor** command.

Graphic Edit - To edit the node graphically do one of the following:

- From the **InfoWater Control Center** -> **InfoWater** button -> **Edit** pull-down menu choose the **Move Node** command. Left click once on the junction to select it and choose the new position and left click again to move the junction.

- Choose the **Move Node** icon from the **InfoWater Edit Network** toolbar. Left click once on the junction to select it and choose the new position and left click again to move the junction.

Note: If the user has not enabled "**Auto Length Calculation**" under the **InfoWater Control Center -> InfoWater** button **-> Tools -> Project Preferences** menu, all pipes connected to that node will not have their lengths automatically changed.

Edit the Data for a Group of Junctions

Group edits can be made by either creating a domain or by creating a selection.

- Using the Domain Manager: Create a domain selecting the specific junctions that you want to edit. Refer to the section on Domain Manager for details on the domain creation process. Once the domain has been created, choose the **DB Editor** command from the **InfoWater Control Center -> InfoWater** button **-> Edit** pull-down menu. Depending on the data you want to edit choose the appropriate table type from among the different database tables. Choose Domain as the **Data Scope** and click on **OK**.

- Using the User Selection: Select the **DB Editor** option from the **InfoWater Control Center -> InfoWater** button **-> Edit** pull-down menu. Depending on the data you want to edit choose the appropriate table type from among the different database tables. Choose User Selection as the **Data Scope** and choose the **Select** command. Now select the desired junctions and right click and choose **Enter** to access the DB Tables for the selected junctions.

Once the specific tables have been opened, enter or Block Edit data to make changes in bulk.

Group Edit for certain specific features may be done by choosing the **Group Editing** (**on Domain** and/or **Selection**) option under the **InfoWater Control Center -> InfoWater** button **-> Edit** pull-down menu. Refer to the section on Group Editing to learn more.

Delete a Junction

To delete a junction, select the **Delete Node** icon  from the **InfoWater Edit Network** tool bar**.** InfoWater prompts the user to confirm deletion (if the **Delete Confirmation** command is enabled through the **InfoWater Control Center -> InfoWater** button **-> Tools** menu -> **Project Preferences** command -> **Operation Settings** tab).

Note: All pipes attached to the Junction being deleted will also be deleted. The Junction and associated pipes are deleted and saved in the InfoWater recycle bin and can be recalled from the database using the **InfoWater Control Center -> InfoWater** button **-> Utilities -> Recall** command. However, if **Auto Database Packing** is enabled (**InfoWater Control Center -> InfoWater** button **-> Tools** menu **-> Project Preferences** command **-> Operation Settings** tab), then any element deleted is permanently deleted from the database and cannot be recalled.

Recall a Deleted Junction

To recall a deleted junction, go to **Utilities** menu from the **InfoWater Control Center -> InfoWater** button, point to **Recall**, then select

Node. In order to perform the recall, the user must know the ID of the junction that was deleted. You may choose the **Show Deleted Node** command to obtain a list of nodes deleted from the InfoWater project but saved in the InfoWater recycle bin.

Other Related Topics - Data Elements**,** Network**,** Pipes**,** Pumps**,** Storage Nodes**,** Valves**.**

Storage Nodes

Storage nodes are those points where a free water surface exists (water at atmospheric pressure) and the hydraulic head is equal to the free water surface. Storage nodes can be further separated into two categories:

- **Reservoirs** - Remain at a constant water level irrespective of the flow unless they are specified as variable-head reservoirs (EPS only). Reservoirs have unlimited volume and are generally used to represent a lake or other inexhaustible supply source.

- **Tanks** - Distinguished from reservoirs as having a known finite volume and water surface levels change with time (EPS only) as water flows into or out of them.

How do I...

- Create a storage node?
- Edit a storage node?
- Delete a storage node?
- Recall a deleted storage node?

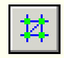
Create a Storage Node

To create a storage node, do the following:

- Click on the **Add/Insert Tank** and/or **Reservoir** icon / on your **InfoWater Edit Network** toolbar.

Once the command is initiated, select the location of the new node by left mouse clicking anywhere on the map display. Suggest a new

ID and/or a description on the **Tank** and/or **Reservoir Identification** dialog box and click on the **OK** button to accept the ID and create the storage node. The tank and/or reservoir should visually appear at the specified location.

Note: You can also use the **Digitize Network** icon  from the **InfoWater Edit Network** to create a pipe and node network at the same time. Once digitized, the user is able to add relevant modeling data in the **Attribute Browser** window.

Types of Tanks

There are 2 different types of tanks the user is able to input in InfoWater. Click on any tank type below to learn more.

0. *Cylindrical Tank*
1. *Variable Area Tank*

Types of Reservoirs

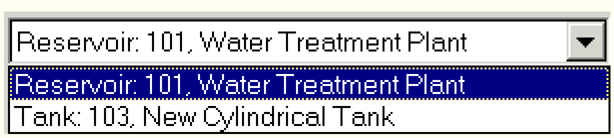
There are 2 different types of Reservoirs the user is able to input in InfoWater. Click on any reservoir type below to learn more.

0. *Fixed Head Reservoir*
1. *Variable Head Reservoir*

Edit a Storage Node

Graphic Selection - To select a Storage Node for data edit do one of the following:

- Choose the **Select** icon  from the **Attribute Browser** window or from the **InfoWater Edit Network** toolbar and click on the specific tank or reservoir.
- If a tank and/or reservoir has been prior selected and is a part of the Browse history, it can be re-selected by choosing it from the **Browse-History** section of the **Attribute Browser** window.



Data Edit - To edit the data related to a storage node, first select the tank and/or reservoir using the tools described above. Once selected, edit related data by adding/modifying the **Attribute Browser** window. Additionally modifications may be done by editing the **DB Tables** that may be accessed from the **InfoWater Control Center -> InfoWater** button **-> Edit** Pull-down menu -> **DB Editor** command.

Graphic Edit - To edit the storage node (tanks or reservoirs) graphically do the following:

- Choose the **Move Node** icon from the **InfoWater Edit Network** toolbar. Left click once on the storage node to select it and choose the new position and left click again to move the tank and/or reservoir.

Note: If the user has not enabled "**Auto Length Calculation**" under the **InfoWater Control Center -> InfoWater** button **-> Tools -> Project Preferences** menu **-> Operation Settings** tab, all pipes connected to that node will not have their lengths automatically changed.

Edit a Group of Storage Nodes

Group edits can be made by either creating a domain or by creating a selection.

- Using the Domain Manager: Create a domain selecting the specific tanks and/or reservoirs that you want to edit. Refer to the section on Domain Manager for details on the domain creation process. Once the domain has been created, choose the **DB Editor** command from the **InfoWater Control Center -> InfoWater** button **-> Edit** pull-down menu. Depending on the data you want to edit choose the appropriate table type from among the different database tables. Choose as the **Data Scope** and click on **OK**.

- Using the User Selection: Select the **DB Editor** option from the **InfoWater Control Center -> InfoWater** button **-> Edit** pull-down menu. Depending on the data you want to edit choose the appropriate table type from among the different database tables. Choose as the **Data Scope** and choose the **Select** command. Now select the desired tanks and/or reservoirs and right click and choose **Enter** to access the DB Tables for the selected junctions.

Once the specific tables have been opened, enter or Block Edit data to make changes in bulk.

Group Edit for certain specific features may be done by choosing the **Group Editing** (**on Domain** and/or **Selection**) option under the **InfoWater Control Center -> InfoWater** button **-> Edit** pull-down menu. Refer to the section on Group Editing for further details.

Delete a Storage Node

To delete a storage node, select the **Delete Node** icon  from the **InfoWater Edit Network** tool bar**.** InfoWater prompts the user to confirm deletion (if the **Delete Confirmation** command is enabled through the **InfoWater Control Center -> InfoWater** button **-> Tools** menu -> **Project Preferences** command -> **Operation Settings** tab).

Note: All pipes attached to the storage node being deleted will also be deleted. The tank and/or reservoir and associated pipes are deleted and saved in the InfoWater recycle bin and can be recalled from the database using the recall command. However, if **Auto Database Packing** is enabled (**InfoWater Control Center -> InfoWater** button **-> Tools** menu -> **Project Preferences** command -> **Operation Settings** tab), then any element deleted is permanently deleted from the database and cannot be recalled.

Recall a Deleted Storage Node

To recall a deleted storage node (tanks and/or reservoirs), go to **Utilities** menu from the **InfoWater Control Center -> InfoWater** button, point to **Recall**, then select **Node**. In order to perform the recall, the user must know the ID of the storage node that was deleted. You may choose the **Show Deleted Node** command to obtain a list of nodes deleted from the InfoWater project but saved in the InfoWater recycle bin.

Other Related Topics - Data Elements**,** Junction Nodes**,** Network**,** Pipes**,** Pumps**,** Valves**.**

Valves

A control valve regulates either flow or pressure in a distribution system. Valves are modeled as nodes in InfoWater.

How do I...

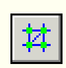
- Create a valve (types)?
- Edit a valve?
- Delete a valve?
- Recall a deleted valve?

Create a Valve

To create a valve, do the following:

- Click on the **Add/Insert Valve** icon  on your **InfoWater Edit Network** toolbar.

Once the command is initiated, select the location of the new node by left mouse clicking anywhere on the map display. Suggest a new ID and/or a description on the **Valve Identification** dialog box and click on the **OK** button to accept the ID and create the valve. The valve should visually appear at the specified location.

Note: You can also use the **Digitize Network** icon  from the **InfoWater Edit Network** to create a pipe and node network at the same time. Once digitized, the user is able to add relevant modeling data in the **Attribute Browser** window.

Types of Valves

There are 8 types of valves. Click on any valve to learn more.

0. *Pressure Reducing Valves (PRV)*
1. *Pressure Sustaining Valves (PSV)*
2. *Pressure Breaker Valves (PBV)*
3. *Flow Control Valves (FCV)*
4. *Throttle Control Valves (TCV)*
5. *General Purpose Valves (GPV)*
6. *Float Valves*
7. *Vacuum Breaker Valves*

Other Types of Valves (Pipe Dependent)

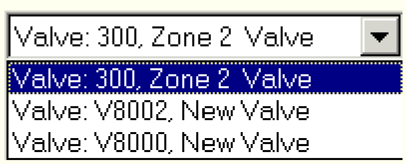
1. *Check Valves*
2. *Flow Totalizers*

Note: To represent a static valve (e.g., gate valve), a minor loss can be assigned to a pipe. Click here to learn more.

Edit a Valve

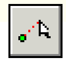
Graphic Selection - To select a Valve for data edit do one of the following:

- Choose the **Select** icon  from the **Attribute Browser** window or from the **InfoWater Edit Network** toolbar and click on the specific valve.
- If a valve has been prior selected and is a part of the Browse history, it can be re-selected by choosing it from the **Browse-History** section of the **Attribute Browser** window.



Data Edit - To edit the data related to a valve, first select the valve using the tools above. Once selected, edit valve related data by adding/modifying the **Attribute Browser** window. Additionally modifications may be done by editing the **DB Tables** that may be accessed from the **InfoWater Control Center -> InfoWater** button **-> Edit** Pull-down menu -> **DB Editor** command.

Graphic Edit - To edit the control valve node graphically do one of the following:

- Choose the **Move Node** icon  from the **InfoWater Edit Network** toolbar. Left click once on the valve to select it and choose the new position and left click again to move the valve.

Note: If the user has not enabled "**Auto Length Calculation**" under the **InfoWater Control Center -> InfoWater** button **-> Tools -> Project Preferences** menu -> **Operation Settings** tab, all pipes connected to that valve will not have their lengths automatically changed.

Edit a Group of Valves

Group edits can be made by either creating a domain or by creating a selection.

- Using the Domain Manager: Create a domain selecting the specific valves that you want to edit. Refer to the section on Domain Manager for details on the domain creation process. Once the domain has been created, choose the **DB Editor** command from the **InfoWater Control Center -> InfoWater** button **-> Edit** pull-down menu. Depending on the data you want to edit choose the appropriate table type from among the different database tables. Choose Domain as the **Data Scope** and click on **OK**.

- Using the User Selection: Select the **DB Editor** option from the **InfoWater Control Center -> InfoWater** button **-> Edit** pull-down menu. Depending on the data you want to edit choose the appropriate table type from among the different database tables. Choose User Selection as the **Data Scope** and choose the **Select** command. Now select the desired valves and right click and choose **Enter** to access the DB Tables for the selected valves.

Once the specific tables have been opened, enter or Block Edit data to make changes in bulk.

Group Edit for certain specific features may be done by choosing the **Group Editing** (**on Domain** and/or **Selection**) option under the **InfoWater Control Center -> InfoWater** button **-> Edit** pull-down menu. Refer to the section on Group Editing for further details.

Delete a Valve

To delete a valve, select the **Delete Node** icon  from the **InfoWater Edit Network** tool bar**.** InfoWater prompts the user to confirm deletion (if the **Delete Confirmation** command is enabled through the **InfoWater Control Center -> InfoWater** button **-> Tools** menu -> **Project Preferences** command -> **Operation Settings** tab).

Note: All pipes attached to the valve being deleted will also be deleted. The valve and associated pipes are deleted and saved in the InfoWater recycle bin and can be recalled from the database using the **Recall** command. However, if **Auto Database Packing** is enabled (**InfoWater Control Center -> InfoWater** button **-> Tools** menu -> **Project Preferences** command -> **Operation Settings** tab), then any element deleted is permanently deleted from the database and cannot be recalled.

Recall a Deleted Valve

To recall a deleted valve, go to **Utilities** menu from the **InfoWater Control Center -> InfoWater** button, point to **Recall**, then select **Node**. In order to perform the recall, the user must know the ID of the valve that was deleted. You may choose the **Show Deleted Node** command to obtain a list of nodes deleted from the InfoWater project but saved in the InfoWater recycle bin.

Important Rules About Valves:

- Control Valves cannot be directly connected to pipes with Check Valves.

- Pressure settings for valves are pressures and not total head (or hydraulic grade line elevation).

Other Related Topics - Data Elements**,** Junction Nodes**,** Network**,** Pipes**,** Pumps**,** Storage Nodes**.**

Pumps

A pump imparts energy to a fluid thereby raising its hydraulic head. The relationship describing the head imparted to a fluid as a function of its flow rate through the pump is termed the pump characteristic curve. Pumps are modeled as *Nodes* in InfoWater.

Because pumps impart energy to a water system, they may be analyzed during an **EPS** for Energy Management.

How do I...

- Create a pump (types of pumps)?
- Edit a pump?
- Edit a Group of pumps?
- Delete a pump?
- Recall a deleted pump?
- Analyze a VFD pump?

Create a Pump

To create a pump, do the following:

- Click on the **Add/Insert Pump** icon  on your **InfoWater Edit Network** toolbar.

Once this command is initiated, select the location of the new pump by left mouse clicking anywhere on the map display. Suggest a new ID and/or a description on the **Pump Identification** dialog box and click on the **OK** button to accept the ID and create the pump. The pump should visually appear at the specified location.

Note: You can also use the **Digitize Network** icon from the **InfoWater Edit Network** toolbar to create a pipe and node network at the same time. Once digitized, the user is able to add relevant modeling data in the **Attribute Browser** window.

Types of Pumps

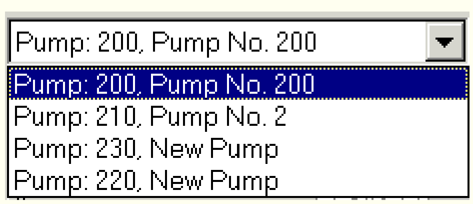
There are 4 different types of pumps the user is able to input into InfoWater. Click on any pump to learn more.

0. *Constant Power Input*
1. *Design Point Curve*
2. *Exponential 3-Point Curve*
3. *Multi-Point Curve*

Edit a Pump

Graphic Selection - To select a Pump for data edit do one of the following:

- Choose the **Select** icon  from the **Attribute Browser** window and click on the specific pump.
- If a pump has been prior selected and is a part of the Browse history, it can be re-selected by choosing it from the **Browse-History** section of the **Attribute Browser** window.



Data Edit - To edit the data related to a pump, first select the pump using the tools above. Once selected, edit pump related data by adding/modifying the **Attribute Browser** window. Additionally modifications may be done by editing the **DB Tables** that may be accessed from the **InfoWater Control Center -> InfoWater** button **-> Edit** Pull-down menu -> **DB Editor** command.

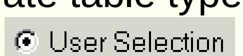
Note: If the user has not enabled "**Auto Length Calculation**" under the **InfoWater Control Center -> InfoWater** button **-> Tools -> Project Preferences** menu, all pipes connected to that node will not have their lengths automatically changed.

Edit the Data for a Group of Pumps

Group edits can be made by either creating a domain or by creating a selection.

- Using the Domain Manager: Create a domain selecting the specific pumps that you want to edit. Refer to the section on Domain Manager for details on the domain creation process. Once the domain has been created, choose the **DB Editor** command from the **InfoWater Control Center -> InfoWater** button **-> Edit** pull-down menu. Depending on the data you want to edit choose the appropriate table type from among the different database tables. Choose 

as the **Data Scope** and click on **OK**.

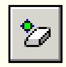
- Using the User Selection: Select the **DB Editor** option from the **InfoWater Control Center -> InfoWater** button **-> Edit** pull-down menu. Depending on the data you want to edit choose the appropriate table type from among the different database tables. Choose 

as the **Data Scope** and choose the **Select**  command. Now select the desired pumps and right click and choose **Enter** to access the DB Tables for the selected pumps.

Once the specific tables have been opened, enter or Block Edit data to make changes in bulk.

Group Edit for certain specific features may be done by choosing the **Group Editing** (**on Domain** and/or **Selection**) option under the **InfoWater Control Center -> InfoWater** button **-> Edit** pull-down menu. Refer to the section on Group Editing for further details.

Delete a Pump

To delete a pump, select the **Delete Node** icon  from the **InfoWater Edit Network** toolbar**.** InfoWater prompts the user to confirm deletion (if the **Delete Confirmation** command is enabled through the **InfoWater Control Center -> InfoWater** button **-> Tools** menu -> **Project Preferences** command -> **Operation Settings** tab). (Note: All pipes attached to the pump being deleted will also be deleted.) The pumps and associated pipes are deleted and saved in the InfoWater recycle bin and can be recalled from the database using the **Recall** command.

Note: If **Auto Database Packing** is enabled (**InfoWater Control Center -> InfoWater** button **-> Tools** menu -> **Project Preferences** command -> **Operation Settings** tab), then any element deleted is permanently deleted from the database and cannot be recalled.

Recall a Deleted Pump

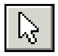
To recall a deleted pump, go to **Utilities** menu from the **InfoWater Control Center -> InfoWater** button, point to **Recall**, then select **Node**. In order to perform the recall, the user must know the ID of the pump that was deleted. You may choose the **Show Deleted Node** command to obtain a list of nodes deleted from the InfoWater project but saved in the InfoWater recycle bin.

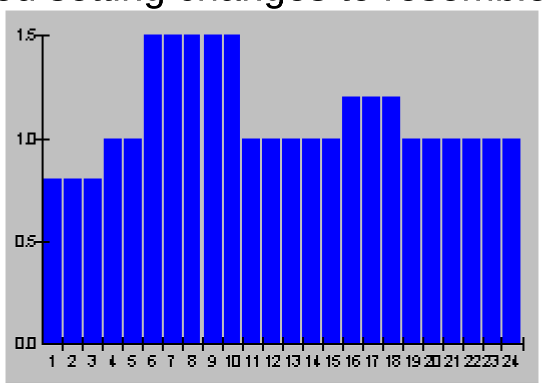
Variable Speed Pumps

In addition to the normal simple controls and rule based controls that can be applied to a pump, a pump's operation can also be described by assigning it a time pattern of relative speed settings. For variable speed pumps, the pump curve shifts as the speed changes.

Variable speed pumps can be considered by specifying that their speed setting be changed under these same types of conditions. By definition, the original pump curve supplied to the program has a relative speed setting of 1. If the pump speed doubles, then the relative setting would be 2; if run at half speed, the relative setting is 0.5 and so on. Changing the pump speed shifts the position and shape of the pump curve .

Assigning a Known Speed Pattern to a Pump

To assign a known speed pattern to a pump, highlight the pump with the **Select** tool  from the **Attribute Browser** window. Choose the **Control** command from the **Tools** icon  on the **Attribute Browser** window. When the **Pump Control** dialog box appears, select the **Pattern** radial button under the **Method** section and from the **Pattern ID** drop down box, select the pattern that describes the pump speed settings. Such a pattern may resemble the following: (Notice how the speed setting changes to resemble throttling throughout the day.)



Using a VSP/FPP Analysis to Create a Speed Pattern

In many instances, the user will not know how the speed of the pump changes over time. Because of this variable, InfoWater allows for the creation of this pattern through the use of a VSP/FPP Analysis which is done through the Standard tab of the Run Manager. Click here to learn more about conducting a VSP/FPP Analysis to create a variable speed pattern for a pump.

Additionally simple controls and rule based controls may also be applied to pumps to specify pump operations.

Note: Only pumps described by constant power input (hp or kW) can be connected to pipes with check valves. Pumps described by characteristic curves must not be connected to pipes with check valves.

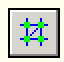
Note: Constant power pumps will allow flow reversal and should not be used under a zero flow condition (no flow passing through the pump).

Other Related Topics - Data Elements, Junction Nodes, Network, Pipes, Storage Nodes, Valves.

Digitize Network

The Digitize Network command is most helpful when digitizing pipes and nodes in bulk. The order of the pipe and node creation does not matter when using the Digitize Network command. This method is most useful when creating a network "on the fly". Data corresponding to the different elements may be entered once the network is in place through the database tables.

Methodology

- Launch the **InfoWater Edit Network** toolbar from the **View** menu **-> Toolbars** command
- Choose the **Digitize Network**  icon from the **InfoWater Edit Network** toolbar and click on it.
- Select an upstream node if one exists else left click on the part of the network where you wish to create one.
- Select intermediate vertices to identify pipe curvature by left clicking at various points.
- Finally double click to indicate the position of the end node. Continue the process till your entire network has been digitized or till you are satisfied with the digitization process.
- Once the digitization process has been completed, enter data into the element database tables by using the **DB Editor** command from the **InfoWater Control center -> InfoWater** button **-> Edit** pull-down menu.

Note: You may change the **Next Node Type** under the **Digitize** section of your **InfoWater Control center -> InfoWater** button **->**

Tools pull-down menu **-> Project Preferences** command -> **Operation Settings** tab.

Other Related Topics - Data Elements**,** Junction Nodes**,** Pipes**,** Pumps**,** Storage Nodes**,** Valves**.**

User Interface

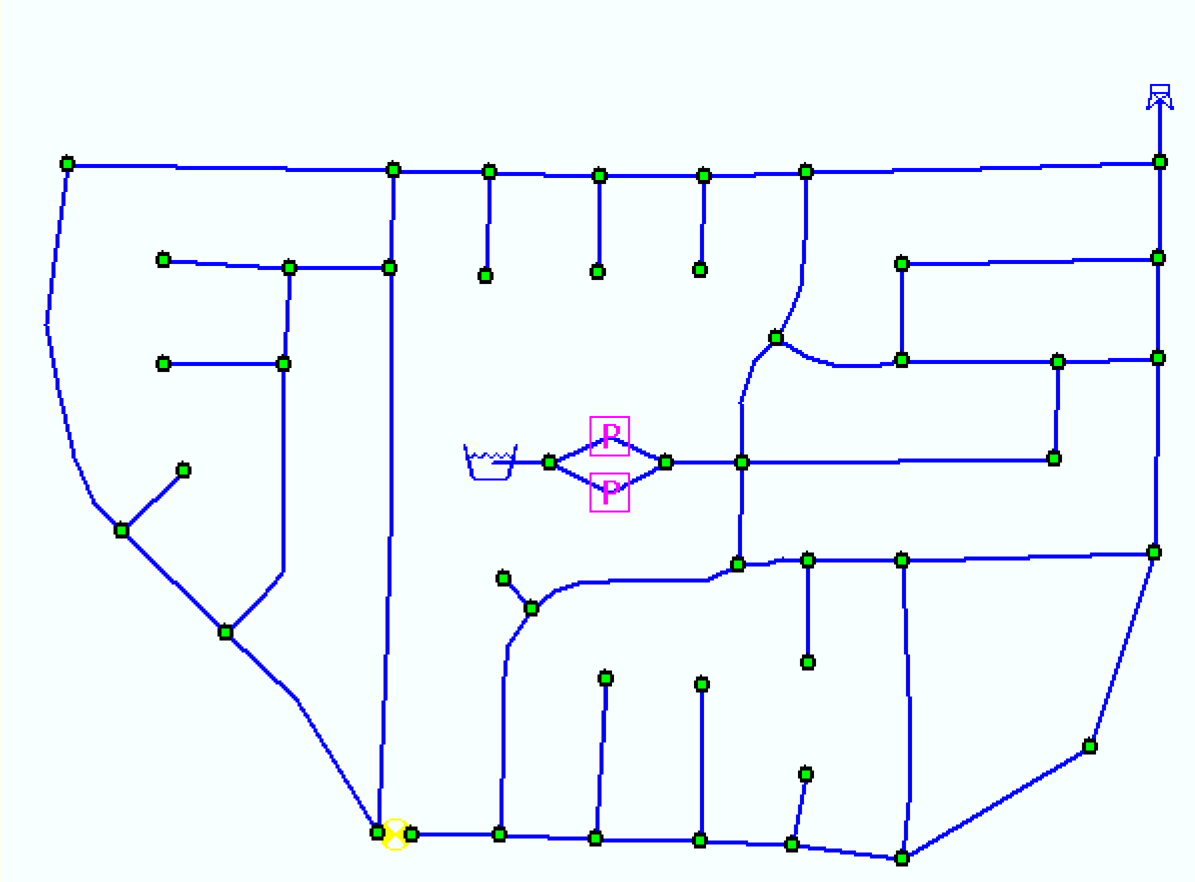
The InfoWater user interface has been created in such a fashion as to facilitate model creation and review output results. To view each component of the user interface, please select from any of the following:

- Drawing Area
- ArcMap pull-down menus
- ArcMap Toolbars
- InfoWater Toolbars & Icons
- InfoWater Control Center
- Table of Contents
- Attribute Browser

Drawing Area

The InfoWater drawing area displays the actual network schematic. You may use a combination of InfoWater and ArcMap commands to create, modify, edit and maintain your InfoWater projects. You may bring in Base maps and/or other external drawings into ArcMap and use it to create your distribution system.

Any of the InfoWater elements such as pipes, pumps, valves, junctions, tanks and reservoirs may be created in the drawing area using the InfoWater Create menu commands. Data may be associated with each of the elements in this drawing. These values will be stored in the InfoWater database and may be edited at any time. Each element has a different symbol to identify them with. For any of the different available InfoWater commands refer the InfoWater help file.



Other Related Topics - ArcMap Pull-Down Menus, ArcMap Toolbars

ArcMap Pull-Down Menu

The ArcMap Pull-Down menu also referred to as the Main Menu provides different features to create, maintain and customize your ArcMap projects. Since InfoWater runs within the ArcMap framework, you may use any of the ArcMap commands available to customize your InfoWater project.

Use the **Open** and **New** commands from the **File** pull down menus to open an existing InfoWater project or to create a new one. In both cases you would need to initialize InfoWater to access the different InfoWater commands provided through the different InfoWater toolbars.

For more information on the ArcMap pull down menus refer to the ArcMap help file.

Other Related Topics - ArcMap Toolbars, Drawing Area

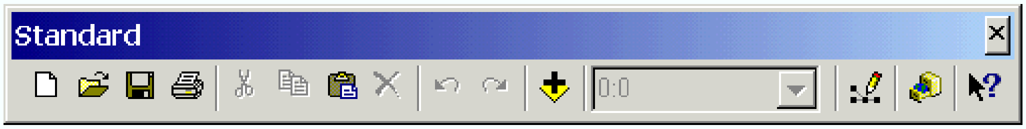
ArcMap Toolbars

InfoWater is developed as a customized version of the ArcMap graphical user interface. Individual or logically grouped modeling and model data management functions are available as commands within ArcMap. The InfoWater and the ArcMap toolbars are bundled together and may be accessed from the **View -> Toolbars** command. Additionally you may right click on any section of your ArcMap/InfoWater user interface other than the Drawing Area to invoke any of the toolbars. Also you may use the Customize tool bar to choose the different tool bars that you want to invoke. The activated tool bars may be docked on any part of your graphical user interface.

For more information on the ArcMap tool bars refer to your ArcMap help file. Detailed instructions on the usage and the capabilities of the InfoWater toolbars are provided in the InfoWater toolbars section.

The following are the different tool bars provided by ArcMap.

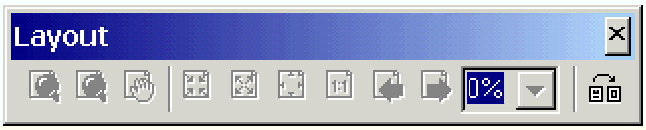
Standard



Tools

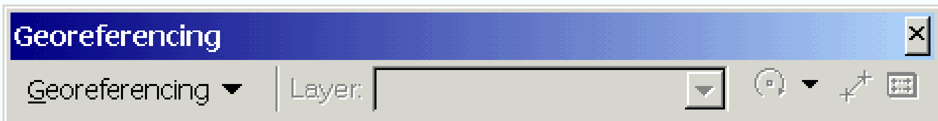
Draw

Layout

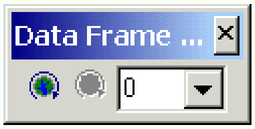


Effects

Georeferencing



Data Frame Tools



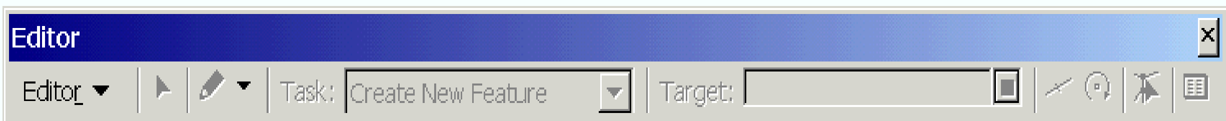
Utility Network Analyst

Spatial Adjustment

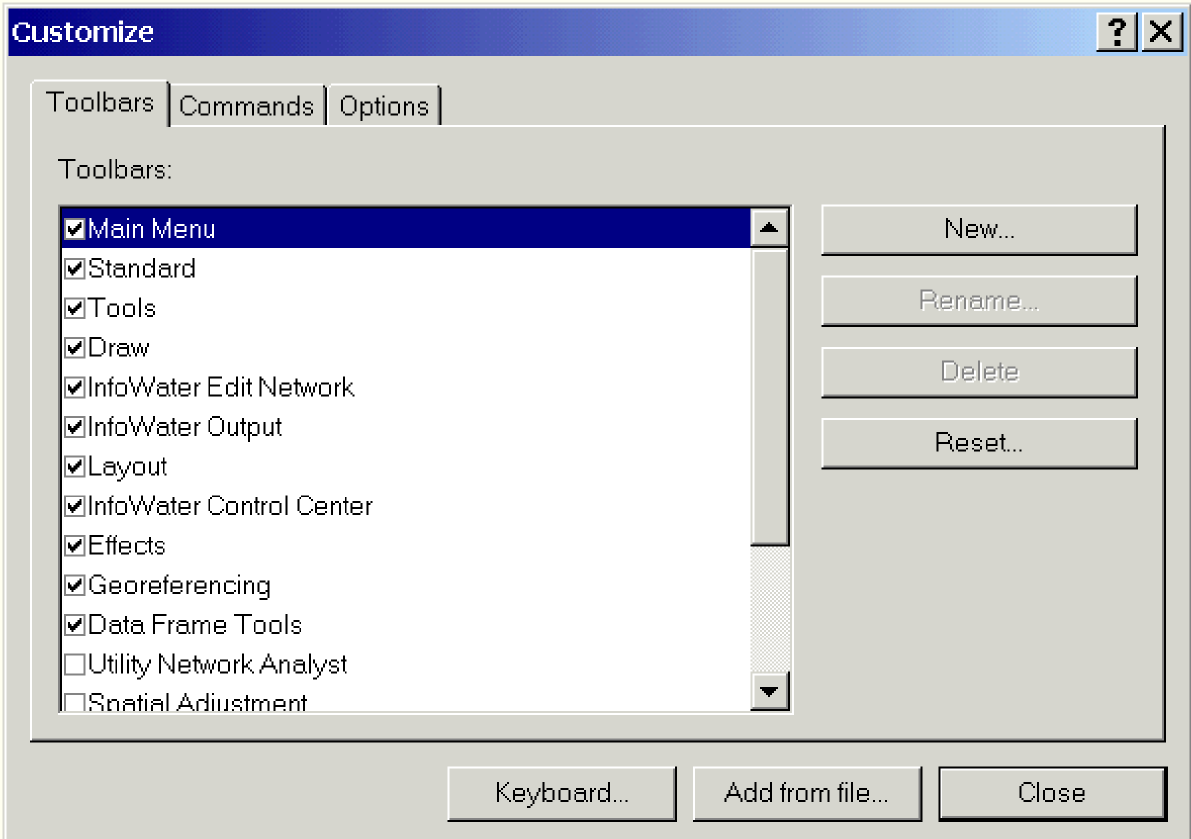
Graphics

Edit Cache

Editor



Customize



View Source

Other Related Topics - ArcMap Pull-Down Menus, Drawing Area

InfoWater Toolbars & Icons

InfoWater provides the following toolbars and icons. These icons provide access to many of the InfoWater Commands. All these commands found here may be accessed through the InfoWater button. A lot of them may also be found on the different sections of the Attribute Browser and the InfoWater Table of Contents.

Click on any toolbar to learn more about the associated icons.

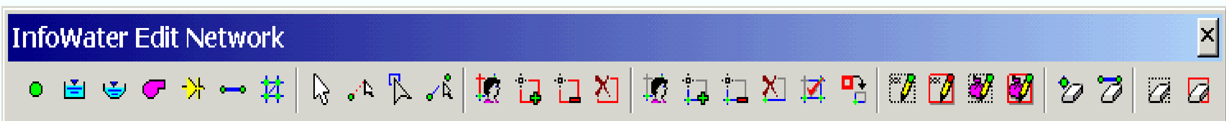
Control Center

The **InfoWater Control Center** is a collection of shortcut buttons to access commonly-used InfoWater commands. The Control Center may be invoked from the Toolbars command under the View menu or by right clicking on any part other that the Drawing Area of the InfoWater user interface. Click here to learn more about the InfoWater Control Center.

[InfoWater Edit network]

The **InfoWater Edit Network** toolbar provides various features to create data elements (such as Junctions, Tanks, Reservoirs, Pumps, Valves, Pipes and Network), select specific elements for data edit. Also graphical edit commands such as Move Node, Edit Vertices, Redraw Pipes may be accessed. Domain Manager and Facility Manager may also be accessed from here. Also delete commands are available here.

Click on any section below to learn more:



InfoWater Output

The **InfoWater Output** toolbar provides a short cut to the Run Manager and the Output Source. You can also change the time step of display.

Click on any section below to learn more:

Control Center Overview

The InfoWater Control Center is a collection of shortcut buttons to access commonly-used InfoWater commands. The Control Center may be invoked from the **Toolbars** command under the **View** menu or by right clicking on any part other that the Drawing Area of the InfoWater user interface.

You may move the Control Center to any location on the screen. Additionally, you may "dock" the Control Center to the top or sides of the InfoWater display window. To do so, place the cursor in the background area (not on a button) in the Control Center, hold the cursor down, and drag the Control Center to the desired location.

Click on any portion of the InfoWater Control Center and it will launch the appropriate InfoWater dialog box or will give you more insight into the command.

Click on any section below to learn more.

InfoWater Button

All the InfoWater commands may be accessed through the InfoWater menu bars.

Click on any section below for more information.

Additionally InfoWater commands may be accessed through the different InfoWater toolbars including the InfoWater Control Center, InfoWater Edit

Network and the InfoWater Output toolbars, the Attribute Browser

and from the Table of Contents.

Table of Contents Overview

The InfoWater Table of Contents provides a centralized location to access the various tools to create, edit, operate and analyze InfoWater models. The Table of Contents dialog box can be invoked by using the **Table of Contents** command from the **View** menu.

The InfoWater Table of Contents provides five major capabilities for the modeling and analysis of water distribution systems:

Display: The display tab displays all the different layers available in your InfoWater project. This is a standard tool provided by ArcMap. To learn more about the different uses of the Display tab refer to the ArcMap helpfile.

Source: The source tab also allows you to modify the different layers available in your InfoWater project. To learn more about the Source tab refer to the corresponding section in the InfoWater helpfile.

Contour: Use the contour tab to plot nodal input and output contours. To learn more about the different contouring options provided by InfoWater click here.

Operation Data: The Operation Data tab contains all of the operational features of InfoWater. Click here to learn more about the Operation Data feature of InfoWater.

Annotation: The Annotation tab is used to color code pipes or nodes based on input and/or output results. Click here to learn more about the InfoWater Annotation Manager.

Click on any section below to learn more:

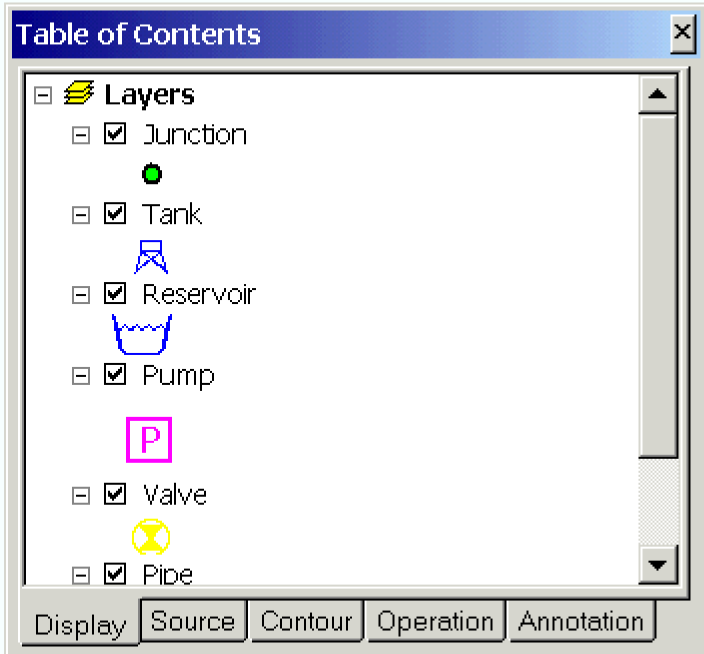
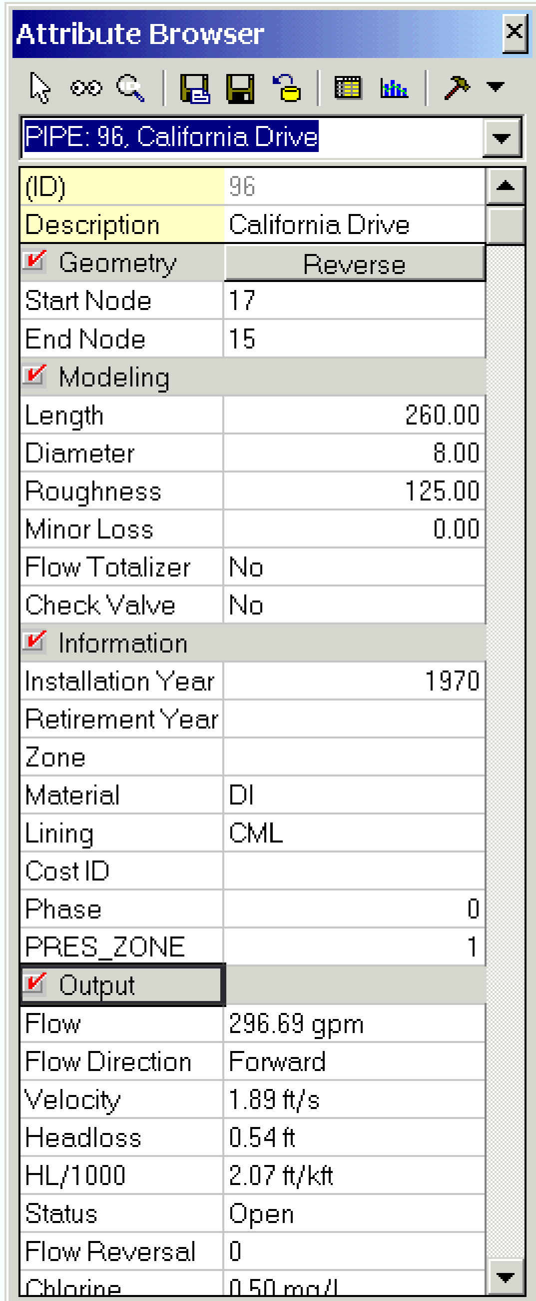


Table of Contents
Layers
Junction
Tank
Reservoir
Pump
P
Valve
Pipe
Display
Source
Contour
Operation
Annotation

Attribute Browser

The **Attribute Browser** provides you with tools to view, modify and alter data pertaining to any one InfoWater element such as a pipe, junction, valve, pump, tank and/or a reservoir. It provides also a quick means to inspect output data relating to that element. The Output data section is available only after a successful model run. Additionally other features allow you to locate (search) elements, create default values and assign specific values through the Tools menu.

Click on any section below to learn more:



Other Related Topics - Tools menu

Preferences

The Preferences command is used to control the InfoWater system settings. All changes made on the Preferences dialog box must be applied prior to closing the dialog box. Many preferences set with this command will be reflected as the default choices on other dialog boxes. You may change those settings as desired on those dialog boxes.

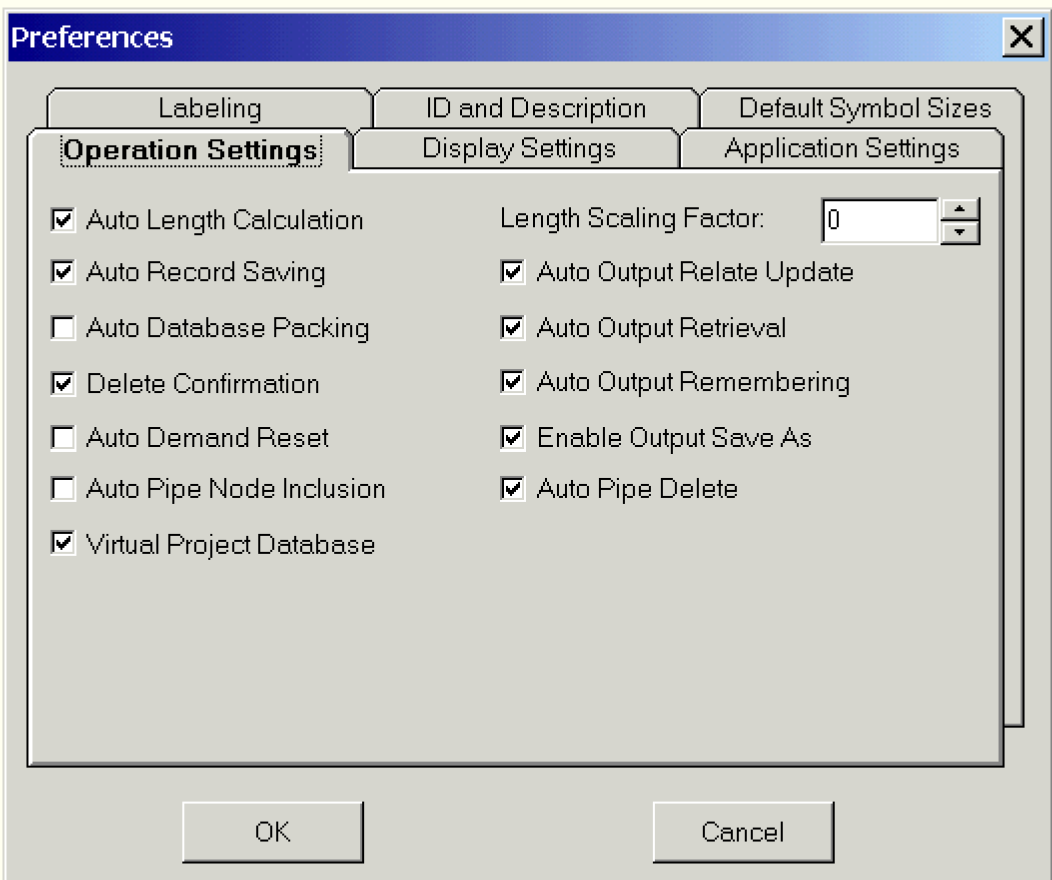
To display Preferences, from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu select **Project Preferences**. The following different options are available from the Preferences dialog box.

- Operation Settings - Specify the different operation settings here.
- Display Settings - Use this to specify the display properties.
- ID and Description - ID and Descriptions may be set here.
- Labelling - Specify your Map label preferences here.
- Application Settings - All application related data may be specified here.
- Default Symbol Sizes - Specify the default symbol sizes for all the InfoWater data elements here.

Operation Settings

The Operation tab allows the user to control project settings options relating to the way data is controlled in InfoWater. All changes made on the Preferences dialog box must be saved prior to closing the dialog box. Many preferences set with this command will be reflected as the default choices on other dialog boxes. You may change those settings as desired on those dialog boxes.

To display Operation Settings, from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Project Preferences -> Operation Settings**. Click on any portion to learn more.

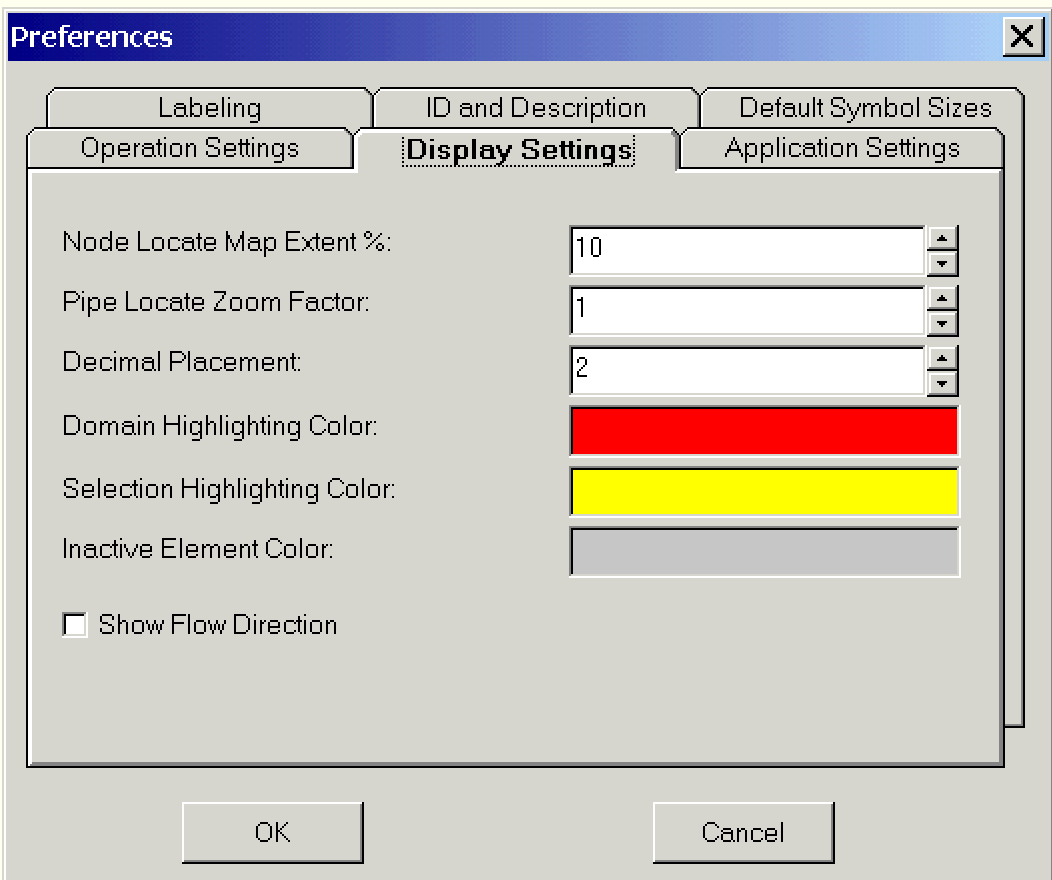


Other Related Topics - Preferences, Preferences - Application Settings tab, Preferences - Default Symbol Sizes tab, Preferences - Display Properties tab, Preferences - ID and Description tab, Preferences - Map Settings tab

Display Settings

The Display tab controls visual effects and features within InfoWater. All changes made on the Preferences dialog box must be saved prior to closing the dialog box. Many preferences set with this command will be reflected as the default choices on other dialog boxes. You may change those settings as desired on those dialog boxes.

To choose Display Settings dialog box, from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Project Preferences -> Display Settings**. Click on any portion to learn more.

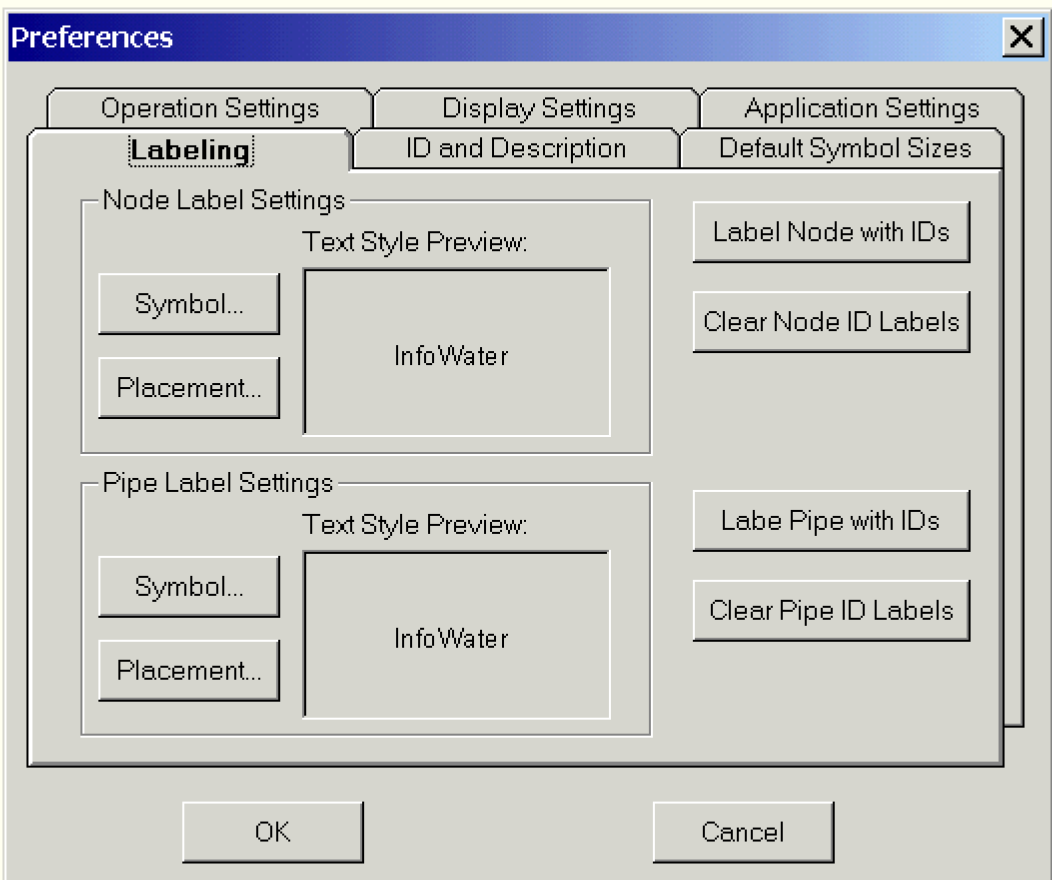


Other Related Topics - Preferences, Preferences - Application Settings tab, Preferences - Default Symbol Sizes tab, Preferences - ID and Description tab, Preferences - Map Settings tab, Preferences - Operation Settings tab

Labelling

This section of the Preferences dialog box provides you with tools to customize your label settings. All changes made on the Preferences dialog box must be saved prior to closing the dialog box. Many preferences set with this command will be reflected as the default choices on other dialog boxes. You may change those settings as desired on those dialog boxes.

To display the Labelling tab, from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Project Preferences -> Labelling**. Click on any portion below to learn more.

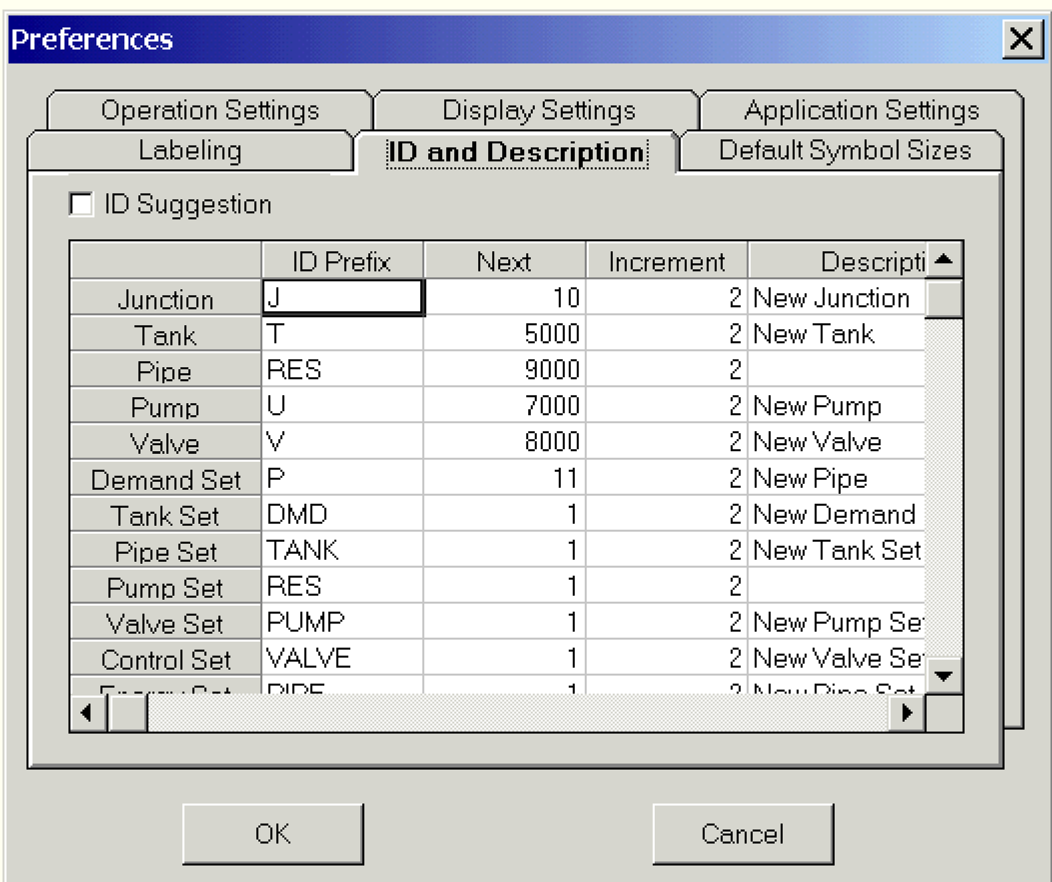


Other Related Topics - Preferences, Preferences - Application Settings tab, Preferences - Default Symbol Sizes tab, Preferences - Display Properties tab, Preferences - ID and Description tab, Preferences - Operation Settings tab

ID and Description

The ID and Description tab allows the user to change InfoWater defaults for data element labeling, data sets, curves, patterns, etc. All changes made on the Preferences dialog box must be saved prior to closing the dialog box. Many preferences set with this command will be reflected as the default choices on other dialog boxes. You may change those settings as desired on those dialog boxes.

To display the ID and Description dialog box, from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Project Preferences -> ID and Description**. Click on any portion below to learn more.

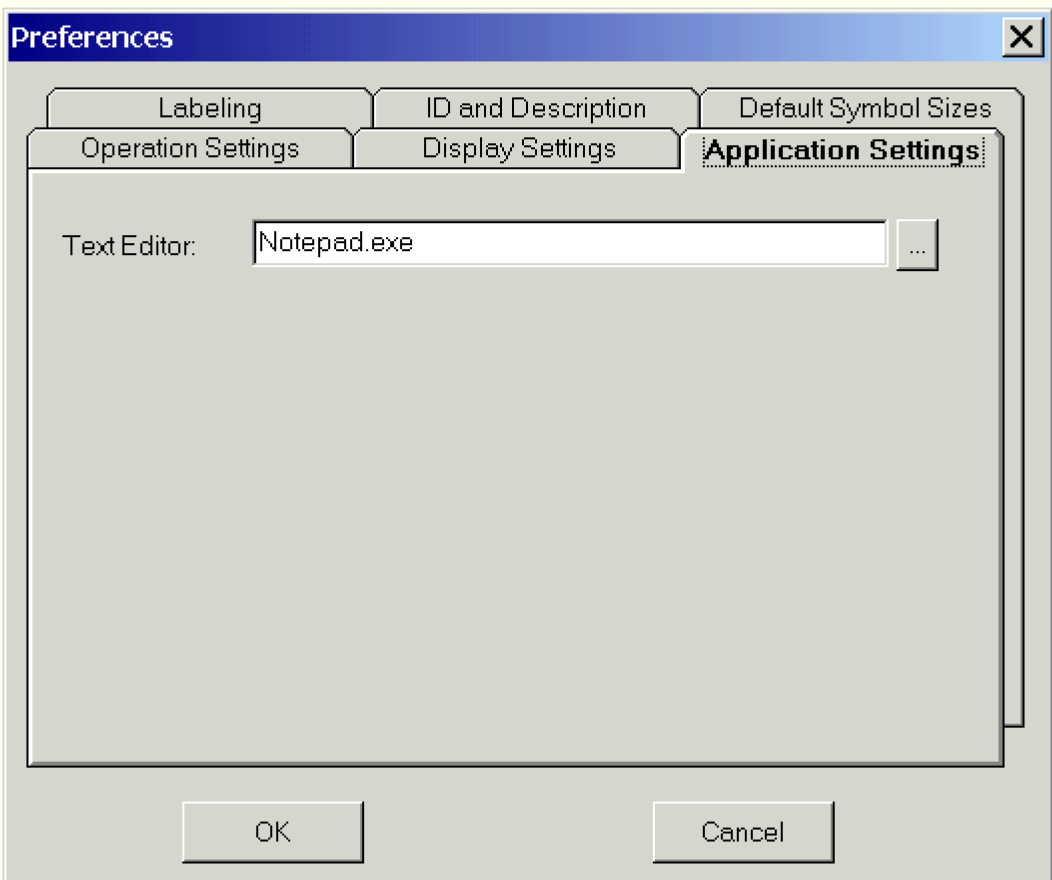


Other Related Topics - Preferences, Preferences - Application Settings tab, Preferences - Default Symbol Sizes tab, Preferences - Display Properties tab, Preferences - Map Settings tab, Preferences - Operation Settings tab

Application Settings

The Preferences - Application Settings command is used to control the InfoWater Application settings. All changes made on the Preferences dialog box must be saved (by clicking the **OK** button) prior to closing the dialog box. Many preferences set with this command will be reflected as the default choices on other dialog boxes. You may change those settings as desired on those dialog boxes.

To display Application Settings, from the **InfoWater Control Center -> InfoWater** button **-> Tools** sub-menu, select **Project Preferences -> Application Settings**. Click on any portion below to learn more.

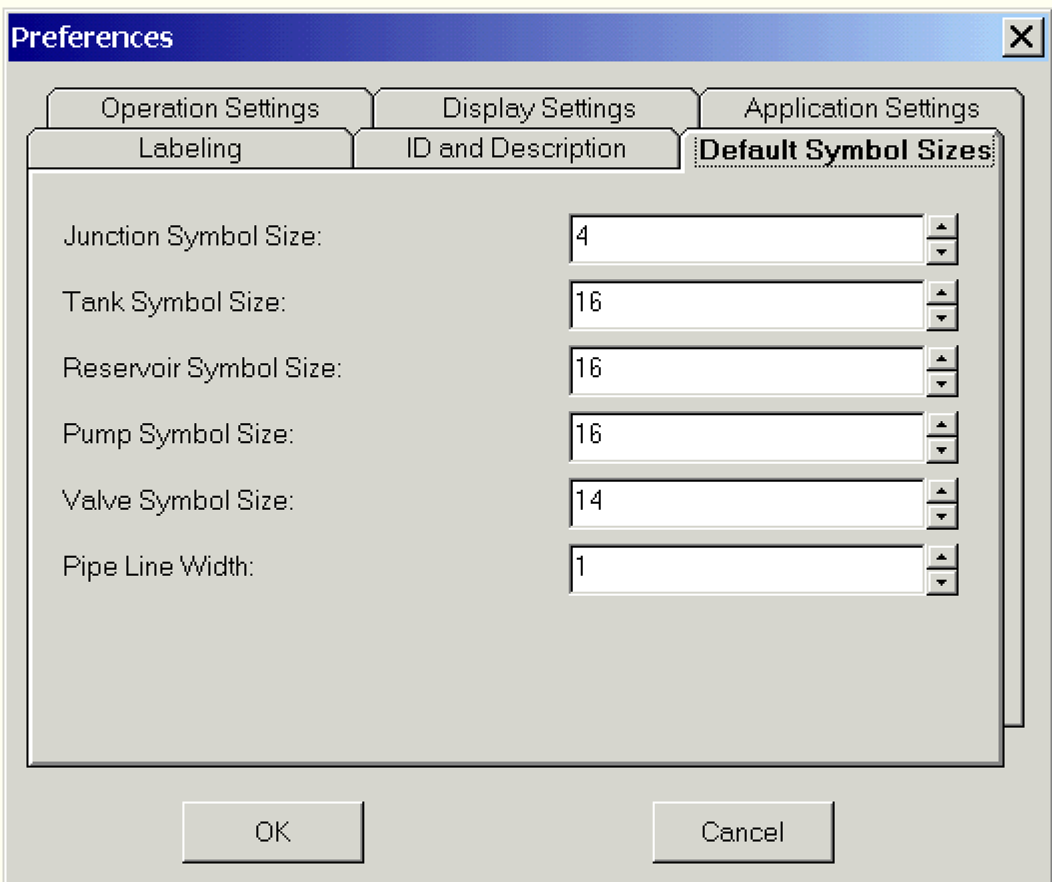


Other Related Topics - Preferences**,** Preferences - Default Symbol Sizes tab**,** Preferences - Display Properties tab**,** Preferences - ID and Description tab**,** Preferences - Map Settings tab**,** Preferences - Operation Settings tab

Default Symbol Sizes

The Default Symbols Size tab is used to adjust the size (graphical appearance) of the InfoWater data elements. All changes made on the Preferences dialog box must be saved prior to closing the dialog box. Many preferences set with this command will be reflected as the default choices on other dialog boxes. You may change those settings as desired on those dialog boxes.

To display Default Symbol Sizes, from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Project Preferences -> Default Symbol Sizes**. Click on any portion to learn more.



Other Related Topics - Preferences, Preferences - Application Settings tab, Preferences - Display Properties tab, Preferences - ID and Description tab, Preferences - Map Settings tab, Preferences - Operation Settings tab

Running a Model

Once all data elements have been located and their databases populated, a hydraulic analysis (also known as Running a Model) can be performed on the system. Click here for more information about the InfoWater Data Elements.

A model may be run by using one of the two following methods:

Run Manager :

The Run Manager dialog box may be accessed from the **InfoWater Control Center -> InfoWater** button **-> Tools** pull down menu or from the **InfoWater Output** toolbar by clicking on the Run Manager icon .

The Run Manager allows the following hydraulic analyses to be performed:

- Standard - A standard hydraulic simulation that evaluates flows, velocities, pressures, etc. Two types of Standard simulations can be conducted using InfoWater:

    Steady State Modeling

    Extended Period Simulation (Water Quality Modeling & Energy Management Modeling)

- Fireflow - A standard hydraulic simulation (based on the current active scenario) that includes fireflow demands as assigned by the user.
- SCADA - A standard hydraulic simulation (based on the current active scenario) that adjusts system demand/flows and imports a SCADA file for comparison.
- Hydrant Curve - Creates a hydrant curve (based on the current active scenario) for the selected junction node.
- System Curve - Creates a system curve (based on the current active scenario) for the selected junction node.
- Pressure Dependent Demand - A hydraulic simulation that allows the user to model variations in demand with respect to changes in system pressure.

- VSP/FPP Analysis - Use this to determine the pump speed required to maintain a target downstream pressure for a variable speed pump. Perform prior to running a standard hydraulic simulation.

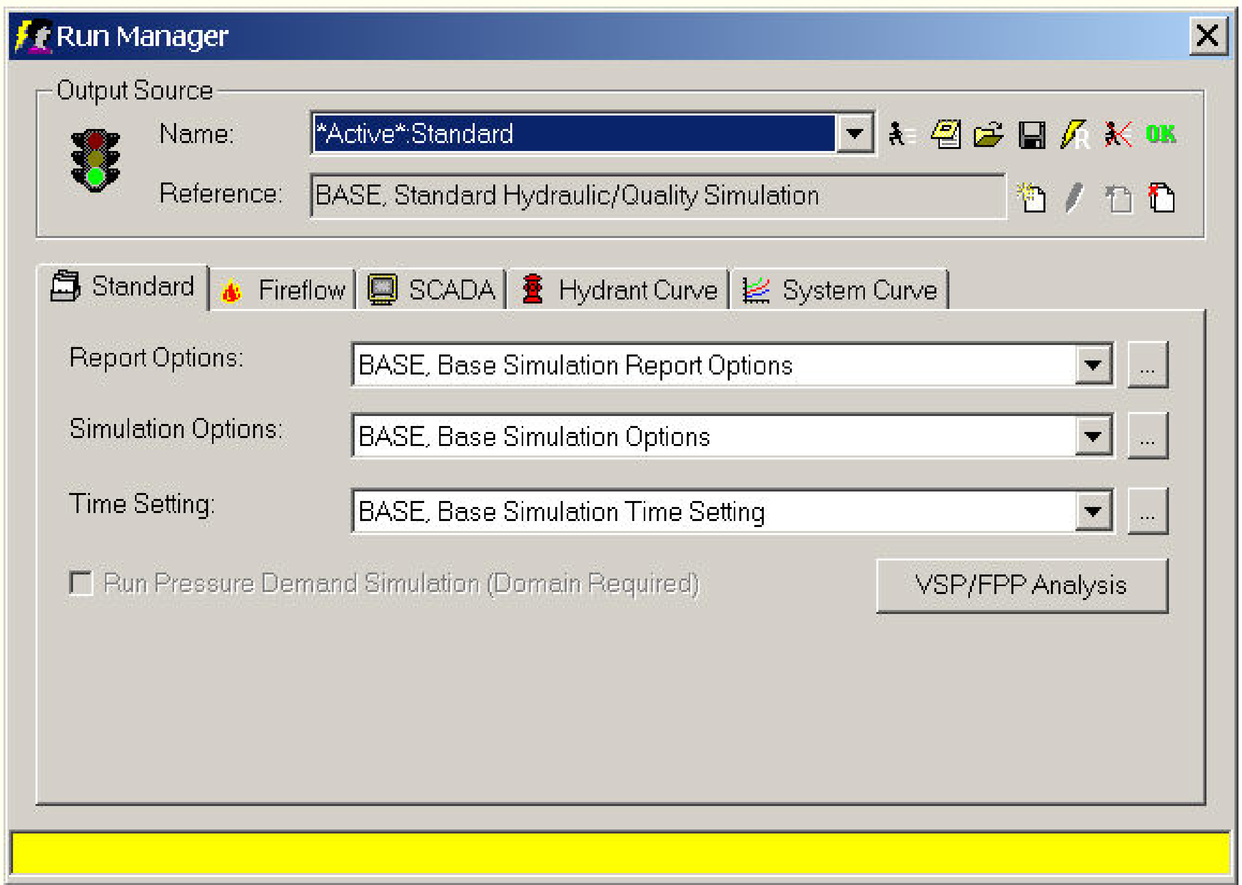
Batch Simulation:

To run a batch process (standard simulations only), use the Batch Simulation Manager.

Run Manager

The Run Manager is used to perform simulations and to manage simulation output sources (results). Five individual simulation types are available from the Run Manager; Standard simulations (Hydraulics and Water Quality), Fireflow simulations, SCADA, Hydrant Curves, and System head curves. You may also use the Batch Run Manager to run multiple standard simulations with one operation.

To activate the Run Manager, from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Run Manager**. Click on any portion of the Run Manager dialog box for more information.



A Standard Simulation is the most frequently run model and it represents the "hydraulic" model.

Specify the Report Options, Simulation Options and the Time Setting for the simulation run "on the fly". These options can also be accessed through the tool bar icons and the InfoWater Edit pull down menu.

After a model is run, the most recent simulation run results are referred to as the *active* output source. The icons on the Run Manager allow the user to run simulations and create new *active* output sources, save output sources for later use, or load output sources for visualization, analysis, and comparison with other output sources. Use the Output Report Manager to see simulation results in either report or graph formats.

When you run a simulation, InfoWater uses modeling data associated with the currently active scenario. If you have not developed a custom scenario, the *active* scenario is the “BASE”. To run a model based on a different scenario, activate the desired scenario using the Active Scenario command and then use the Run Manager to perform the simulation for with the selected scenario.

Other Related Topics - Running a Model, Run Manager Methodology, Batch Run Manager, Batch Run Methodology

---

Steady State

Many features of InfoWater including SCADA, Energy Management, Water Quality, and Logic Rules are dependent upon a model being analyzed over a series of timesteps (Extended Period Simulation). A steady state model does not involve timesteps. It is a model run that encompasses a single "snap-shot" of the water system as it pertains to demands, pumping data and tank elevations.

As a result, steady state simulations do not allow the user to see the bigger picture of what may, or may not, be occurring in a distribution system. For example, tanks cycling, water age, energy consumption, etc. While a steady state simulation does not provide this information, for some water systems (particularly closed systems with non-fluctuating demand), a steady state simulation is more than adequate.

Methodology

Follow the steps below to run a Steady State Model:

- Create your model and populate the Databases with all the element information.
- Specify Steady State - From the **Table of Contents -> Operation** tab choose **Simulation Time**. Once there, double click the BASE simulation time. Click on the Steady State option to place a check in the box. Once this is done, close the dialog box by clicking the OK button.
- **Running a Steady State Model -** Select **Run Manager** from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu or by clicking on the Run Manager icon from the **InfoWater Output** toolbar to run the steady state model.
- Specify the Report Options and the Simulation Options "on the fly". Verify the Time settings options as Steady State by clicking on the Browse icon
- Click on the Run icon in your Run Manager dialog box.
- Your Model has now been run. Verify your results by going to **InfoWater Control Center -> InfoWater** button **-> Tools** menu -> **Output Report Manager**.

Other Related Topics - Running a Model**,** Specify Steady State Run

---

EPS Modeling

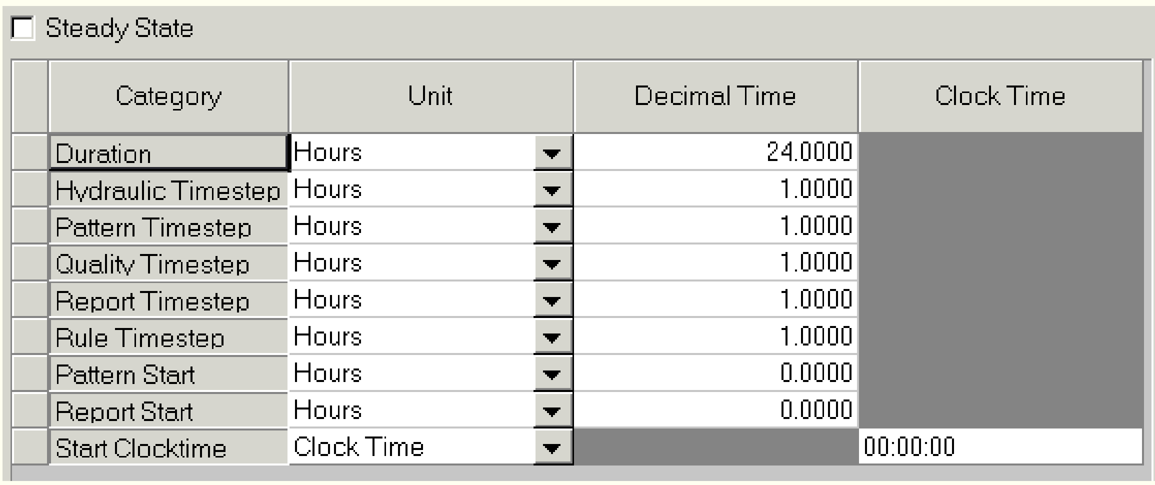
An extended period simulation (EPS) is any modeling run that evaluates system hydraulics for a duration greater than a single hydraulic timestep (steady state). The user is provided many tools in InfoWater that allow customization of an EPS simulation.

Many features of InfoWater including SCADA, Energy Management, Water Quality, and Logic Rules are dependent upon a model being analyzed over a series of timesteps (Extended Period Simulation).

Methodology

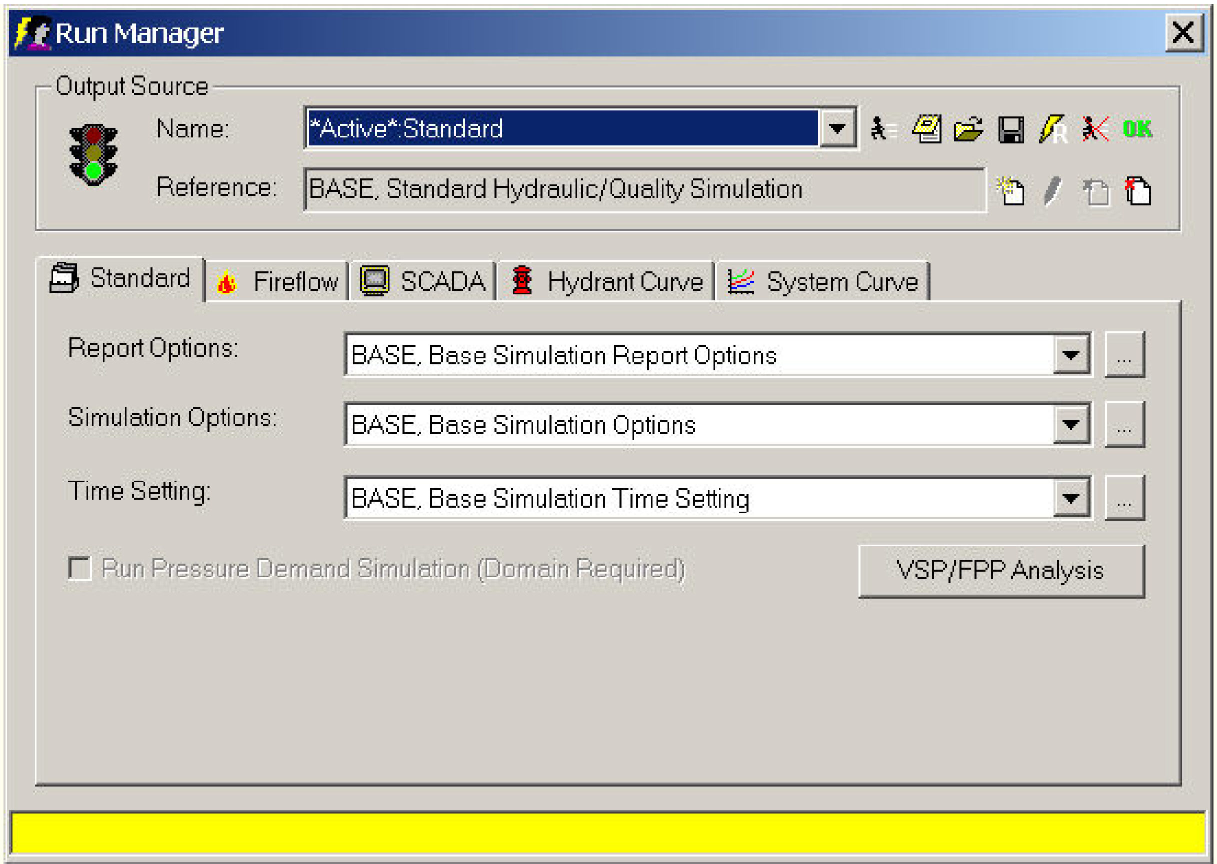
To run an extended period simulation do the following:

- From the InfoWater **Table of Contents -> Operation** tab choose **Simulation Time**. Once there, double click the **BASE** simulation time to see the dialog box below. Uncheck the **Steady State** option and specify the different Run options. Click any section below to learn more.



Category	Unit	Decimal Time	Clock Time
Duration	Hours	24.0000	
Hydraulic Timestep	Hours	1.0000	
Pattern Timestep	Hours	1.0000	
Quality Timestep	Hours	1.0000	
Report Timestep	Hours	1.0000	
Rule Timestep	Hours	1.0000	
Pattern Start	Hours	0.0000	
Report Start	Hours	0.0000	
Start Clocktime	Clock Time		00:00:00

- Select **Run Manager** from the **InfoWater Control Center -> InfoWater** button -> **Tools** menu or by clicking on the **Run Manager** icon  from the InfoWater **Output** toolbar to run the EPS model. Click on any section below for more information.



- Specify/alter or customize the **Report Options** and the **Simulation Options** "on the fly" from the **Run Manager** dialog box. Verify the **Time Settings** options as Extended Period by clicking on the **Browse** icon .
- Click on the **Run** icon  in the **Run Manager** dialog box to run your EPS model.
- After a model is run, the most recent simulation run results are referred to as the *active* output source. The icons on the Run Manager allow the user to run simulations and create new *active* output sources, save output sources for later use, or load output sources for visualization, analysis, and comparison with other output sources. Use the Output Report Manager to see simulation results in either report or graph formats.

- When you run a simulation, InfoWater uses modeling data associated with the currently active scenario. If you have not developed a custom scenario, the *active* scenario is the “BASE”. To run a model based on a different scenario, activate the desired scenario using the Active Scenario command and then use the Run Manager to perform the simulation for with the selected scenario.

Other Related Topics - Running a Model

Water Quality

With InfoWater, you can perform several types of water quality analyses in conjunction with a hydraulic simulation. InfoWater supports the following types of water quality analyses. These options may be chosen on the Quality panel on the Simulation Options dialog box. Click on any link to learn more.

- *Chemical propagation*
- *Water age*
- *Source tracing*

Click here to learn more about water quality simulations and running a water quality model.

Other Related Topics - Running a Model**,** Water Quality Simulation Methodology

---

Energy Management Modeling

InfoWater offers comprehensive energy management capabilities to assist in minimizing energy consumption and identifying optimal pump scheduling and operating policies. Since the only device imparting energy to a water system is a pump, all energy management features are pump specific.

- Properties of Energy Management
- Data Required for an Energy Analysis
- Assigning Energy Data
- Running an Energy Simulation

Properties of InfoWater Energy Management Analysis

Several important properties of the InfoWater energy management tools include:

- *Energy Cost Analysis for User-Specified Pumps*
- *Running an Energy Management Simulation*
- *Energy Management Simulation Results*
- *Alternatives Management*

Note: A properly calibrated hydraulic model of the water distribution system is required for pump energy management applications. In addition, an **EPS** must be specified to perform an energy management simulation. To enable an EPS model, go to the **Table of Contents -> Operation** tab **-> Simulation Time** folder. Double click on the **Simulation Time** folder and once again double click on the

Base Simulation Time Settings option (or the appropriate Simulation Time set) By specifying a Simulation Time that has the "Steady State" check box unchecked, the user is ensuring that an EPS model is run.

Required Data

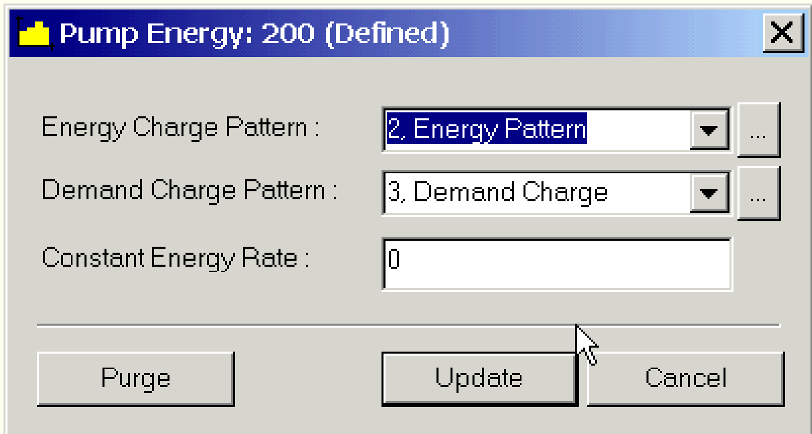
The energy management module calculates all essential electricity costs for pumping stations. It handles time-varying energy rates and demand charges for both constant efficiency and variable efficiency (efficiency vs. flow data) pumps in the system. To perform an energy management analysis, the following data is needed for system pumps:

Patterns - Two types of patterns are used to associate energy management conditions to a model.

- *Energy Charge Pattern* is defined as the charging rate per unit of energy usage ($/kWh) over time (EPS). Click here to see an energy price pattern.
- *Demand (or capacity) Charge Pattern* is defined as the cost associated with the maximum power consumed within the charging (billing) period ($/max. kW). (This is a one-time charge due to peak power used during each billing period. If you do not specify this information, InfoWater will not include any charge for maximum power consumption.) Click here to see an demand charge pattern.

Energy Pattern

On the **Attribute Browser** window select the **Tools** icon and choose **Energy Pattern** to launch the Pattern dialog box as shown below. Specify the Energy Charge pattern, the Demand Charge pattern and/or the Energy Rate. Click on Update to save and exit from the dialog box.



Curves - A pump efficiency curve is used to describe the relationship between the overall pump efficiency as a function of flow - expressed as a percentage. The overall pump efficiency is the wire-to-water (total) efficiency which is the product of the pump efficiency and driver efficiency. Click here to see a pump efficiency curve. See Energy Efficiency below to learn more about pump efficiency

---

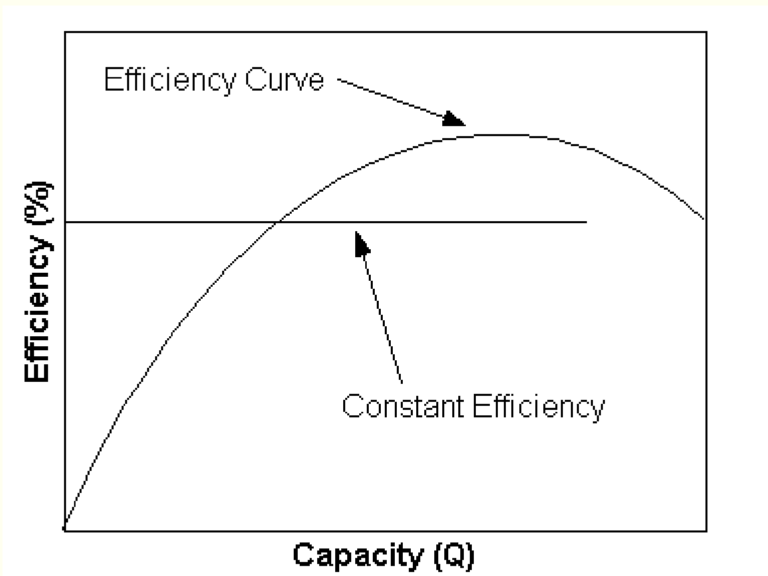
Note: The various speed settings for variable speed pumps are user-specified. Also, both constant and variable speed pumps can be modeled.

Energy Efficiency

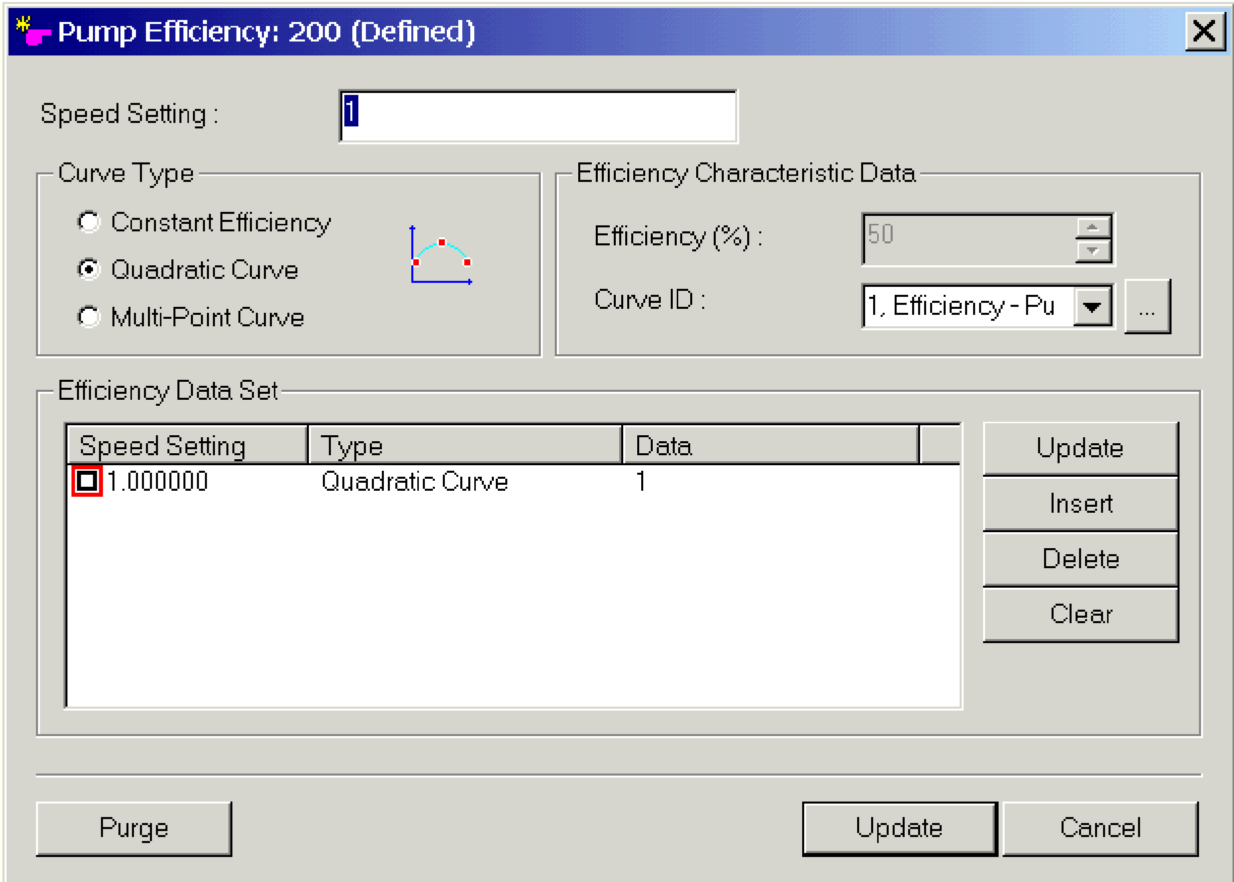
InfoWater provides two options for representing pump efficiency:

- *Constant Efficiency*
- *Variable Efficiency Curve*

The figure below illustrates constant efficiency and a variable efficiency curves for a given pump.



From the **Attribute Browser** window select the **Tools** icon  and choose **Energy Efficiency** to launch the Efficiency dialog box as shown below. Click on any section for more information.



Assigning Energy Data

From the InfoWater **Table of Contents -> Operation** tab, select the **Simulation Options** folder and double click on the "BASE, Base Simulation Option" to launch the **Simulation Options** dialog box. Click on the Energy tab to globally assign the same energy price (or pattern), pump efficiency and demand charge for all pumps.

Alternately each pump may be assigned unique energy and demand charge patterns, energy price and pump efficiency data via the **Attribute Browser**. Click on the following to learn how to create and apply a pattern and a curve to a pump.

Note: While it is possible to assign a demand charge pattern to an individual pump, it is not possible to assign a global demand charge pattern. Click here to learn more about energy pricing and the difference between energy pricing and demand charges.

Energy data are assigned to pumps in three steps:

1. Develop the energy charge pattern (you may optionally include a demand charge pattern).
2. Define efficiency curves.
3. Select the pump(s) to include in an energy costing simulation.
4. From the **Attribute Browser** window select the **Tools** icon and choose **Energy Pattern** or **Energy Efficiency** to launch the individual dialog boxes and assign the data.

Running an Energy Costing Simulation

Prior to running an energy costing simulation model, confirm that the **Run Energy Management Simulation** option has been enabled. To confirm this, via the Simluation Options under the **Operation** tab of the **Table of Contents**, open the simulation option that will be used for the energy analysis and select the **Energy** tab. Check the "**Run Energy Management Simulation**" check box, adjust the Energy options as necessary and close the simulation options dialog box.

To run an energy costing simulation, open the Run Manager dialog box, choose the Standard tab, set the appropriate simulation options and choose the **Run** button on the dialog box. Because the Run Energy Management Simulation option was enabled from the Simulation Options, InfoWater will perform the energy management analysis.

Upon successful completion of an energy costing simulation, open the Output Report Manager and then open either an Energy Report or the Energy Graph. The Energy Report displays one record for each pump assigned energy data prior to running the simulation.

Energy Pricing

An **Energy Price** is the price you pay for electricity. For example, if electricity is being purchased for a flat rate or $0.12/kWh, then you would assign a flat rate of $0.12 in the Energy Price box for a selected pump or, more than likely, to all pumps globally (see **Simulation Options -> Energy** tab). However, if you are on a varying rate schedule (as most users are), then you would create a pattern that represents this schedule and assign it to a single pump or to all pumps globally (again, see **Simulation Options -> Energy**). While it is unlikely that you would have different energy costs across your system, InfoWater also allows you to assign energy costs to each pump independently (ex. $0.14 for one pump and $0.18 for another).

The same logic holds true for **Demand Charges**. There are periods in the day when your water agency has agreed with your electric provider not to use power. If you do, then you will have to pay a heavy penalty (a demand charge) for using power during a penalty period. It is very unlikely that you will have a flat demand charge that could be globally assigned to all pumps (see **Simulation Options -> Energy**). More than likely, your utility will have a time varying rate schedule that they will penalize you for using power during a specific time of the day (usually afternoons in the summer time). To assign this demand charge to a pump, create a demand charge pattern that represents the actual charges in dollars as specified in your rate structure and assign that demand charge to each pump independently via the **Attribute Browser** (see section on Assigning Energy Data above). InfoWater does not allow demand charge patterns to be assigned globally.

Other Related Topics - Energy Management, Running a Model

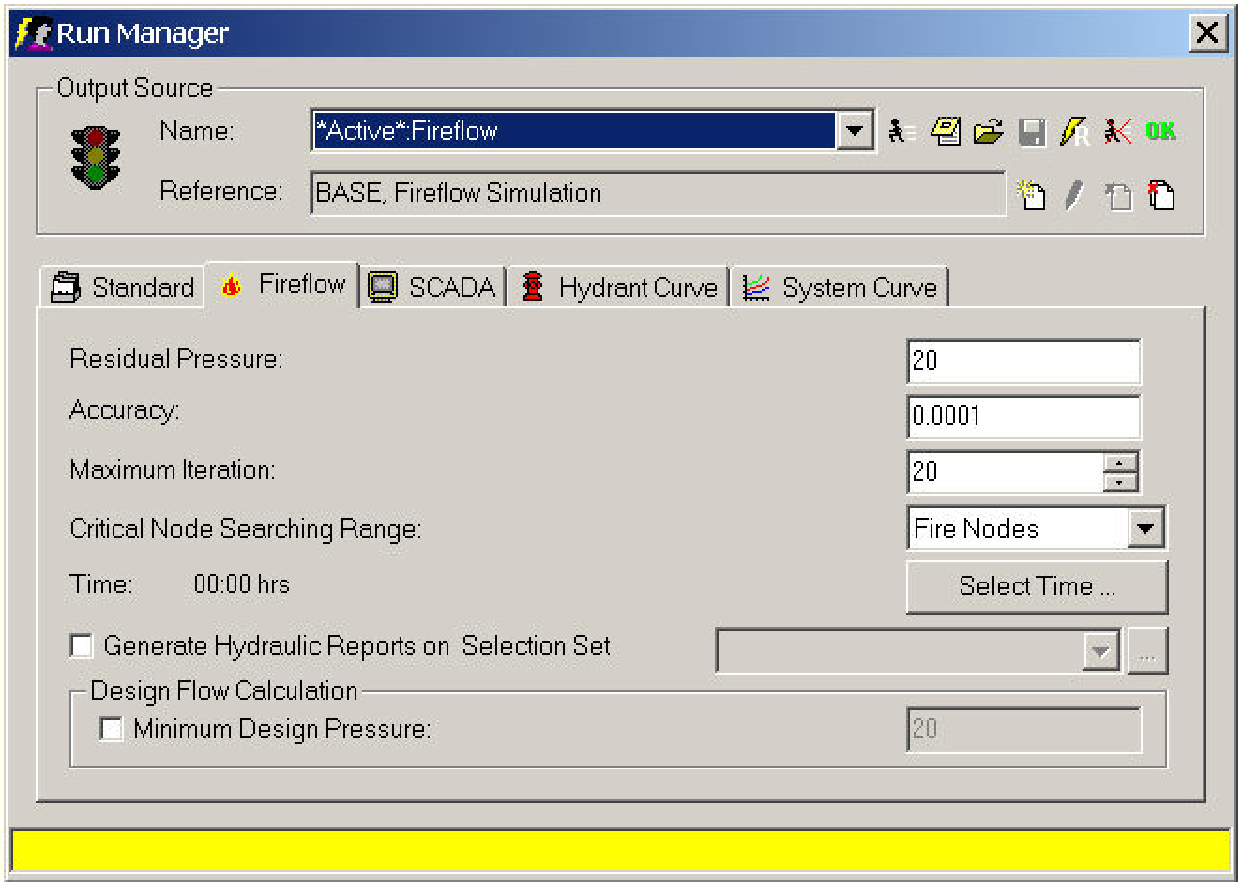
Fire Flow Modeling

Use this to conduct Fire Flow Modeling Analysis.

InfoWater provides comprehensive fire flow modeling capabilities. Fire demand requirements can be specified based on the various types of building structure (e.g., single-family residential, multi-family residential, commercial, industrial, etc.) or other fire demand characteristic.

Based on the fire flows specified, InfoWater computes residual pressures, available flows at any user-specified minimum pressure (usually 20 psi) and the design flows for each hydrant location in the distribution system. Click here to learn more about fire flow demands and how to model a fire flow simulation.

Click on any portion below to learn more.



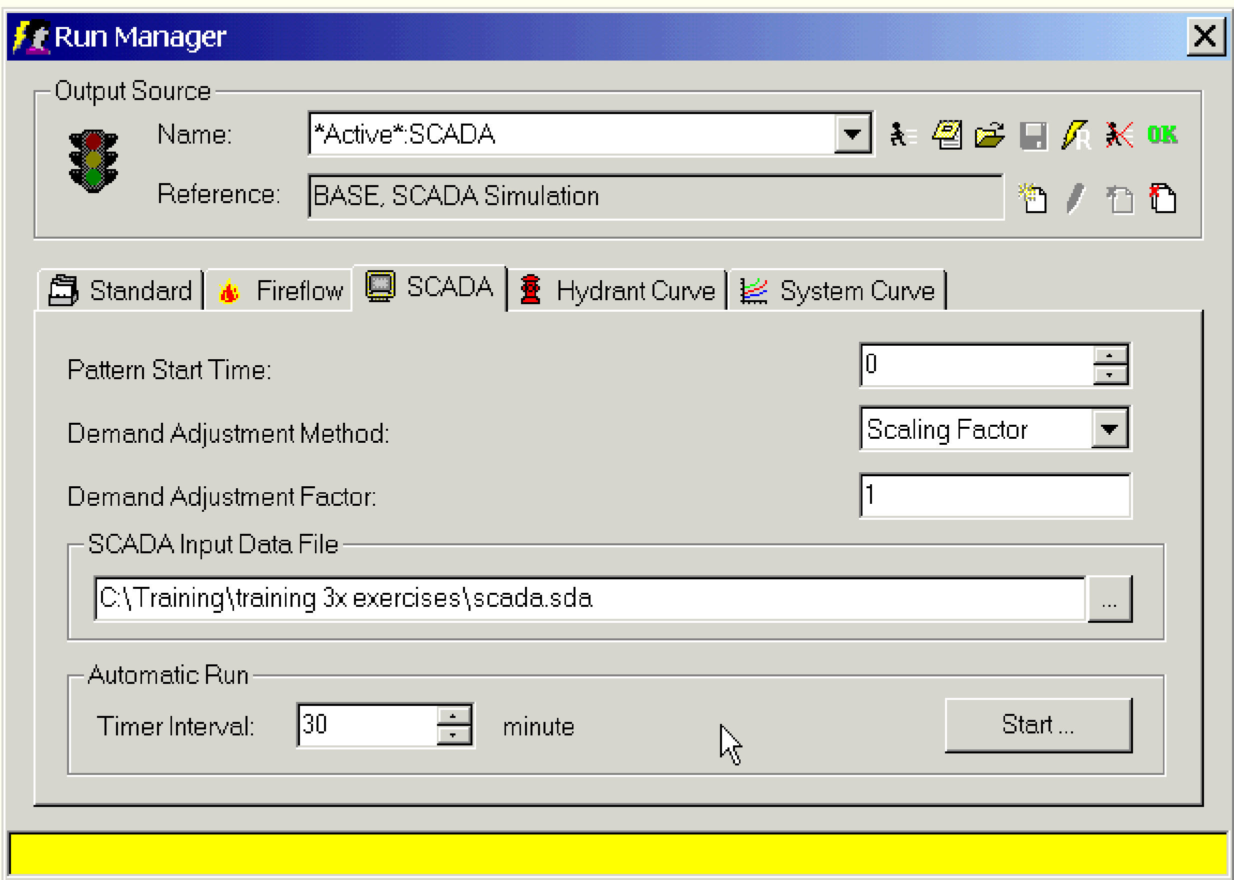
Other Related Topics - Running a Model, Fire Flow Methodology

SCADA Modeling Dialog Box

To run a SCADA simulation, from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Run Manager** to see the dialog box below. Select the SCADA tab, specify the desired parameters, and choose the Run button.

There are two ways of running a SCADA simulation. The first is to perform a One-Time simulation run of the SCADA input while the second is to set up an Automatic Run to perform a SCADA analysis on a regular schedule. To learn more about the SCADA methodology click here.

Click on any section below to learn more.



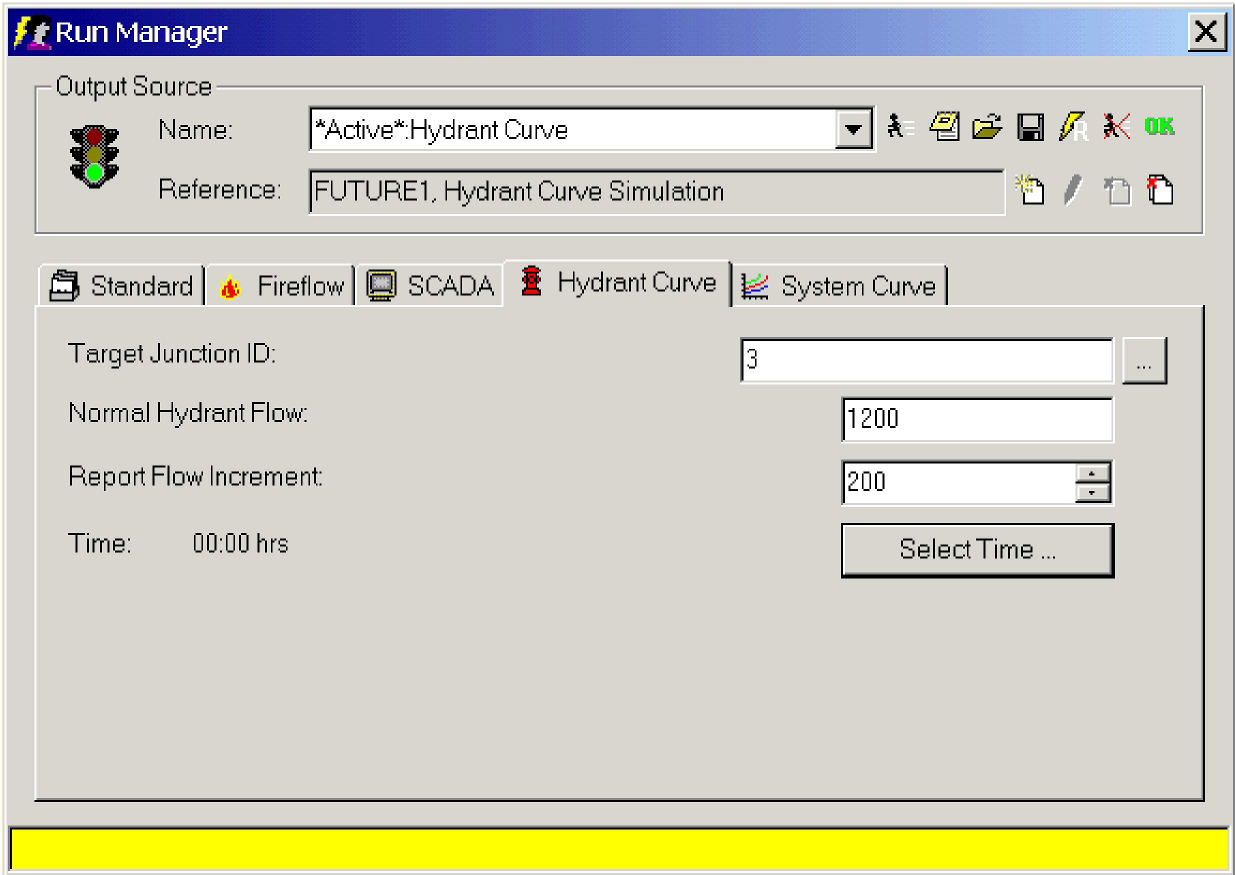
Other Related Topics - Running a Model, Real Time Data Connection Dialog Box, Real Time Data Connection Methodology, SCADA Integration, SCADA Interface, SCADA System Data Edit

Hydrant Curve Dialog Box

The fire hydrant rating curve shows the relationship between residual pressure and the available flow for any hydrant. Complete residual pressure vs. available flow calculations are directly performed ranging from the imposed static demand to the maximum available flow at atmospheric pressure, for any user-specified flow increments. Hydrant curves are often required by fire departments who are responsible for ensuring that a specific hydrant can adequately control a potential fire demand and maintain minimum pressures. To learn more about the Hydrant Curve methodology click here.

To develop a hydrant curve, from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Run Manager** to see the dialog box below. Select the **Hydrant Curve** tab, specify the desired parameters, and choose the **Run button** .

Click on any section below to learn more.



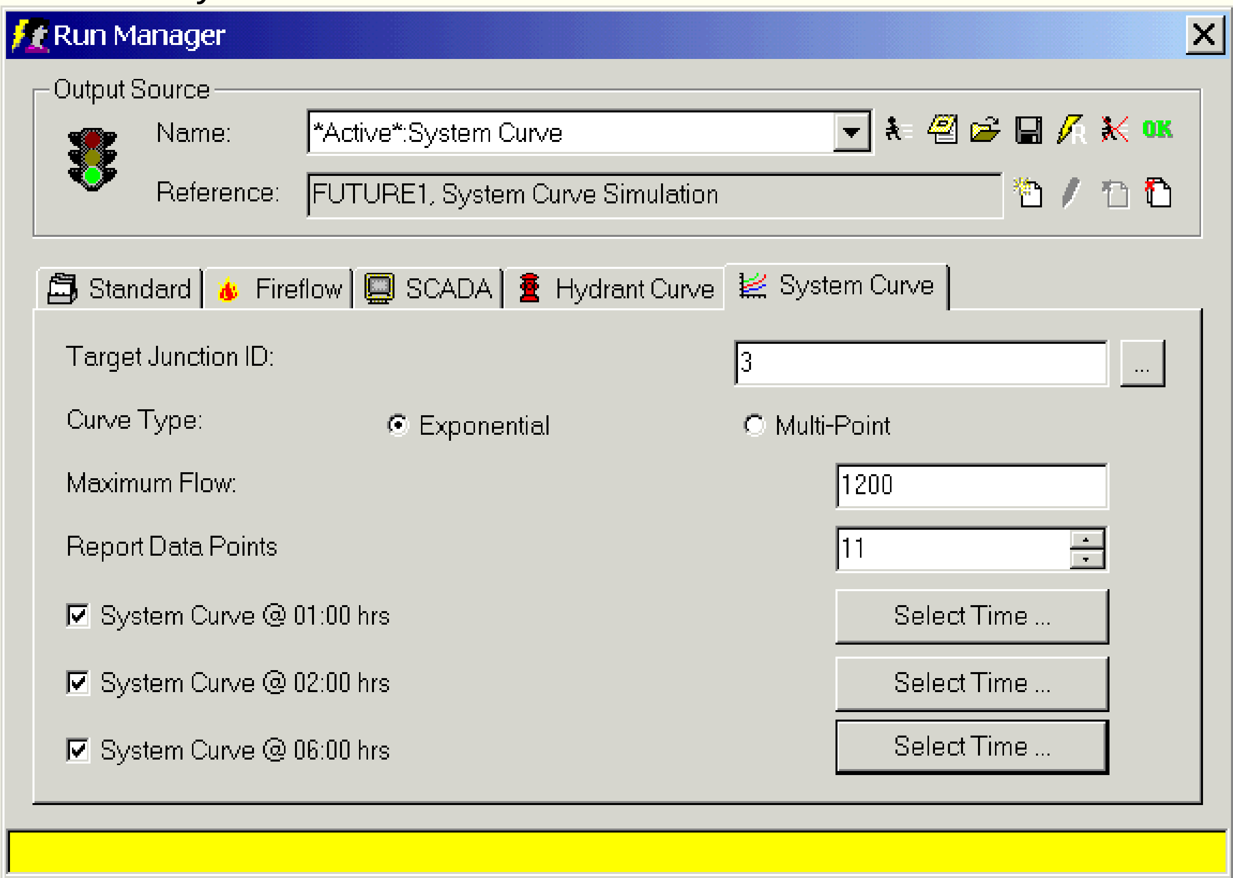
Other Related Topics - [Running a Model](), [Hydrant Curve Methodology]()

System Curve Modeling

System head curves show the relationship between system head and flow capacity for a junction node at different time ranges. This relationship represents the variation in total dynamic head against which pumps will be required to operate under various flow conditions throughout the network. While the user can select any junction node to evaluate - this feature is provided to assist the engineer with determining the expected head curve for delivering a minimum fire flow to a given node that is furthest from a pumping facility.

To develop a system curve for a specific node, from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Run Manager** to see the dialog box below. Select the **System Curve** tab, specify the desired parameters, and choose the Run button. To learn more about the System Curve Modeling methodology click here.

Click on any section for more information:



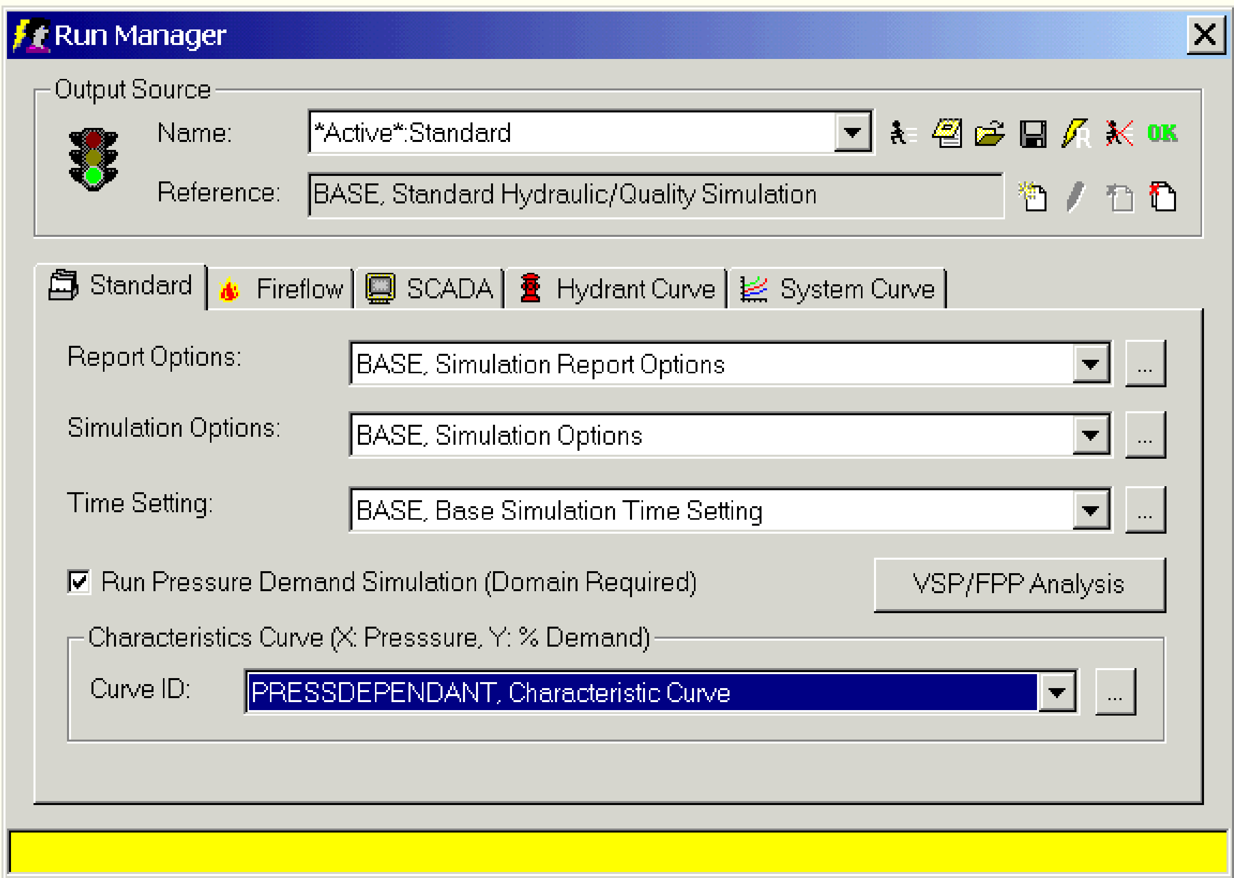
Other Related Topics - Running a Model, System Curve Methodology

---

Pressure Dependent Dialog Box

Pressure dependent demand simulations allow the user to create a hydraulic model that more accurately reflects "real-world" conditions. In some water systems, as pressure drops, the ability to deliver the modeled demand also decreases with respect to a decrease in system pressure. Knowing this, the user can assign a demand vs. pressure curve for selected nodes in the system. Click here for the Pressure Dependent Methodology.

Click on any portion to learn more.



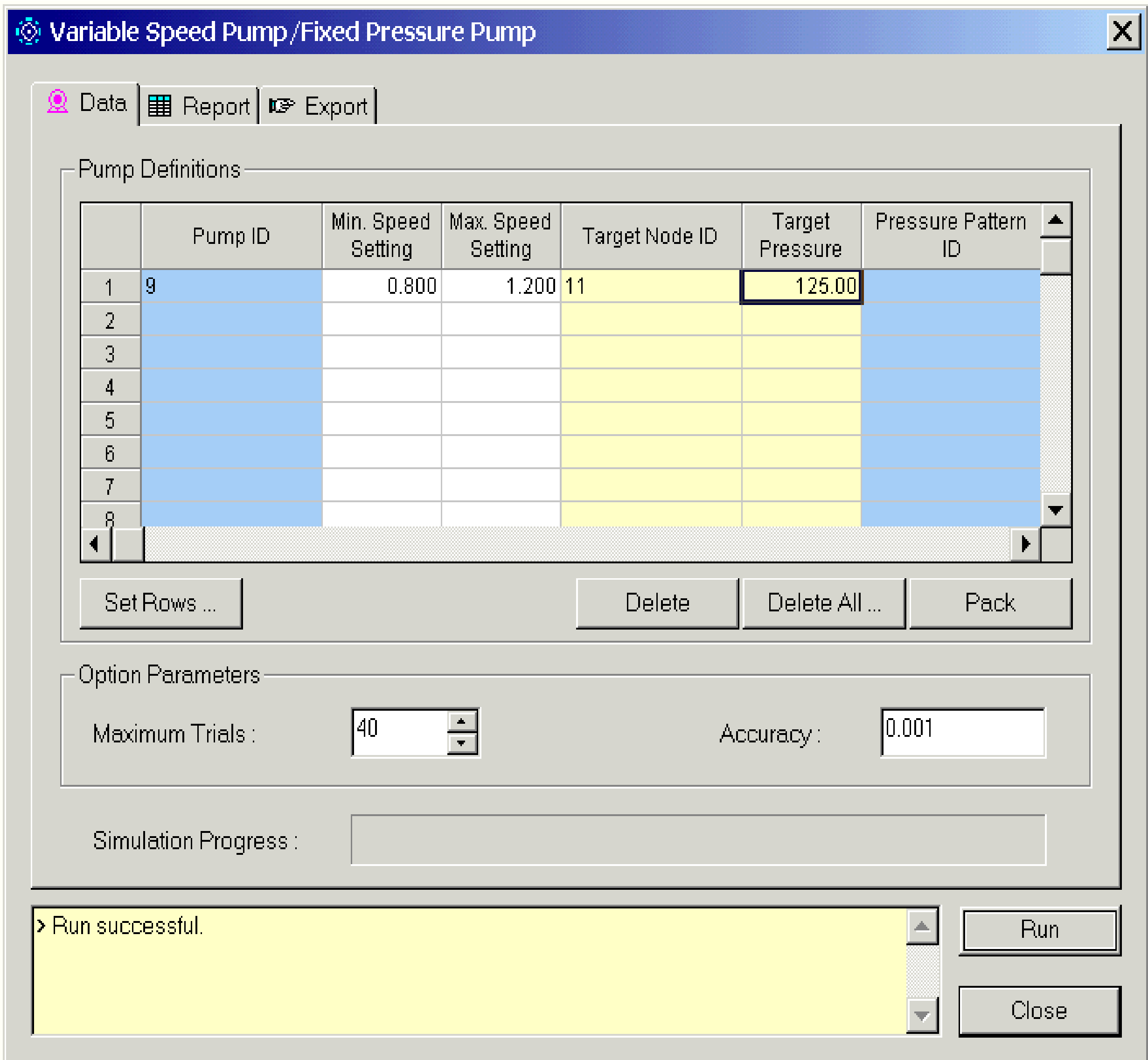
Other Related Topics - Running a Model, Pressure Dependent Demand Methodology

Variable Speed Pump Dialog Box

To conduct a variable speed pump analysis, from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Run Manager** to display the Run Manager dialog box. From the Standard tab, select the VSP/FPP Analysis button to see the following dialog box:

The Data Tab allows the user to input VSP Model parameters. Once the data are entered, click the Run button to conduct the VSP Analysis. To learn more about the VSP methodology click here.

Click on any portion below to learn more.



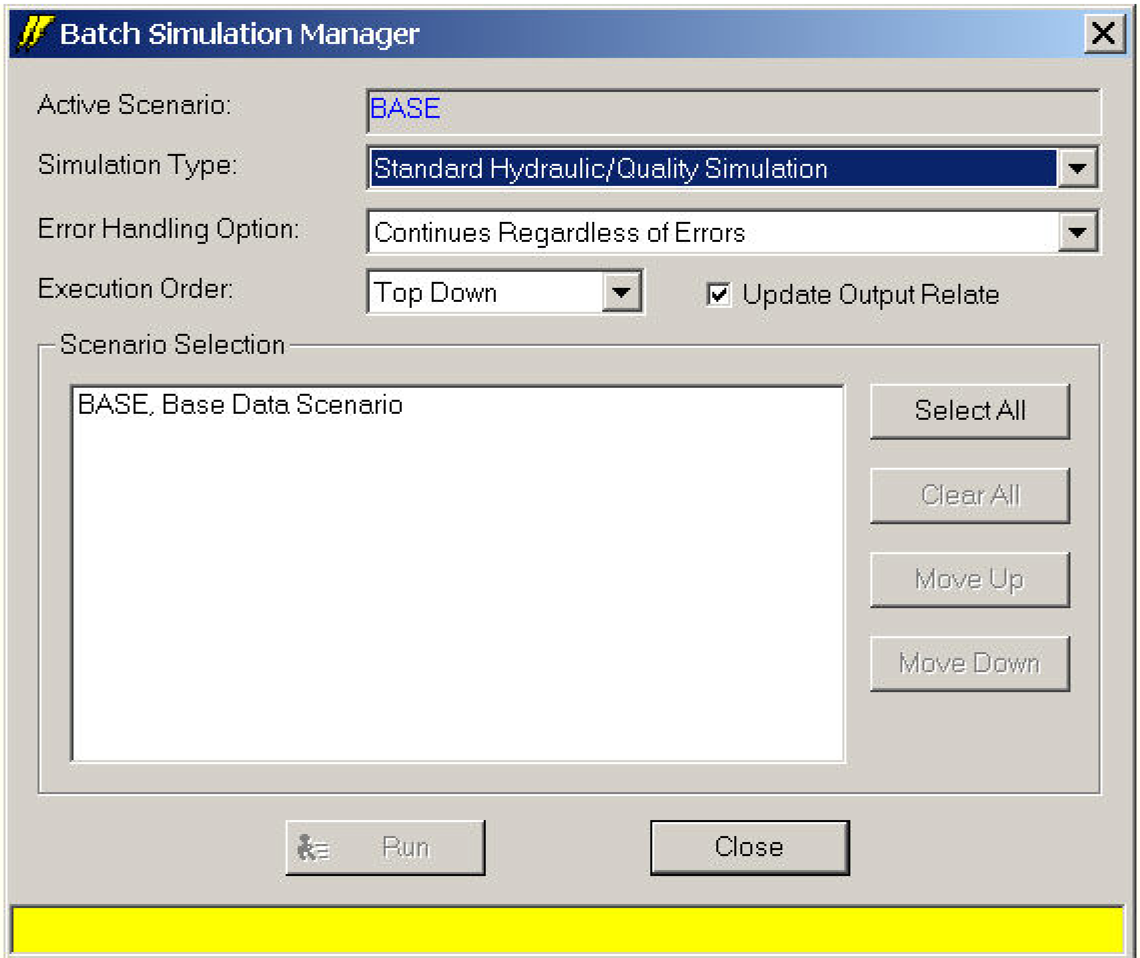
Other Related Topics - Running a Model, Variable Speed Pump Export tab, Variable Speed Pump Report tab, Variable Speed Pumps Methodology

Batch Simulation Manager

The Batch Simulation Manager is used to run models for numerous user-selected scenarios in a single operation. This command is especially useful where several simulations are simultaneously required for a large model. With the Batch Simulation Manager, the user can select the desired scenarios and run each model in a "batch" process. The Batch Simulation Manager is only available for standard simulations and not available for Fireflow, SCADA, etc. model runs.

To activate the Batch Simulation Manager, from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Batch Simulation**.

Click on any portion for more information.



Other Related Topics - [Running a Model](), [Batch Run Methodology](), [Run Manager](), [Run Manager Methodology]()

Controls Overview

InfoWater recognizes three different types of Controls. They are as follows:

- **Initial Status** - Initial status prescribes the initial setting for the InfoWater Project element at the beginning of the Simulation, such as None, open, closed and setting. These controls on the element may be over written by either the Simple Controls or the Logic Control at any time during the Simulation duration. Click here for more information on Initial Status.

- **Simple Controls** - A simple control allows the user to modify the status of elements (pipe, pump or valve) during a hydraulic simulation - depending on the state of some other element in the system as specified by the user. For example, actual field operations may consist of having a pump turn on and off depending on the high and low water levels in a storage tank. To model this type of situation, the user would place a simple control on the element in question. Simple controls apply to both Steady as well as Extended Period Simulation (EPS) models. Click here for more information on Simple Controls.

- **Logic Controls (Complex Controls)** - Rule Based Logic Controls are used by InfoWater to perform a given action during a hydraulic simulation when a user specified condition is met. However, unlike Simple Controls, rule based controls allow for the creation of multiple conditions to be satisfied before an action is performed. Rule Based controls apply to only Extended Period Simulation (EPS) models. Click here for more information on Complex Controls. Logic Controls need to be turned **On** prior to running your model through the **Table of Contents -> Operation** tab **-> Simulation Options** dialog box by checking the **Rule Control** command. InfoWater will read the Complex Controls only when this option is checked.

Note: Rule Based Controls over-write Simple Controls and Simple Controls will over-write Initial Status for the different data elements.

Initial Status Methodology

Initial Status provides a means to prescribe the status of a pipe, a pump and/or a control valve at the beginning of the simulation. This status will remain unaltered unless over-written by a Rule-Based Control (also known as Programmable Logic Controls and/or Complex Controls) or a Simple Control.

Methodology

To specify the initial status of an element do the following:

- Select the appropriate element in your InfoWater model by clicking on it. (see Pipe Initial Status, Pump Initial Status, Valve Initial Status)
- From the **Attribute Browser** dialog box select the **Tools** icon .
- Click on the Initial Status option to launch the Initial Status dialog box.
- Choose the appropriate status and click on **Create** to create the Initial Status.

Other Related Topics - Controls Overview**,** Pipe Initial Status**,** Pump Initial Status**,** Rule Based Control Dialog Box**,** Rule Based Control Methodology**,** Simple Control Dialog Box**,** Simple Controls Methodology**,** Valve Initial Status

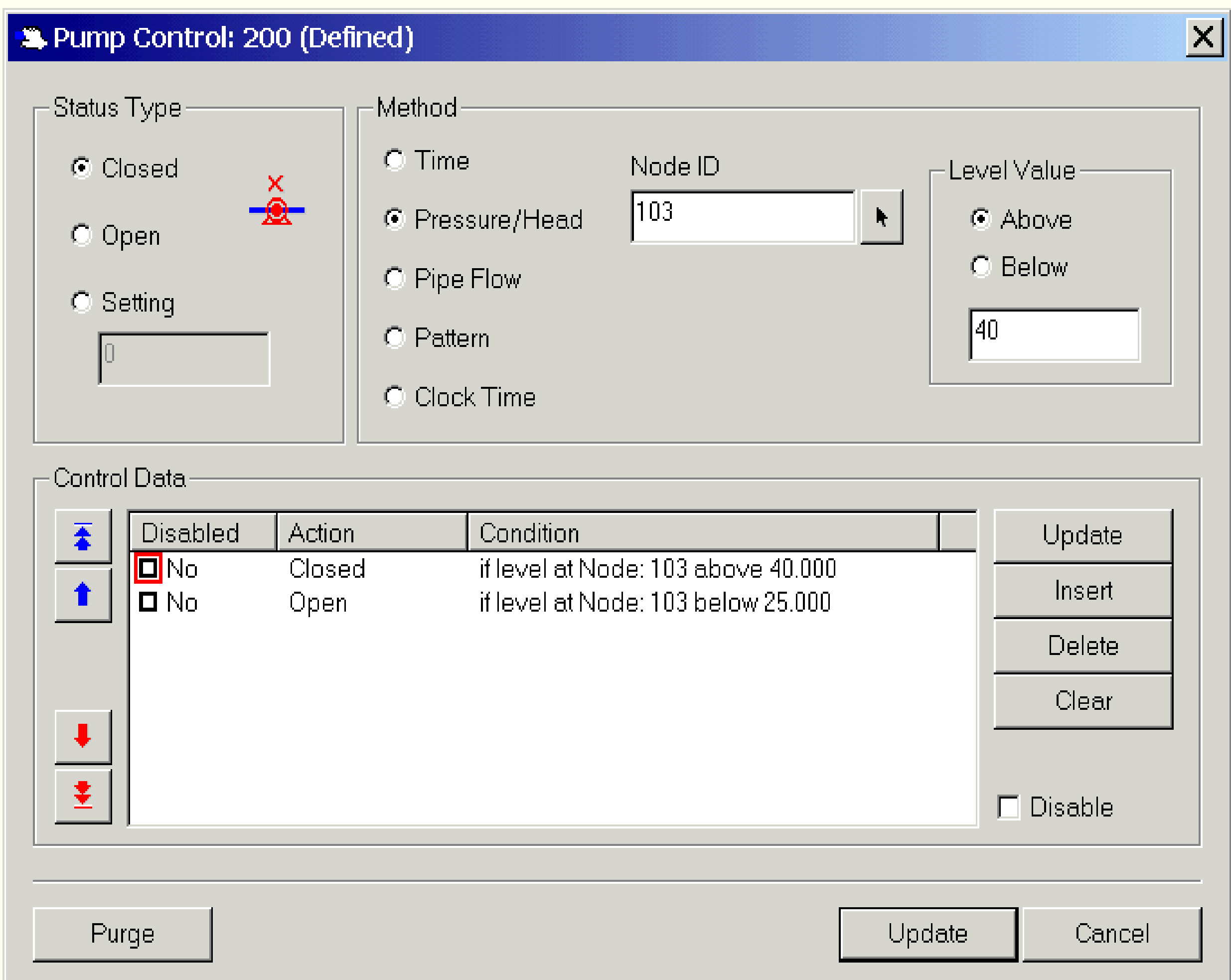
---

Simple Control Dialog Box

The Simple Controls provides a means to establish status controls on a Pipe, Valve and/or a Pump in your Model. Based on the criteria specified the element will be active, closed or will have a setting. The Status Type section defines the action while the condition is ascribed by the Method specified, the Node ID and the Level Values.

In cases where multiple controls are established on an element, the lower placed controls takes precedence and will over-write conflicting controls higher up. Click here for more information on the Simple Controls creation process.

Click on any portion below for more information.



Other Related Topics - Controls Overview, Initial Status Methodology, Pipe Initial Status, Pump Initial Status, Rule Based Control Dialog Box, Rule Based Control Methodology, Simple Controls Methodology, Valve Initial Status

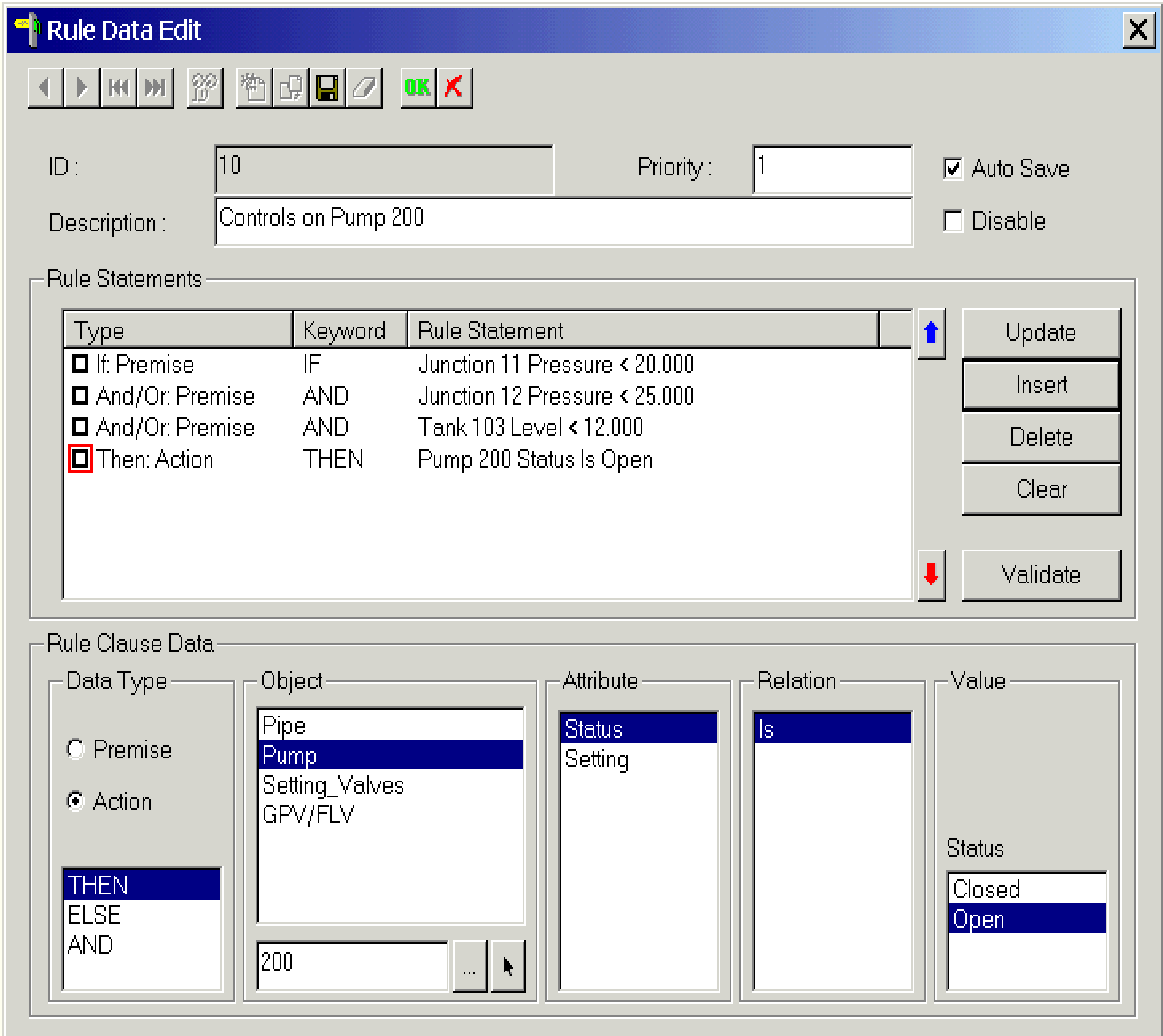
Rule Data Edit Dialog Box

Rule Based Logic Controls are used by InfoWater to perform a given action during a hydraulic simulation when a user specified condition is met. However, unlike Simple Controls, rule based controls allow for the creation of multiple conditions to be satisfied before an action is performed.

The basic building blocks for rule based logic statements are multiple premises (conditions) encountered in the distribution system (if Junction 11 has a pressure below 20 psi and/or Junction 12 has a pressure below 25 psi and/or tank 103 has a level below 12 feet) and their resulting action statements (then turn pump 200 on).

To implement logical controls in a simulation, from the **InfoWater Control center -> InfoWater** button **-> Edit** menu, select **Rule-Based Control**. Once initiated, the dialog box below will appear. The Rule Data Edit dialog box is used to create and maintain logical control rules. Any logical controls you specify with this dialog box are associated with the active (currently loaded) logic set. The dialog box is structured to guide you through the development of your logical control rules. Each section of the dialog box, starting with the Data Type box on the left, allows you to build a new component of your control logic or to associate other components with logic statements. For more information on the Logical Controls creation process, click here.

Click on any portion below for more information.



Note: Logic Sets need to be turned on from the Simulation Options dialog box for InfoWater to utilize logic sets for a simulation. Please refer to section on Enabling Logic Sets.

Other Related Topics - Controls Overview, Initial Status Methodology, Pipe Initial Status, Pump Initial Status, Rule Based Control Methodology, Simple Control Dialog Box, Simple Controls Methodology, Valve Initial Status

Reports, Graphs and Output

After a simulation is run, the next step is to view the results. InfoWater allows you to view data in report or graph form. You can also compare data between two separate model runs (scenarios), import a model run from another InfoWater session or customize a report. Click on any of the following to learn more.

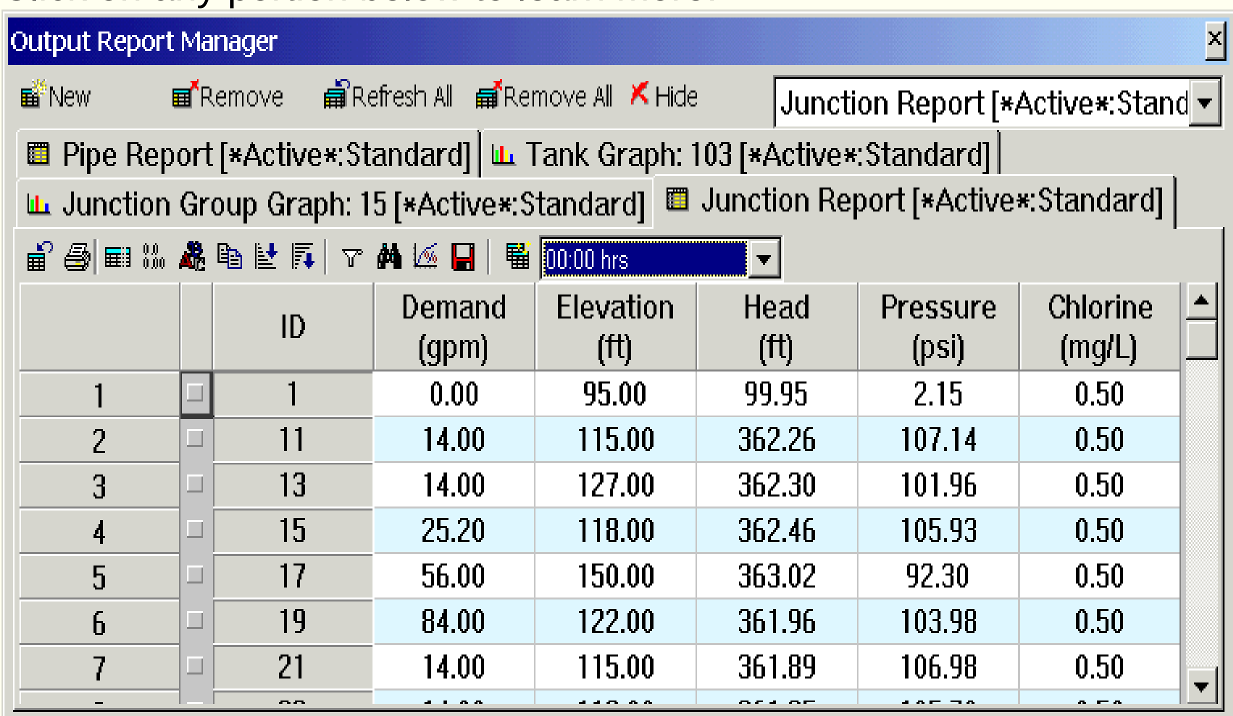
- Output Report Manager - Allows the user to view a report or graph for any *active* or previously run model (output source).
- Available Report Types - Displays the full selection and options available for viewing output in a report format.
- Available Graph Types - Displays the full selection and options available for viewing output in a graph format.
- How to Compare Output Data - A brief tutorial of how to compare hydraulic model results between two models.
- Customized Report - Allows the user to view only those data fields that are desired.
- Query Report - More powerful than a customized report - also allows refined results through database queries.
- Query Summation Report - Use in conjunction with DB Queries to determine and evaluate the different attributes associated with the InfoWater data elements.
- Output Relate - Creates an external database file that contains the results for a hydraulic simulation. Required for viewing results in third party software applications.
- Output Source - Every time a model is run it is stored internally as an output source for future viewing and comparison of results.

Output Report Manager

The Output Report Manger is where the user can generate reports and graphs for any element(s) as modeled in InfoWater.

To initiate the Output Report Manager, from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Output Report Manger**.

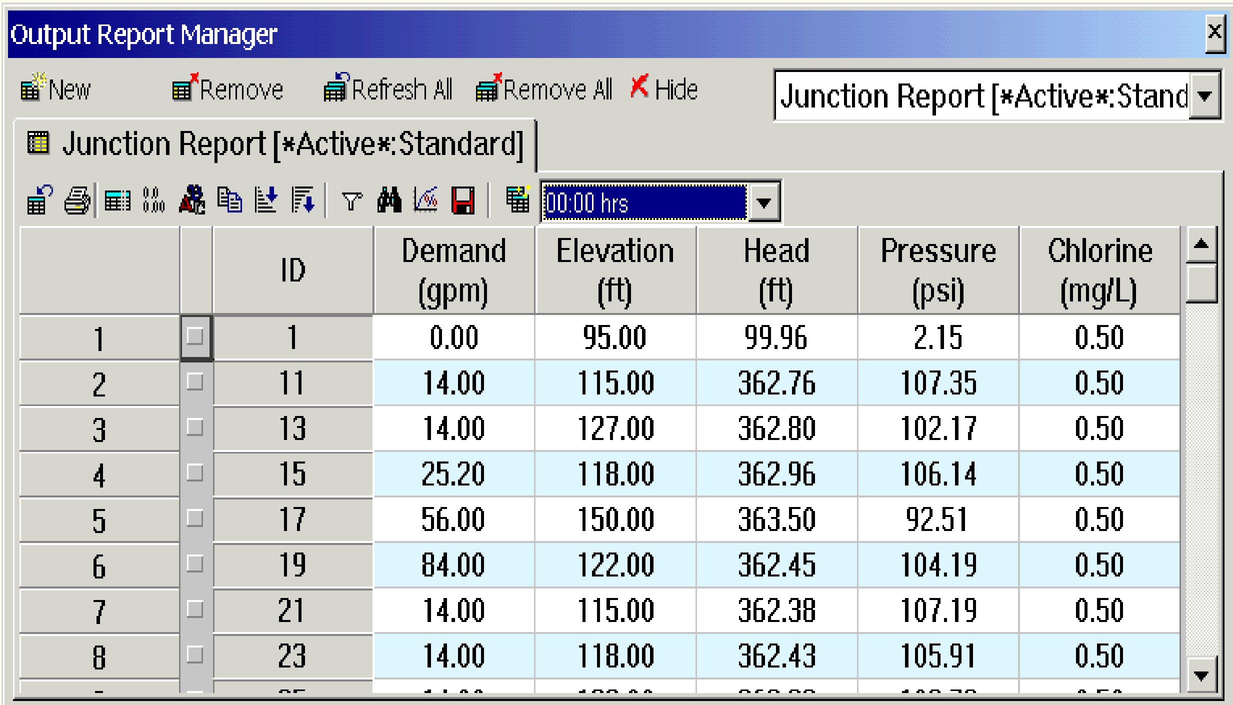
Click on any portion below to learn more:



		ID	Demand (gpm)	Elevation (ft)	Head (ft)	Pressure (psi)	Chlorine (mg/L)
1		1	0.00	95.00	99.95	2.15	0.50
2		11	14.00	115.00	362.26	107.14	0.50
3		13	14.00	127.00	362.30	101.96	0.50
4		15	25.20	118.00	362.46	105.93	0.50
5		17	56.00	150.00	363.02	92.30	0.50
6		19	84.00	122.00	361.96	103.98	0.50
7		21	14.00	115.00	361.89	106.98	0.50

Available Report Types

Reports may be viewed in the Output Report manager as shown below. Click on any portion for more information:



		ID	Demand (gpm)	Elevation (ft)	Head (ft)	Pressure (psi)	Chlorine (mg/L)
1		1	0.00	95.00	99.96	2.15	0.50
2		11	14.00	115.00	362.76	107.35	0.50
3		13	14.00	127.00	362.80	102.17	0.50
4		15	25.20	118.00	362.96	106.14	0.50
5		17	56.00	150.00	363.50	92.51	0.50
6		19	84.00	122.00	362.45	104.19	0.50
7		21	14.00	115.00	362.38	107.19	0.50
8		23	14.00	118.00	362.43	105.91	0.50

The following report types are available from the Output Report Manager. Choose a report for more information:

Standard Analysis: A Standard InfoWater Analysis may either be a Steady State Analysis or an Extended Period Analysis. A standard EPS analysis may also include results for Water Quality Analysis. All the standard pressure, flow and water quality outputs are included in the following reports.

- Junction Report
- Tank Report

- Reservoir Report
- Valve Report
- Pump Report
- Pipe Report
- Range Reports (Junction, Tank, Reservoir, Pump, valve & Pipe)

Energy Management Analysis: An Energy Management analysis is conducted in conjunction with the Standard Analysis. Hence in addition to all the report types above, Energy Analysis provides the following additional report types.

- Energy Cost Report
- Energy Summary Report
- Demand Cost Report

SCADA Analysis: SCADA analysis provides the following report types in addition to the standard analysis report types. They include the SCADA comparison Flow report, the SCADA comparison pressure report and finally the SCADA alarm reports.

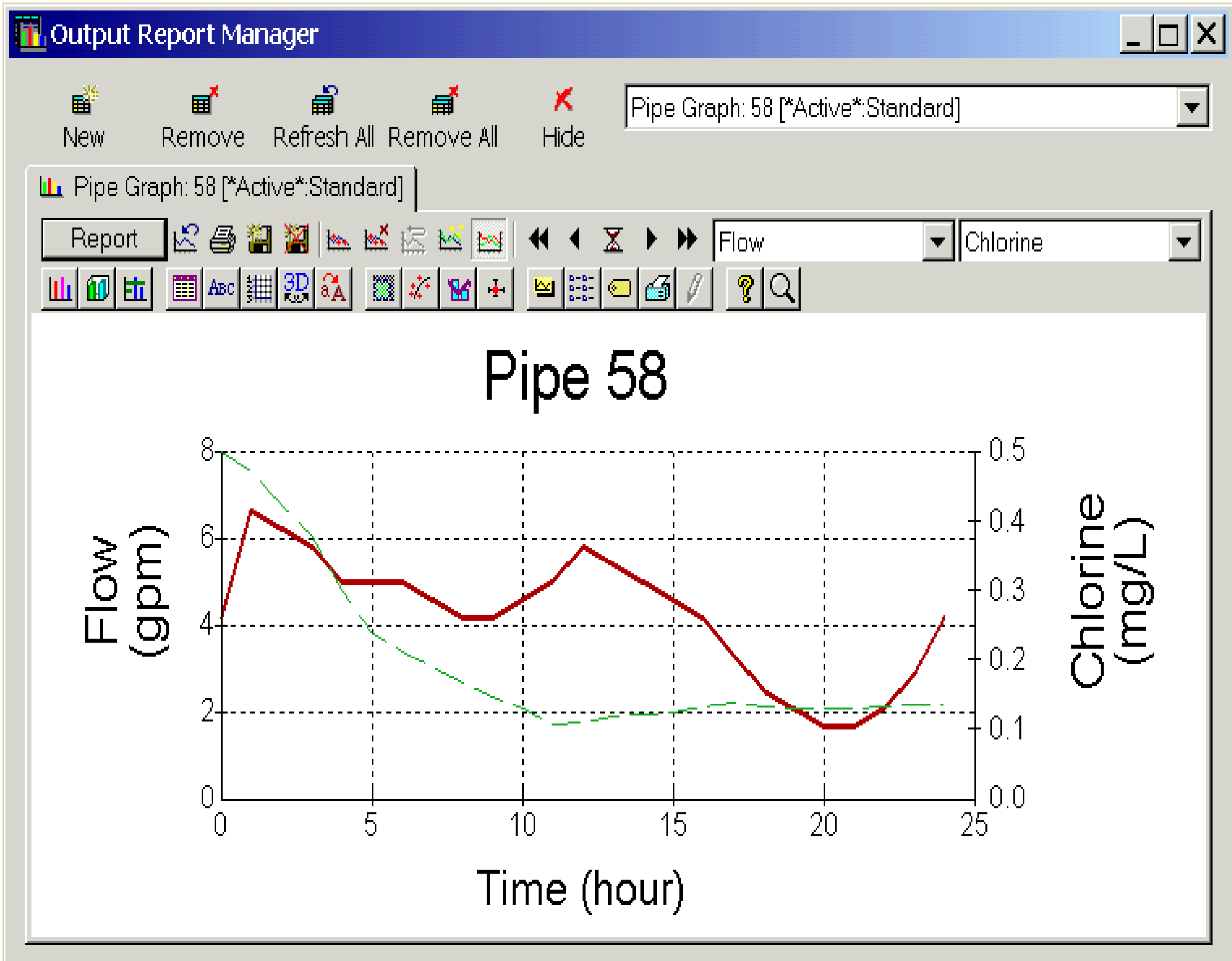
- SCADA Flow
- SCADA Pressure
- SCADA Alarm

Fire-Flow Analysis: Fire-Flow analysis provides two reports, the Hydrant report or the fire-flow report and the fire-flow design report. The fire flow reports are used to display, query, and report fire flow simulation results for all fire flow nodes (active junction nodes assigned a fire flow demand at the time the fire flow simulation was run).

- Fire Flow Report
- Fire-Flow Design Report

Available Graph Types

Graphs may be viewed in the Output Report manager as shown below. Click on any portion for more information:



The following graph types are available from the [Output Report Manager](). Choose a graph for more information:

Standard Analysis: A Standard InfoWater Analysis may either be a Steady State Analysis or an Extended Period Analysis (EPS). A standard EPS analysis may also include results for Water Quality

Analysis. All the standard pressure, flow and water quality outputs are included in the following graphs.

- Junction Graph
- Tank Graph
- Reservoir Graph
- Valve Graph
- Pump Graph
- Pipe Graph
- Junction Group Graph
- Tank Group Graph
- Reservoir Group Graph
- Valve Group Graph
- Pump Group Graph
- Pipe Group Graph
- HGL Graph
- Pump Curve
- System Demand
- Average Reaction Rates (%)
- Frequency Graph

Energy Management Analysis: An Energy Management analysis is conducted in conjunction with the Standard Analysis. Hence in addition to all the graph types above, Energy Analysis provides the following additional graph type.

- Energy Curve

SCADA Analysis: SCADA analysis provides only the following graph types

- HGL Graph
- Pump Curve

Hydrant Analysis: Hydrant-curve analysis provides only the following graph type.

- Hydrant Curve

System Curve Analysis: System-curve analysis provides the following graph type.

- System Curve

Steps to Compare Scenario Data

The Output Report Manager offers two comparison tools; a comparison report and a comparison graph. You may display a report or graph showing simulation results for the *active* (currently loaded) scenario, and then use a previously loaded output source storing results from a simulation run for a previously loaded scenario.

How Do I...

- Compare Report Data?
- Compare Graph Data?

Output Report Manager - Comparison Reports

 **Compare Report Icon**

In order to compare tabular data from two sources, you must first create the appropriate output source in which a comparison is to be made. Once this is done, you can click on the compare report icon to compare any output source with the current *active* report being viewed. Click here to learn a step-by-step process for creating comparison reports.

As an example of a comparison report, the illustration below shows result fields from the *active* scenario with the output source FUTURE2.Standard. Notice that the *active* scenario is not preceded with the output source name.

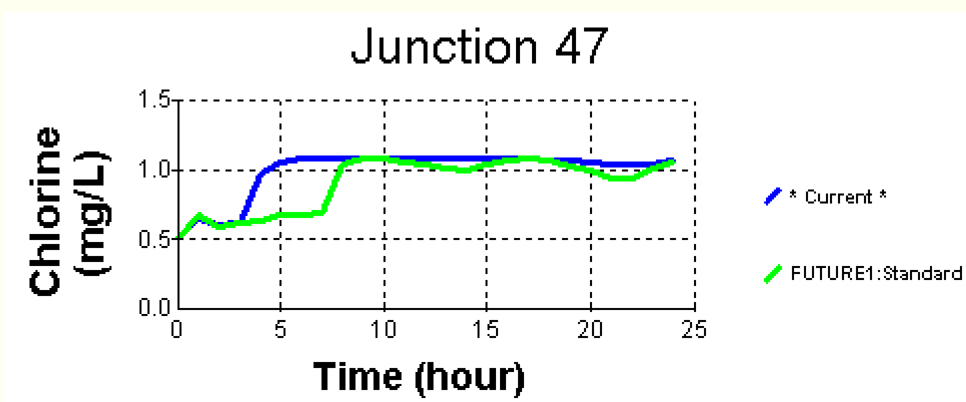
ID	Chlorine (mg/L)	Chlorine [FUTURE2:Standard] (mg/L)	Pressure (psi)	Pressure [FUTURE2:Standard] (psi)
1	0.50	0.50	2.15	2.15
11	0.50	0.50	107.35	107.35
13	0.50	0.50	102.17	102.17
15	0.50	0.50	106.14	106.14

Output Report Manager – Comparison Graphs

Compare Graph Icon

In order to compare graphs from two sources, you must first create the appropriate output source in which a comparison is to be made. Once this is done, you can click on the compare graph icon to compare any output source with the current *active* graph being viewed. Click here to learn a step-by-step process for creating comparison graphs.

As an example of a comparison graph, the illustration below shows values from the *active* scenario compared with the FUTURE1.Standard output source. Notice that the legend refers to the active output source as *Current*



Steps to Develop Comparison Reports and Graphs

The following steps illustrate how to develop a comparison report or graph. These steps assume you have developed two or more scenarios and run a hydraulic simulation for each scenario. In essence, more than one output source is available.

Develop and Run Scenarios

1. Develop the first scenario and then activate that scenario. For this illustration this scenario will be named “FUTURE1”, representing future conditions.

2. Run a simulation for the “FUTURE1” scenario using the Run Manager. Simulation results are stored in the *active* output source and are immediately available for evaluation.

3. Develop the second scenario and then activate the second scenario. For this illustration this second scenario will be named “EXISTING”, representing existing conditions. When you load the “EXISTING” scenario, simulation results for the “FUTURE1” scenario are moved from the *active* output source to a new output source named “FUTURE1”.

4. Now run the second, recently activated scenario named “EXISTING”. Simulation results are stored in the *active* output source, replacing simulation results from the previously-loaded scenario, and are now available for evaluation.

Develop a Comparison Graph

5. Open the Output Report Manager.
6. Choose the “New” button on the Output Report Manager window. The Output Report and Graph dialog box appears on the screen.
7. Choose the *Active*:Standard output source (storing simulation results for the currently loaded “EXISTING” scenario), choose the Graph tab and the choose the junction node option. Once selected, choose “OK”.
8. Pick a junction node from the network map. A graph is displayed showing simulation results from the currently loaded “EXISTING” scenario for the selected junction node.
9. Choose the Compare Graph icon to overlay results from the desired output source. The Select Output Source(s) to Compare dialog box appears on the screen
10. Click once on the FUTURE1:Standard output source and then choose the “OK” button. The graph on the Output Report Manager should now show results for the currently-loaded scenario (“EXISTING”, referred to as *Current* on the graph legend) and the previously-loaded “FUTURE1” scenario.
11. You may now customize the graph as you desire using any of the available graph customization tools.

You may follow the same steps shown above to develop and customize a comparison report.

Customized Report

Depending on the options chosen in the first three steps of Customized Report creation process, data fields are displayed in the Report. Click here for information on the Customized Report creation methodology.

For more information on the different available report types click here.

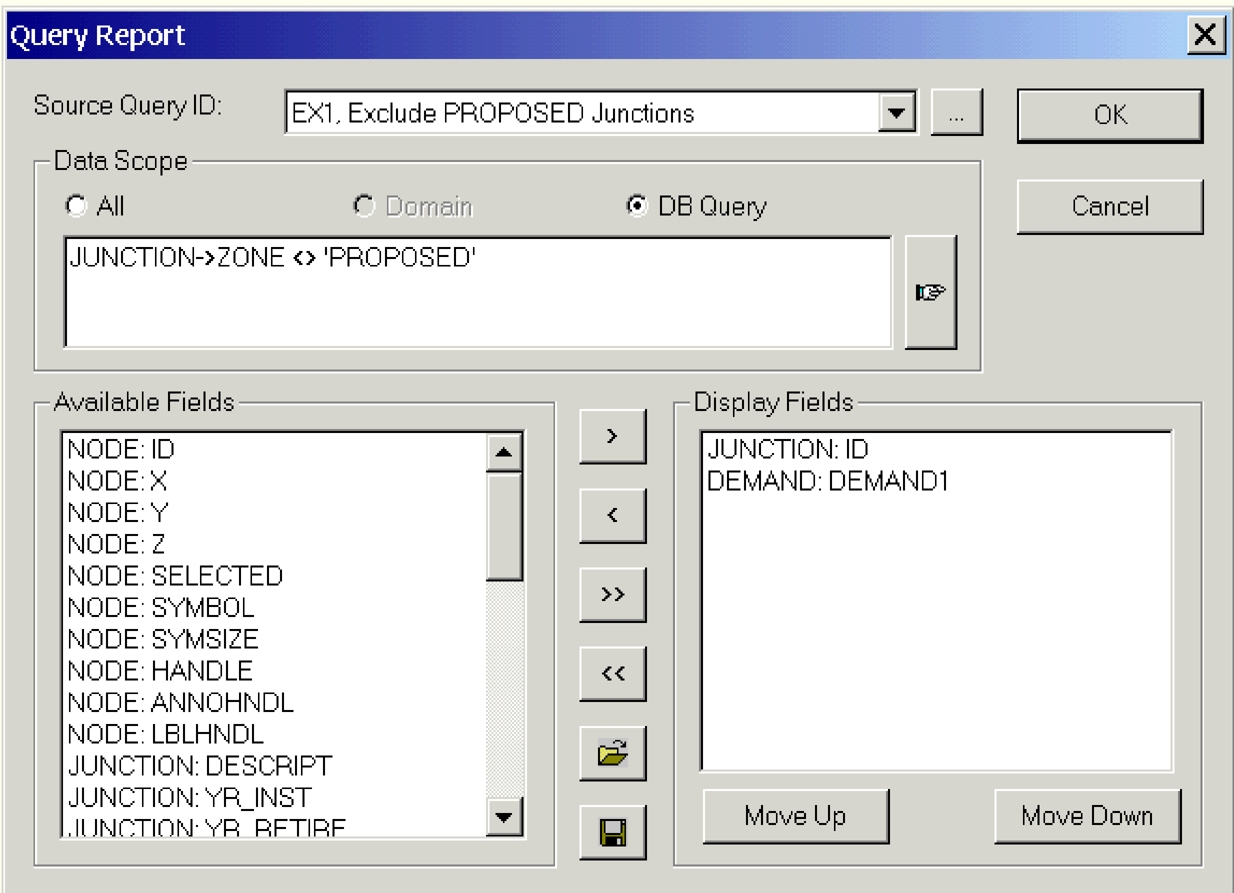
	JUNCTION: ID (Char)	DEMAND: DEMAND1 (Real)	JUNCTION: ELEVATION (Real)	RANGE: DIFFERENCE (gpm)
1	1	0.00	95.00	0.00
2	3	0.00	95.00	0.00
3	5	0.00	115.00	0.00
4	7	14.00	115.00	16.80
5	9	14.00	130.00	16.80

Other Related Topics - Customized Report Methodology, Define Data Source, Define Display Fields, Define Display Scope

Query Report

A query report is used to selectively choose which input and output data will appear in a report. However, unlike a customized report, a query report is required to relate data from an Output Relate with output data in a report format. Click here for more information on the Query Report Creation process.

Click on any portion below for details.



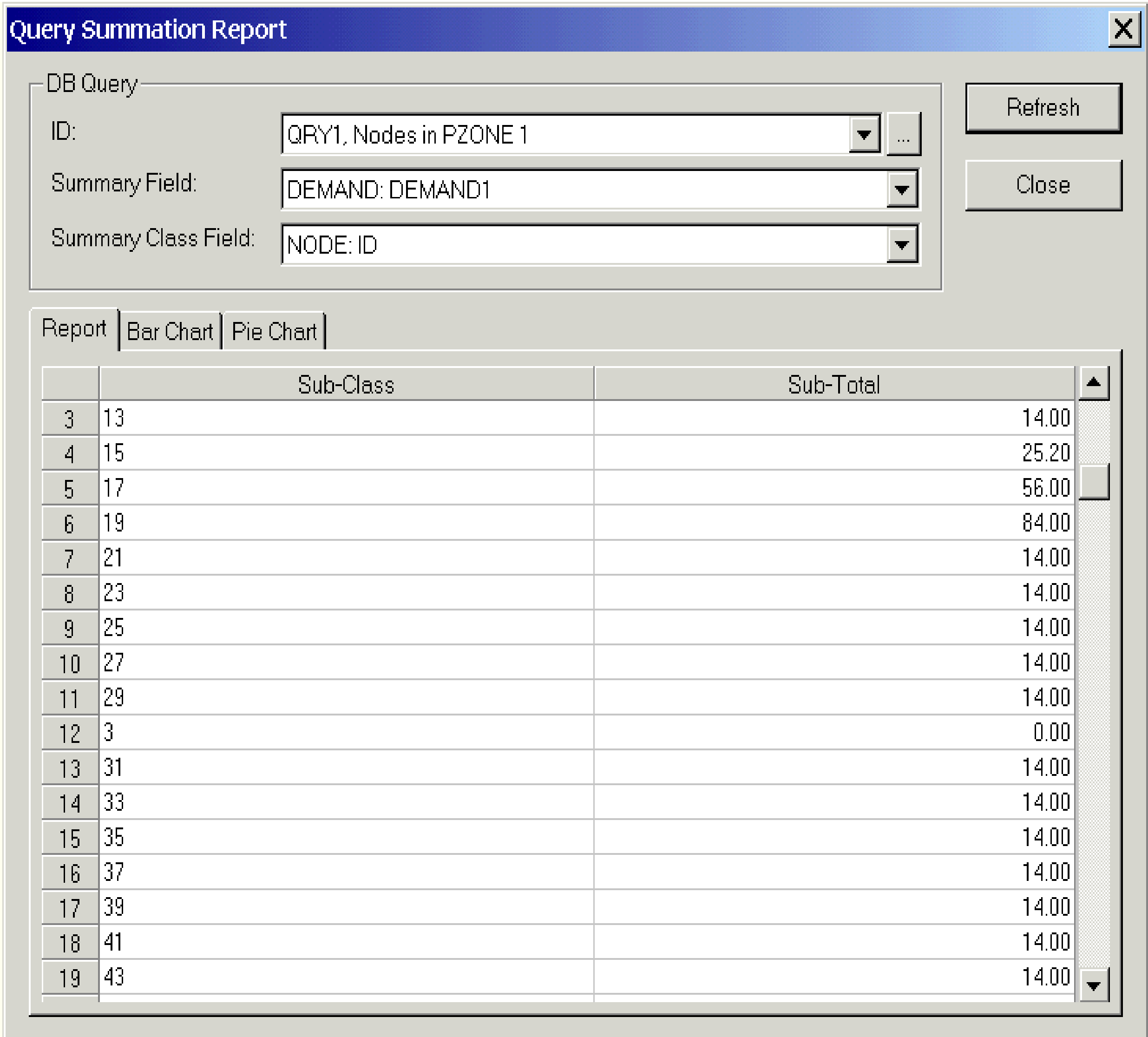
Other Related Topics - Query Report, Query Report Methodology

Query Summation Report

The Query Summation Report may be used in conjunction with DB Queries to determine and evaluate the different attributes associated with the InfoWater data elements (Junctions, Tanks, Reservoirs, Pumps, Valves and Pipes) that satisfy the DB query.

For instance in the following Report you may evaluate all the attributes of junctions in pressure zone 1. To learn more about the Query Summation Report methodology click here.

Click on any portion to learn more.



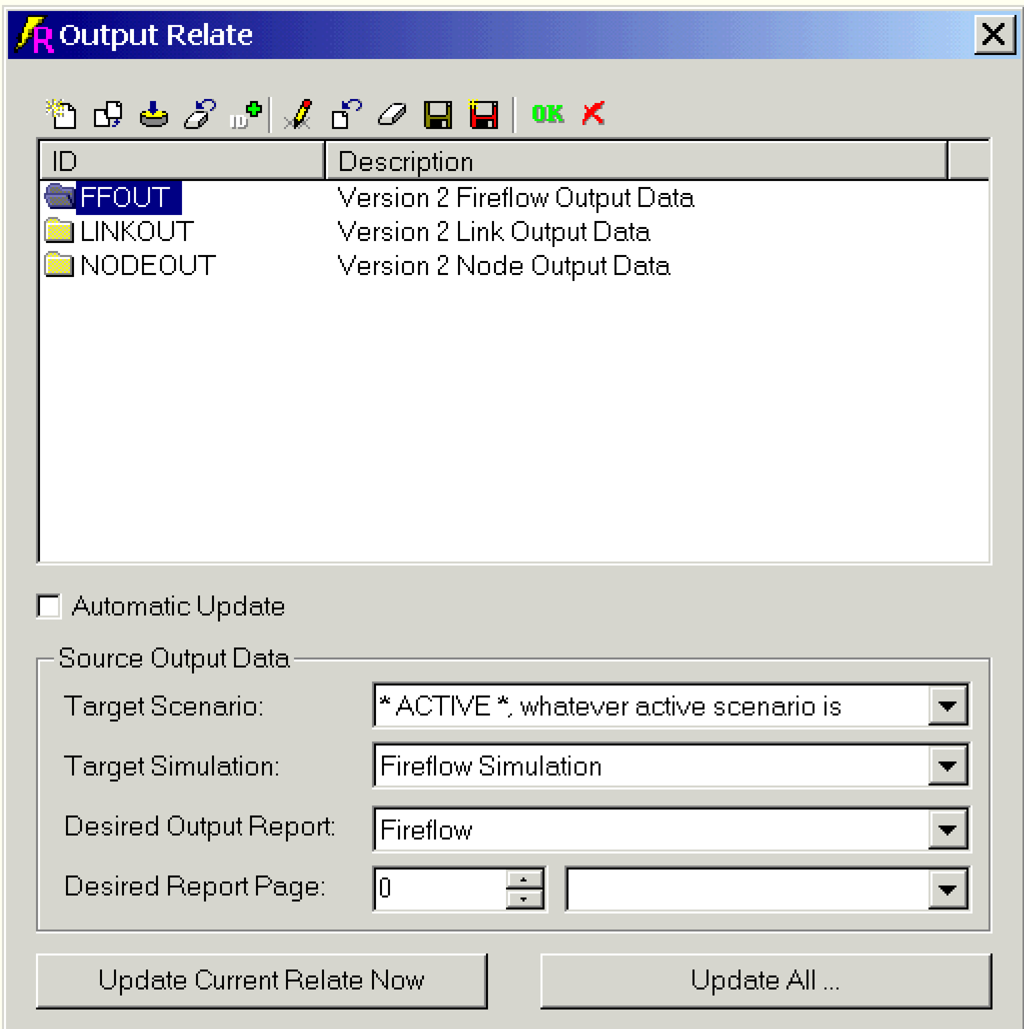
Other Related Topics - Query Summation Report Bar Chart, Query Summation Report Methodology, Query Summation report Pie Chart

---

Output Relate Dialog Box

To create a new Output Relate, from the **Operation** tab of the **Table of Contents**, select the Output Relate folder and right mouse click and select the **New** command. This will initiate the **Output Relate** dialog box as shown below. For a detailed explanation on the Output Relate methodology and its applicability in InfoWater click here.

Click on any section below to learn more.



Other Related Topics - Output Relate Methodology**,** Query Output Relate**,** Query Output relate Methodology**.**

Output Source

An output source is nothing more than an association between InfoWater and the output (.out) file created from a simulation run. InfoWater refers to model runs as output sources, whether they be *active*.Standard or ScenarioX.Standard, every model run is affiliated to an external output file by its "output source" name.

From the icons at the top of the Run Manager, the user has full control over the associations between an output source in InfoWater and its external output file.

Give Me An Example

If the user has created and modeled four custom scenarios, the output for each scenario will be made available for report and graphing purposes under the Output Report Manager (under the All Output Sources box). If the user then wanted to add (load) another model run performed by someone else (perhaps multiple people working on the same project file), then by clicking on the Load Output Source icon, the user can assign an external output file to a user-defined (internal) output source. Once this is done, the newly imported output source can now be compared with other previously run "output source" models (via graphs and reports).

How do I...

- Load (Create) an Output Source?
- Remove an Output Source(s)?
- Rename an Output Source?
- Save an Output File?

More on Output Sources

Each output source is identified by a unique name and is associated with a single output file. The most recently completed analyses are given the following names:

Active.Standard	Output source data (simulation results) for the most recently completed hydraulic and water quality simulation.
Active.Fireflow	Output source data for the most recently completed fireflow simulation.

Active.SCADA	Output source data for the most recently update from SCADA system data.
Active.Hydrant Curve	Output source data for the most recently generated hydrant curve.
Active.System Curve	Output source data for the most recently completed system head curve.

When assigning an output file to an output source, the data type need to match. In other words, a "standard" hydraulic run cannot be imported into a fireflow simulation.

When you switch to another scenario, output sources for standard simulations are immediately available as described above. However, fireflow, SCADA, hydrant curve, or system head curve output sources must be manually loaded.

Note: Hydrant curves and system head curve output sources store only the most recent curve generated for the active scenario. For example, if you activate a scenario entitled “CUSTOM1” and generate hydrant curves for three nodes with IDs 17, 25, and 34 in that order, the hydrant curve output source for the current scenario only stores the hydrant curve for node 34.

Standard Simulations

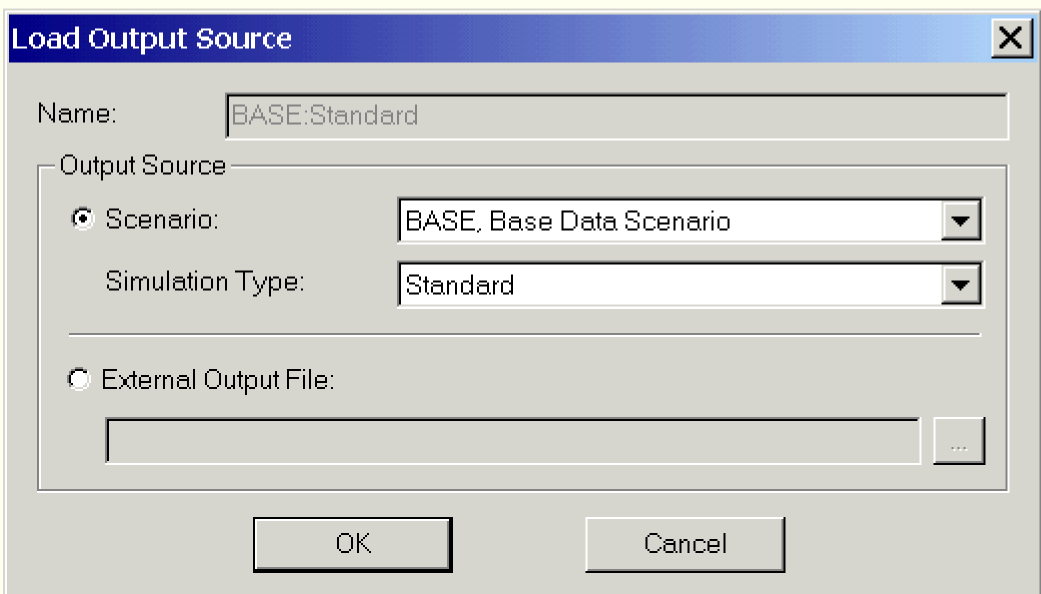
As you run a standard simulation, the output source name for that simulation is *Active*.Standard. The previous *Active* simulation is converted to a user-defined source and renamed based on the active scenario at the time that simulation was performed.

For example, if the previous standard simulation was based on your scenario named “CUSTOM1”, then the output source from the previous run is renamed CUSTOM1:Standard.

Once a scenario is run, the output source will always be tied to the default output file unless the user manually re-assigns that relationship.

Loading Output Sources

To load (create) an output source, from the **InfoWater Control center -> InfoWater** button **-> Tools** menu, select **Run Manager**. At the Run Manager dialog box, select the **Load Output Source** icon . From here, the user will see the following dialog box:



There are two types of output sources that the user can load (see below):

- Scenario - This option is only available if the user has previously removed an existing output source via the "Remove Output Source" icon. By selecting this option, the user can reinitialize a previously run model without having to re-run a scenario simulation.

- External Output File - This option is used to assign an existing .out file to a user specified output source "name" at the top of the dialog box. Once the link is established, the user can now compare existing "output source" data with the newly imported external data.

Remove Output Source(s)

To remove an output source from the current InfoWater project, select either the **Remove Output Source** or the **Remove All Output Sources** icon . The first will remove the currently selected output source as shown in the upper left hand corner of the Run Manager while the latter will remove all output sources.

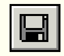
Note: Be advised that removing all output sources will not only remove all external output source linkages, but will also remove all internal scenario outputs.

Rename an Output Source

To rename an output source, from the Run Manager, select the Output Source Name and then click on the **Edit Name** icon to rename the output source.

Saving an Output File

To save an output file, the user must first make active the scenario for which an output is desired. Once a scenario is active, go to the

Run Manager and run the *active* model (standard, fireflow, etc.). After the model has run, select the **Save** icon  to save the model run to a file name other than the default.

System head curve and hydrant curve output sources store only the most recently generated curve for the active scenario. If you wish to permanently store a curve for numerous nodes for the active scenario, you should manually save each hydrant/system curve output source before continuing.

Note: Unless the user has disabled the "**Enabled Output Save As**" feature from the **InfoWater Control center -> InfoWater** button **-> Tools -> Project Preferences** dialog box **-> Operation Settings** tab, all model runs will be automatically saved to their default names under their respective scenario database folders.

Database Management

Nearly every piece of information contained in InfoWater is stored in an external database that is accessible to the user. This open architecture allows for full editing capability in the InfoWater session. Click on any of the features below to learn more.

- DB Editor - The primary tool used for editing any database in InfoWater.
- Group Editing - Allows the user to assign multiple conditions and settings for selected data elements (ex. fireflow, initial status, etc.)
- DB Query - Allows the user to create queries that can be used for other purposes (domains, facility sets, etc.)
- Query Sets - A compilation of DB Queries that can be used for anything from facility sets to changing colors in the map display.
- Selection Sets - Stored facility sets that can be activated at any future time.

DB Editor

The DB Editor provides a means to view all the DB Tables. Any number of DB Tables can be open for inspection, modification and/or data edit. All the data corresponding to the InfoWater Project may be accessed through these DB Tables.

For more information about the different types of data stored in the InfoWater DB Tables, please refer to the section on DB Tables.

The DBEditor allows the user to:

- **View Data** - View (including custom format) modeling data tables.
- **Add/Delete/Modify data -** Click on any field to change it. The fields that cannot be edited are colored in gray.
- **Add/Delete Custom Fields** - Customize data tables by adding user-defined fields. Click here for more information on adding custom fields.
- **Perform Block Edits** - Allows the user to edit table data values, one record at a time or numerous records simultaneously. The Block Editor is a powerful tool when using the used to perform block edits.

Once a table has been opened, the following dialog box will appear.

Click on any of the elements or icons below to learn more about that feature:

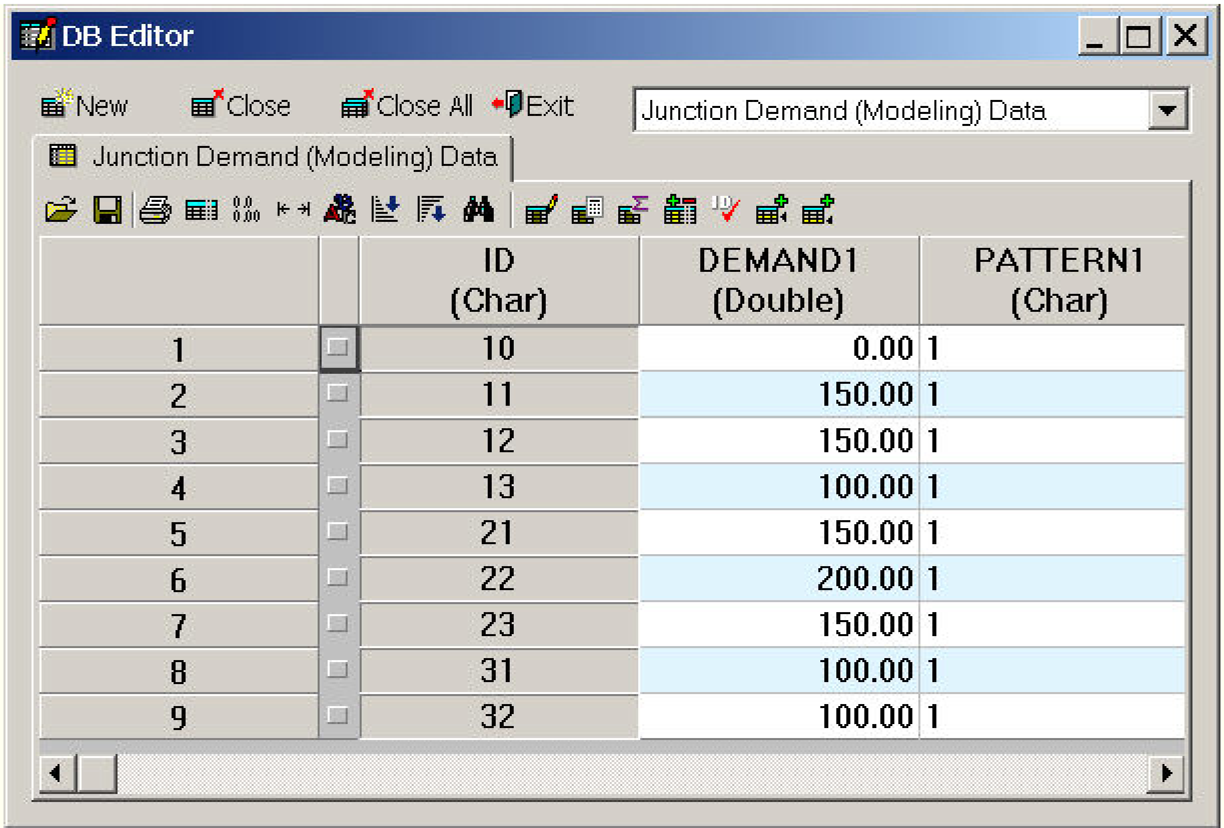

DB Editor
New
Close
Close All
Exit
Junction Demand (Modeling) Data
Junction Demand (Modeling) Data
ID (Char)
DEMAND1 (Double)
PATTERN1 (Char)
1 10 0.00 1
2 11 150.00 1
3 12 150.00 1
4 13 100.00 1
5 21 150.00 1
6 22 200.00 1
7 23 150.00 1
8 31 100.00 1
9 32 100.00 1

Table Records

- **Table Record Number** – A sequential counter of table records. Used for visual reference only.
- **Save Status** – Indicates whether or not a record has been modified. Any records that have been modified will be marked with a red check . To save any modifications, choose the Save button at the top of the table record display.
- **ID** – InfoWater component identifier used for reference by other InfoWater functions and used for modeling. The contents of this field cannot be edited from the Database Editor.
- **Table Fields** – Fields specific to the current table. The field data type (Double), (Num), (Char), etc. is displayed directly below the field name as an indication of the acceptable data format for that field. Field order for fixed fields (those required by InfoWater) cannot be changed. Field order for custom fields can be changed.

Click here for more information on the different InfoWater Tables.

Additional properties

- Empty numeric data will display as “0.00” (depending on user-specified decimal precision). Empty character data will display as empty data cells.
- Selected records (rows), fields (columns), or individual data values (cells) will be highlighted in black.
- Non-editable fields are displayed in gray. These fields are typically graphical/locational in nature and must be edited using other InfoWater functions.

Other Related Topics - Advanced Application Data**,** Control Data Tables**,** Element Geometric Data Tables**,** Element Hydraulic Data Tables**,** Element Information Data Tables**,** Extended Modeling Data Tables**,** Pattern & Curve Data Tables**,** SCADA Settings Tables**,** Water Quality Data Tables**,** DB Tables**.**

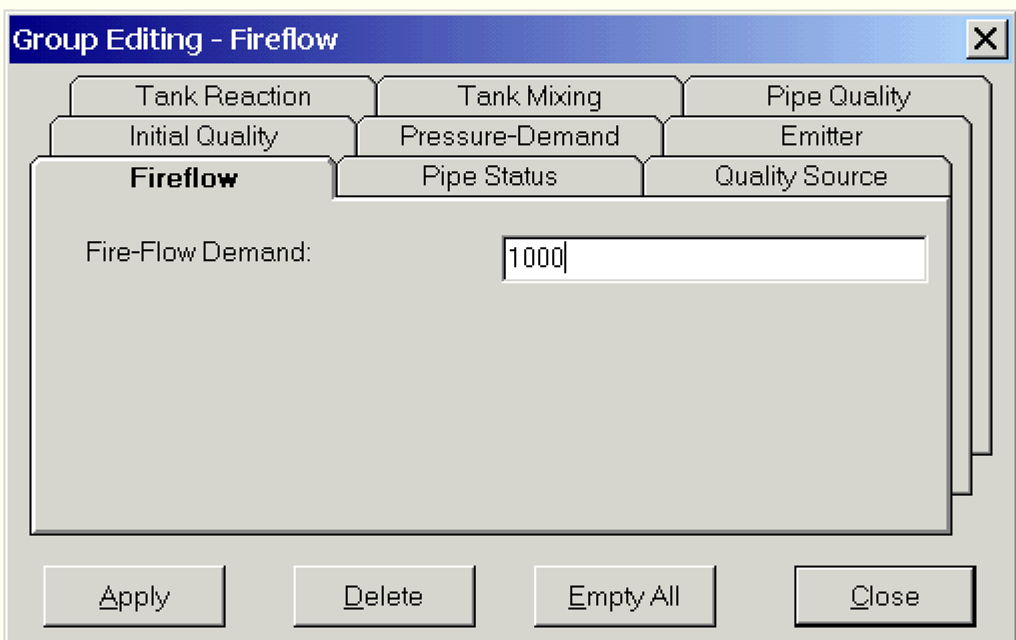
Group Editing (on Domain or Selection)

The Group Editing command allows you to specify modeling properties of multiple network components in a single operation.

To run the command, from the **InfoWater Control Center** -> **InfoWater** button -> **Edit** menu, select the **Group Editing on Domain** or **Selection**. For more information on the Group-Editing Methodology, click here.

Once any of the elements selected for a group edit are populated with data, the information input by the user will also appear in various database tables under the DB Editor. For example, when a group of nodes are assigned a fireflow through the group edit, those nodes will be added to the Fireflow Demand database found under the DBEditor - Folder *4. Extended Modeling Data*.

Click on any section below to learn more:

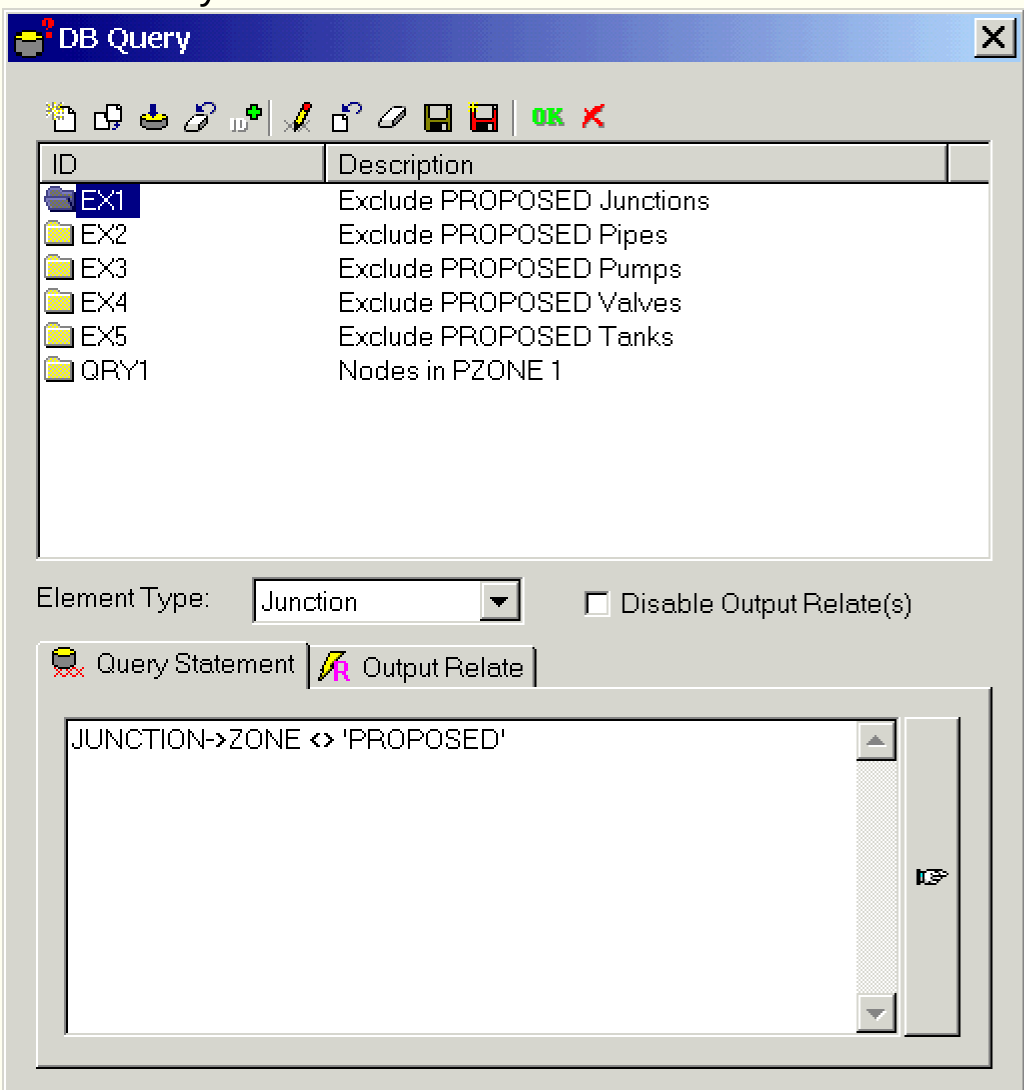


Other Related Topics - Edit Domain Attributes, Edit Selection Attributes, Emitter, Fire-Flow, Group Editing Methodology, Initial Quality, Pipe Quality, Pipe Status, Pressure Dependent Demand, Quality Source, Tank Mixing, Tank Reaction

DB Query Dialog Box

To create a database query, from the **Operation** tab of the **View** menu -> **Table of Contents** command, select **DB Query** and right click and choose the **New** command. For more information on the DB Query Methodology, click here. This will initiate the DB Query dialog box as shown below.

Click on any section for more information:

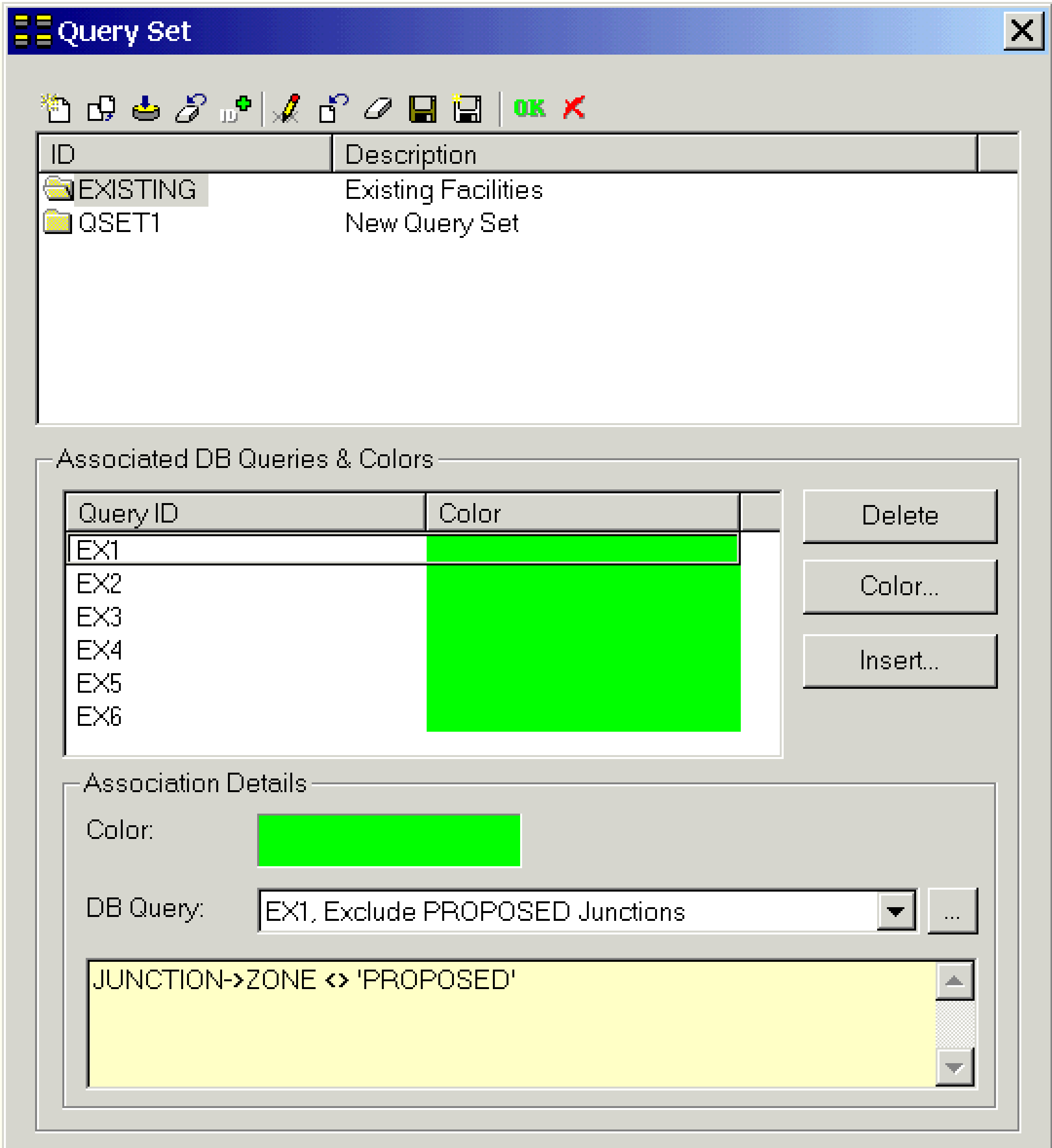


Other Related Topics - DB Query Methodology, Query Builder Dialog Box, Query Set Dialog Box, Query Sets Methodology.

Query Set Dialog Box

To create a query set, from the InfoWater **Table of Contents -> Operation** tab highlight the **Query Set** folder and right mouse click and select **New**. Provide a Query set ID and a description for the new query set. When created, the user is able to add or create the DB Queries that will comprise the Query Set. Click here to learn more about the Query Sets and their applicability in InfoWater.

Click on any portion below to learn more.

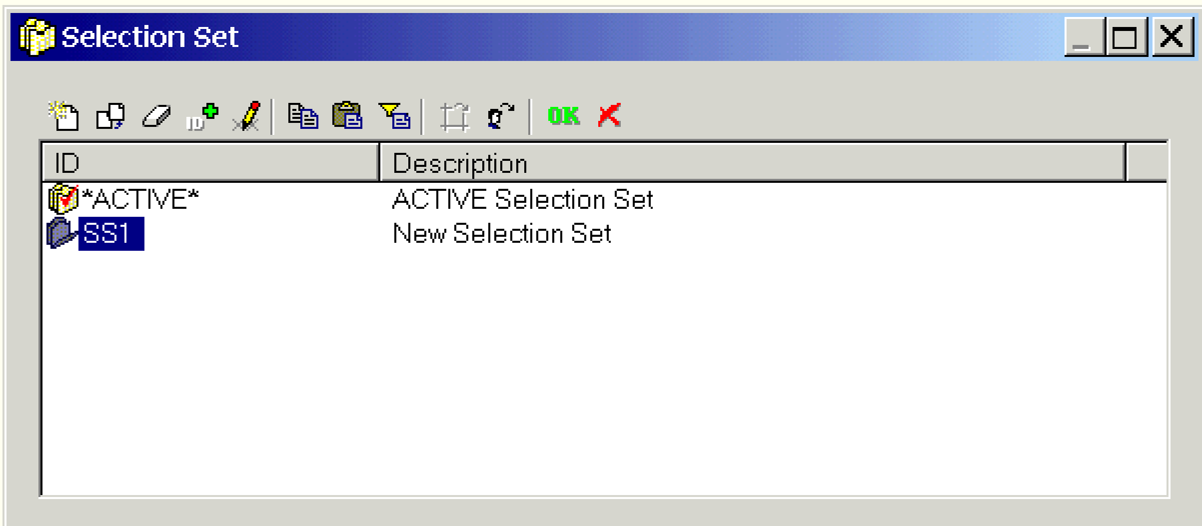


Other Related Topics - DB Query Dialog Box, DB Query Methodology, Query Builder Dialog Box, Query Sets Methodology.

Selection Set Dialog box

Selection sets are collections of data elements that can be saved and later recalled for activation. For more information on the Selection Set Methodology, click here.

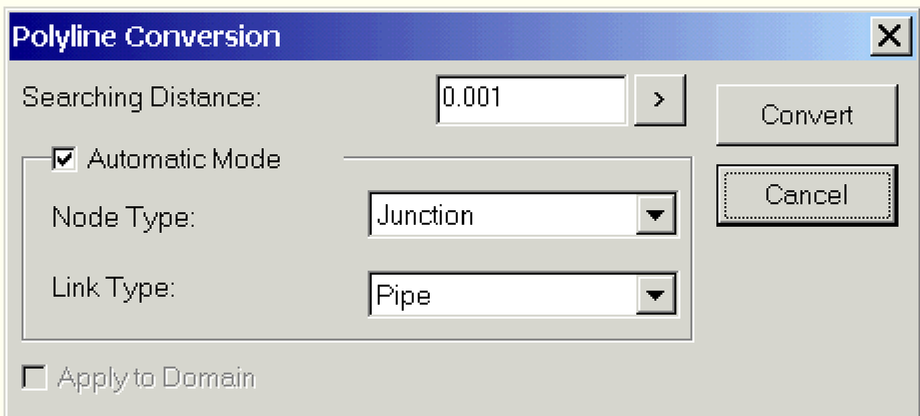
Click on any portion for more details:



Other Related Topics - Selection Set Methodology**.**

Convert Polyline

The convert polyline command is used to automate the InfoWater pipe and node network creation. If the user has a pipe network in another third party application like AutoCAD or Microstation, the network can be bought into InfoWater as a background layer (.dxf or .dwg) and then converted to an InfoWater pipe and node network. To learn more about the Conversion Methodology click here.



Other Related Topics - Convert Polyline Methodology, Data Source, EPANET Export, EPANET Import, EPANET v2.x Methodology, Export, Export Manager, Generate File Export, Generate File Import, Generate File Methodology, Import, Import and Export Overview, Import Manager, ODBC dialog box, ODBC Methodology, Select data Fields Export Process, Select data Fields Import Process, **OLE DB Connection**

Import and Export Overview

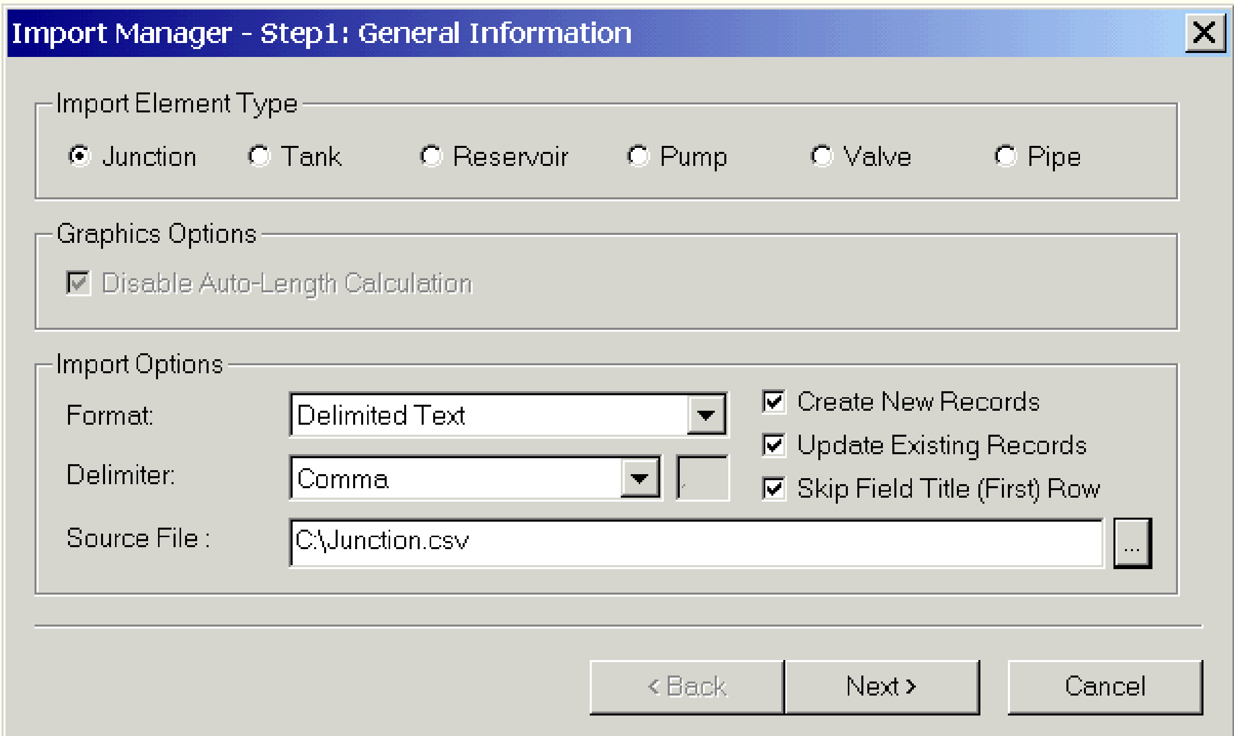
Having the ability to use external data is one of the many features of InfoWater. Likewise, having the ability to view model results outside of InfoWater is also important. Click on any of the links below to learn more about their capabilities.

- Import - Background mapping, external hydraulic models or InfoWater data elements.
- Export - InfoWater data elements, model results or an EPANET input file.
- ODBC - Download and upload data to and from InfoWater by creating data links.
- OLE DB Exchange - Download data from an external database.
- Convert Polylines - Allows the user to create a pipe and node network from an third party software application like AutoCAD, Microstation or ESRI shapefile.
- Import H2OMAP - Create an InfoWater project from a previous H2OMAP project.
- Import H2ONET - Create an InfoWater project from a previous H2ONET project.
- GIS Gateway - Allows exchange of Geodatabase information to and from the model.

Import Manager

Choosing this option launches the InfoWater Import Manager dialog box as shown below. Use this to import data from InfoWater.

Click for details.

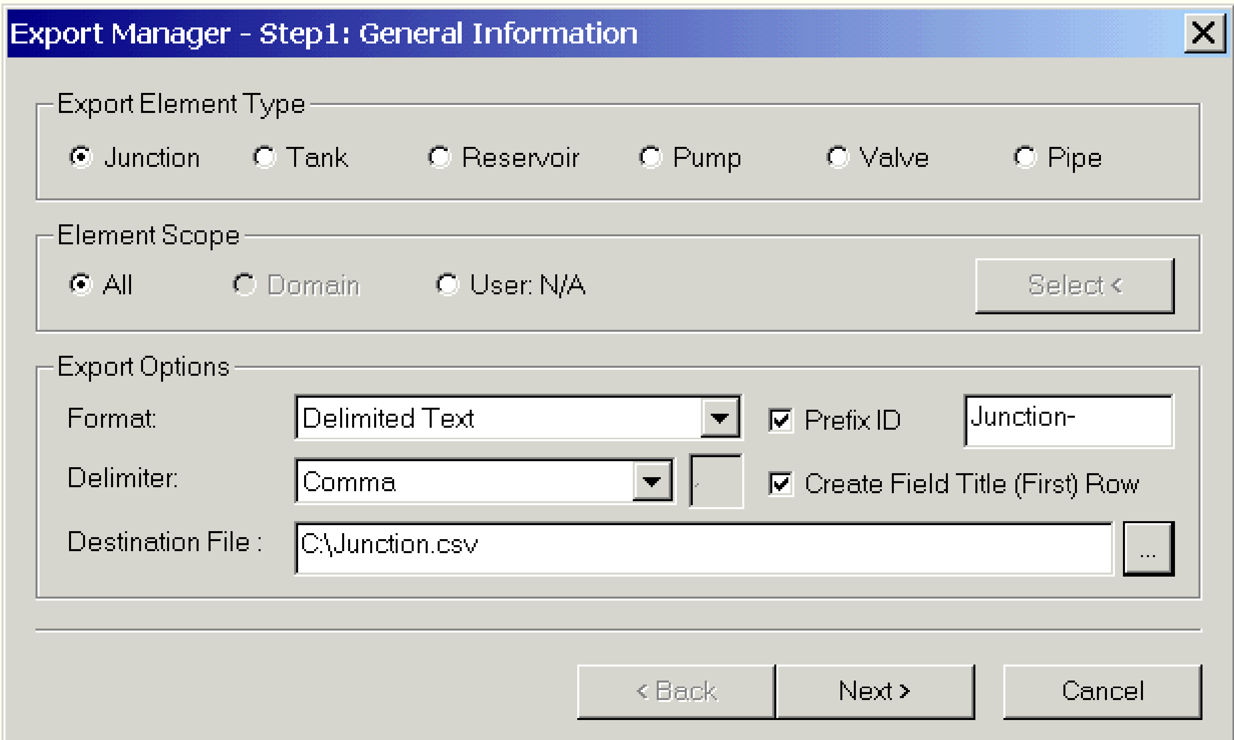


Other Related Topics - Convert Polyline Dialog Box, Convert Polyline Methodology, Data Source, EPANET Export, EPANET Import, EPANET v2.x Methodology, Export, Export Manager, Generate File Export, Generate File Import, Generate File Methodology, Import, Import and Export Overview, ODBC dialog box, ODBC Methodology, Select data Fields Export Process, Select data Fields Import Process, **OLE DB Connection**

Export Manager

Choosing this option launches the InfoWater Export Manager dialog box as shown below. Use this to export data from InfoWater.

Click on any portion to learn more.



Other Related Topics - Convert Polyline Dialog Box, Convert Polyline Methodology, Data Source, EPANET Export, EPANET Import, EPANET v2.x Methodology, Export, Generate File Export, Generate File Import, Generate File Methodology, Import, Import and Export Overview, Import Manager, ODBC dialog box, ODBC Methodology, Select data Fields Export Process, Select data Fields Import Process, **OLE DB Connection**

Scenario Manager

The Scenario Manger is where InfoWater allows the user to create, delete and modify scenarios. The Scenario Manager is what allows the user to create "what if" situations throughout a water system.

Components of an InfoWater scenario

Each of three components of a scenario can be further defined as follows:

- **Simulation set (General)** – Created through the General tab, they define the simulation options (durations, timesteps, type of water quality analysis, etc.) associated with the scenario. There are three different option set types, each storing a logical grouping of simulation options. Click here for more information on simulation sets. Click here for more information on simulation sets.
- **Facility set** – Defines the network facilities (components such as pipes, pumps, valves, junction and storage nodes) to be used in a simulation. Only one facility set can be active at a time (facility sets are created through the Facility tab). Click here for more information on facility sets.
- **Data set** – Stores modeling data (pipe diameter and roughness, nodal demands, etc.) associated with each facility in a separate external database. There are fourteen different data set types, each storing its own unique logical grouping of modeling data. Click here for more information on data sets.

When you define a scenario, you pick the facility, data, and option sets that comprise that scenario. When picking data sets for inclusion in a scenario, you may either specify that a data set associated with a given scenario is included in that scenario independent from other scenarios or alternately may specify that the given data set inherits its contents – properties – from a “parent” scenario.

Once you have configured and created a scenario, you can activate that scenario at any time. Once a scenario is activated, any modifications made to any of the databases related to InfoWater facilities will be changed, but only for the data sets that are related to, and dependent upon, the active scenario. Click here to learn how to activate a scenario.

Using Scenarios in InfoWater

Scenarios in InfoWater may be used for the following:

- Scenarios provides the means to store different facility sets (network layouts & network configurations), operating conditions (type of simulation, duration, operating conditions etc) and data sets (Modeling parameters such as demands, element attributes etc).
- Customizing the Scenarios allow for various Operator studies, modeling "What-if" scenarios, studying the model under different demand conditions, control sets (initial status and simple controls), logic controls, different pump, pipe, valve, junction, tank and reservoir configurations.
- Scenarios also allow you to create and run different fire-flow simulations, energy management analysis and water quality simulations.

Methodology

Do the following to Create a Scenario:

- Launch the Scenario Manager from the **InfoWater Control Center -> InfoWater** button **-> Scenario -> Scenario Manager** command or use the **InfoWater Control Center** Scenario Manager icon .

- Choose the Scenario that you want to create a "Child" of by clicking on that Scenario in the Scenario Display section of the Scenario Manager. Click here for more information on the Parent-Child inheritance mechanism in InfoWater.

- Click on the **New** icon (second from the left at the top of the Scenario Manager dialog box) to create a new scenario.

- Specify an ID and a Description for the Scenario (up to 20 characters, no spaces, no funny characters for the ID and 60 characters for the description) and click on **OK.**

- The New Scenario just created will contain the same Data sets as its parent. On the right window of the Scenario Manager, click on the Data set tab. Click on any Data Set that you want to customize.

- This will launch the Data Set Manager. Choose the Data set you want to clone and click on the clone icon . Specify an ID and Description as above to create the new cloned dataset. Alternatively you could have created a new dataset by clicking on the **New** icon .

- Choose the Newly Created Dataset and click on the OK button to associate it with the Scenario. For more information on the InfoWater datasets, click here.

- Now click on the **Facility** tab (next to the Data Set tab) to choose your Facility Set. Choose from among the different options available to define your Network Facilities. Click here for more information on Facility Sets.

- Once you have chosen your Data Sets and your Facility Sets, customize your Modeling Options by clicking on the **General** tab. For more information on the InfoWater Scenario Modeling Options click here.

- Once the Scenario customization and creation process is completed, click on the  button at the top of your Scenario Manager dialog box to save and exit out of your Scenario manager dialog box.

- Choose your New Scenario on the InfoWater Control Center, Scenario Activation select box. Choose OK on the confirmation box to activate your new scenario.

Note 1: Active Scenarios or their Parents cannot be customized. To customize the Facility sets, General Modeling Options and the Data Sets for active scenarios you would first need to de-activate the scenario i.e., have another Scenario active.

Note 2: The Base Scenario's options cannot be customized.

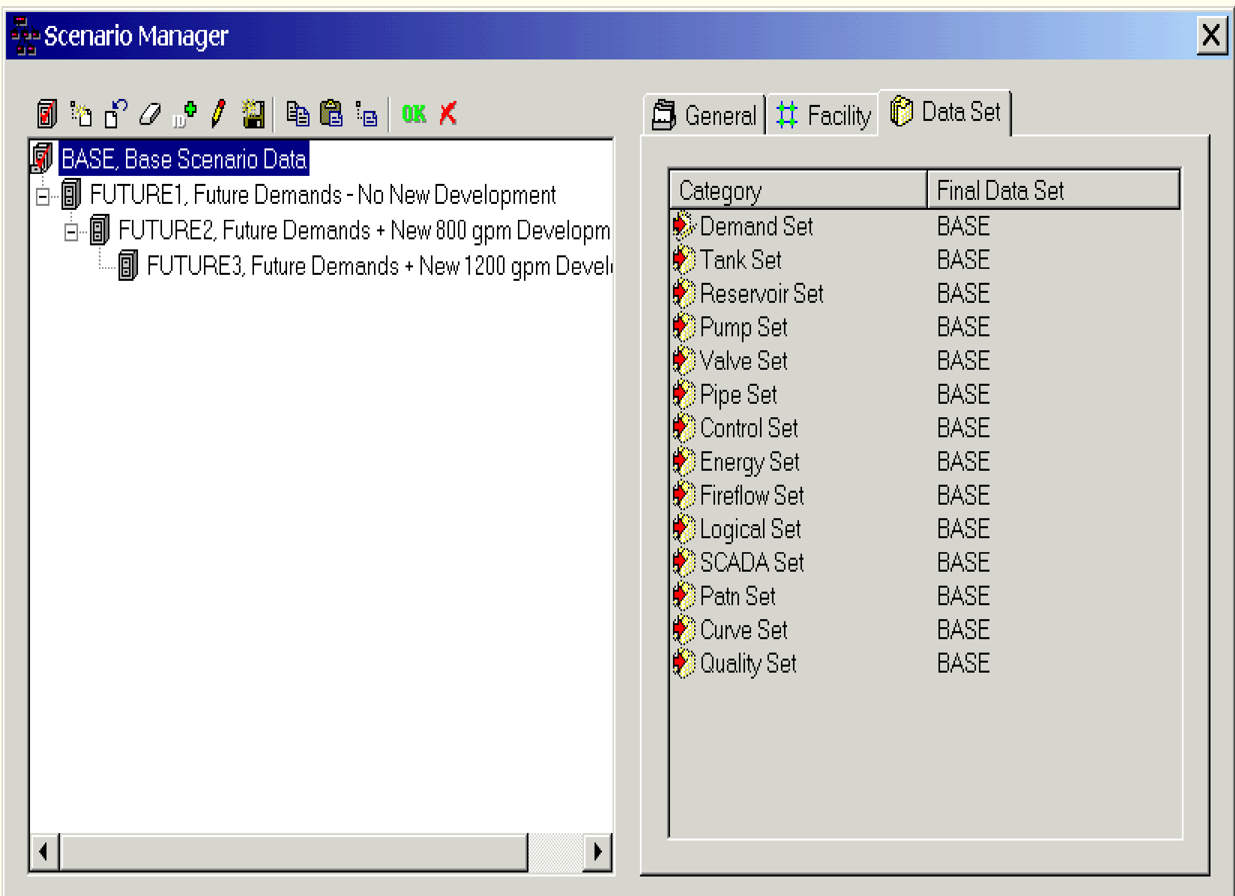
Other Related Topics - Active Scenario**,** Compare Scenarios**,** Compare Scenario Methodology**,** Comparison Report**,** Data Sets Methodology**,** Dataset Manager**,** Facility Sets Methodology**,** General Options Methodology**,** Scenario Manager Data Sets**,** Scenario Manager Facility Sets**,** Scenario Manager General**,** Scenario Manager Main Dialog Box

---

Scenario Manager Dialog Box

A scenario is a group of InfoWater facilities, data sets and conditions that are created to reflect a specific modeling situation. With a scenario, you can develop multiple models that are specific to your water distribution system (ex. Average day demand for a specific pressure zone with unique reporting options). Click here for more information on the Scenario creation Methodology.

Click on any portion below for more information.



Other Related Topics - Active Scenario, Compare Scenarios, Compare Scenario Methodology, Comparison Report, Data Sets Methodology, Dataset Manager, Facility Sets Methodology, General

Options Methodology, Scenario Manager Data Sets, Scenario Manager Facility Sets, Scenario Manager General, Scenario Methodology

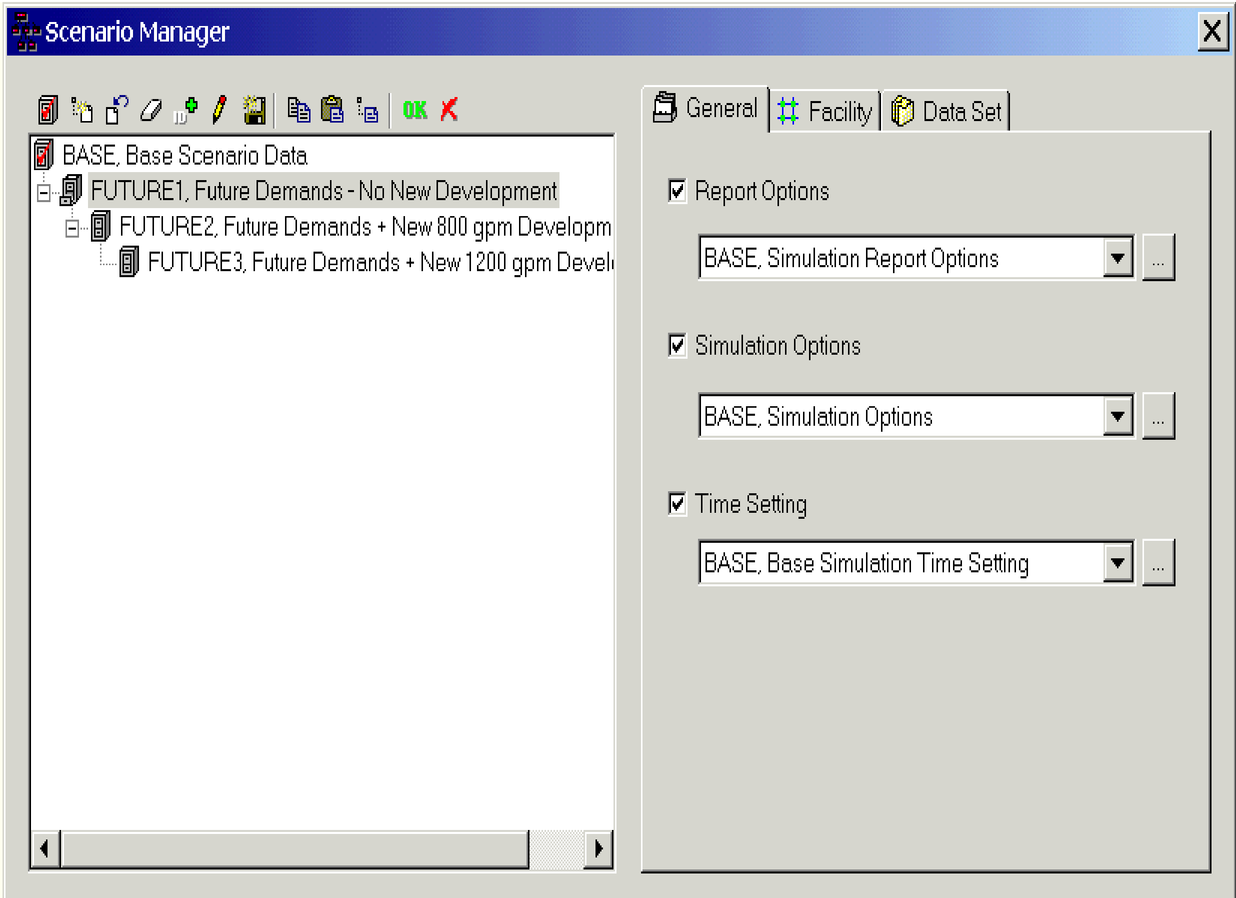
Scenario Manager General Options

The Simulation Set is defined by the following:

- **Report Option Set** – Standard reporting options associated with a scenario.
- **Simulation Option Set** – Simulation options associated with a scenario, including hydraulic and water quality modeling options, whether or not logical controls will be used during the next simulation run, etc.
- **Time Option Set** – Simulation duration and timesteps associated with a scenario.

Each scenario may be customized to have different simulation options such as the Type of simulation, the duration of simulation and other options pertaining to running of a simulation including the simulation report. For more details on the process of creating any of the option sets click here.

Click on any section below to learn more.



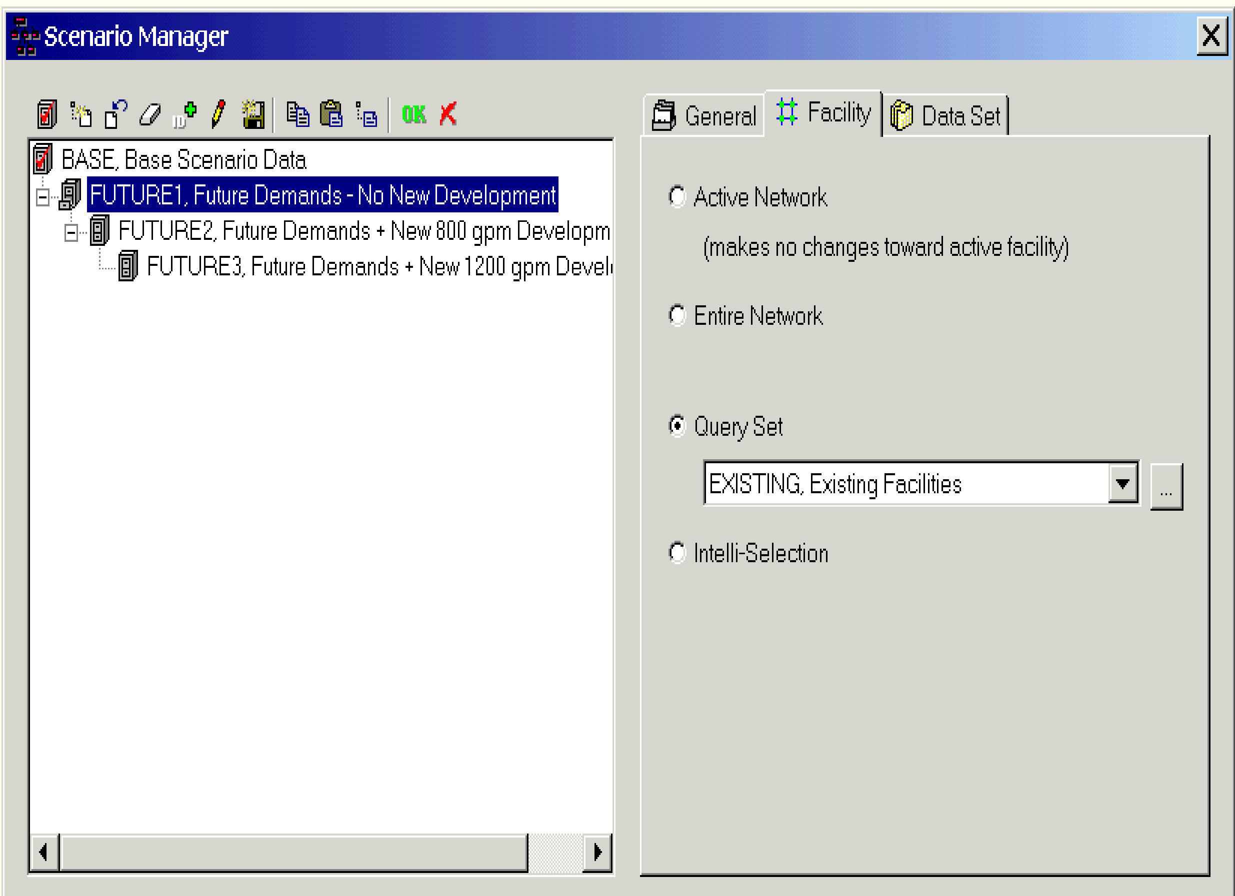
Other Related Topics - Active Scenario, Compare Scenarios, Compare Scenario Methodology, Comparison Report, Data Sets Methodology, Dataset Manager, Facility Sets Methodology, General Options Methodology, Scenario Manager Data Sets, Scenario Manager Facility Sets, Scenario Manager Main Dialog Box, Scenario Methodology

---

Scenario Manager Facility Sets

The facility set defines those network components (pipes, pumps, junctions, etc.) that will be considered during the next simulation. To model a subset of components, the user has many options to choose from. The facility set may include the entire network model or a subset of network components. Only facilities in the active facility set are displayed in InfoWater. Facilities that are not active (i.e., those not in the current facility set) are removed from the map display. Click here to read a detailed, step-by-step process for creating and associating facility sets with scenarios.

Click on any section for more information.



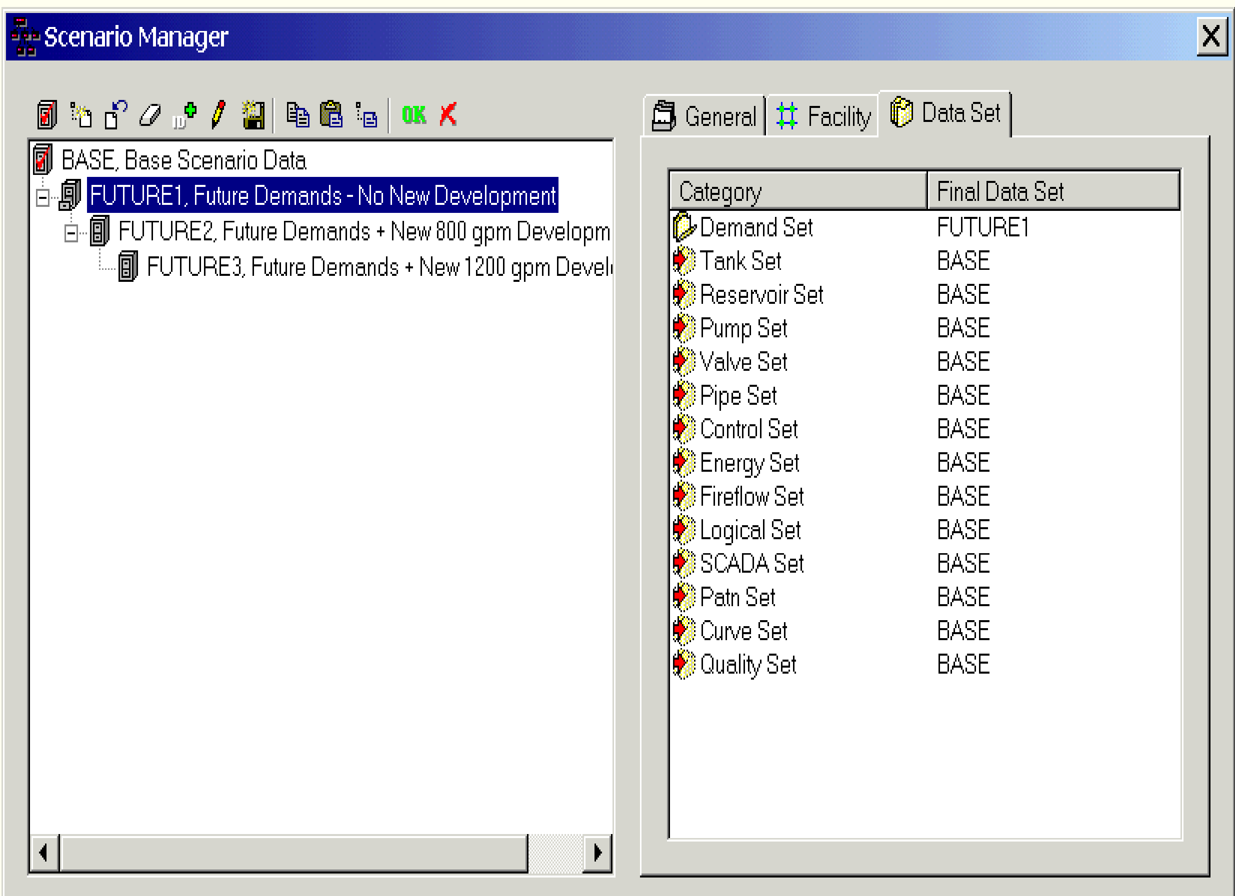
Other Related Topics - Active Scenario, Compare Scenarios, Compare Scenario Methodology, Comparison Report, Data Sets Methodology, Dataset Manager, Facility Sets Methodology, General Options Methodology, Scenario Manager Data Sets, Scenario Manager General, Scenario Manager Main Dialog Box, Scenario Methodology

Scenario Manager Data Sets

A data set is one of three components that comprise a scenario. Data sets provide the capability to take a one-time “snapshot” of data (database information) in the active model and store that information separate from the network itself.

In essence, InfoWater creates separate database tables for each data set created, allowing the user to manipulate the database characteristics of the data set, separate from the "Base" data set. Once new data is stored in a data set, it may be reloaded back into the active model (via a scenario) at any time. Click here for more information on the Data Sets.

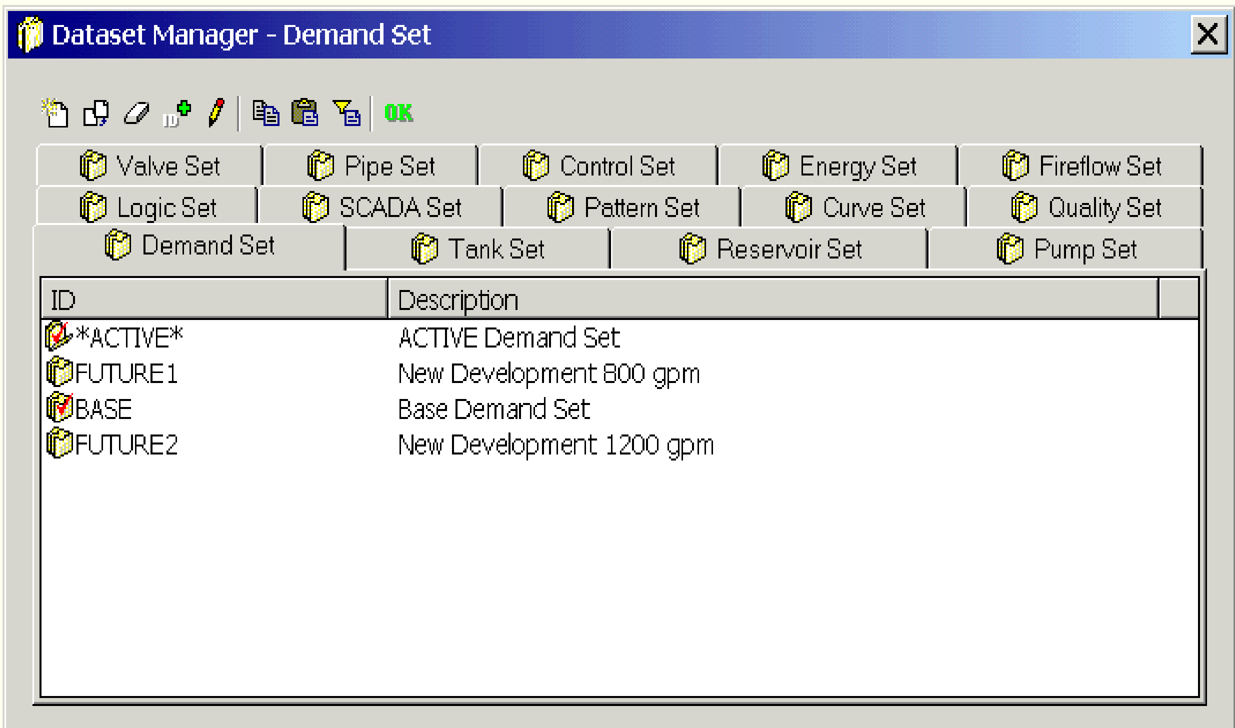
Click on any section for more information:



Other Related Topics - Active Scenario**,** Compare Scenarios**,** Compare Scenario Methodology**,** Comparison Report**,** Data Sets Methodology**,** Dataset Manager**,** Facility Sets Methodology**,** General Options Methodology**,** Scenario Manager Facility Sets**,** Scenario Manager General**,** Scenario Manager Main Dialog Box**,** Scenario Methodology

Dataset Manager

The Dataset Manager is a "quick view" dialog box that allows the user to create, view and edit data sets in one easy to use location. As shown below, the Dataset Manager is used by clicking on any of the tabs to view and edit the listed data set. Use any of the icons at the top of the dialog box to create, edit, clone, copy or delete data sets. Click here to learn more about data sets.

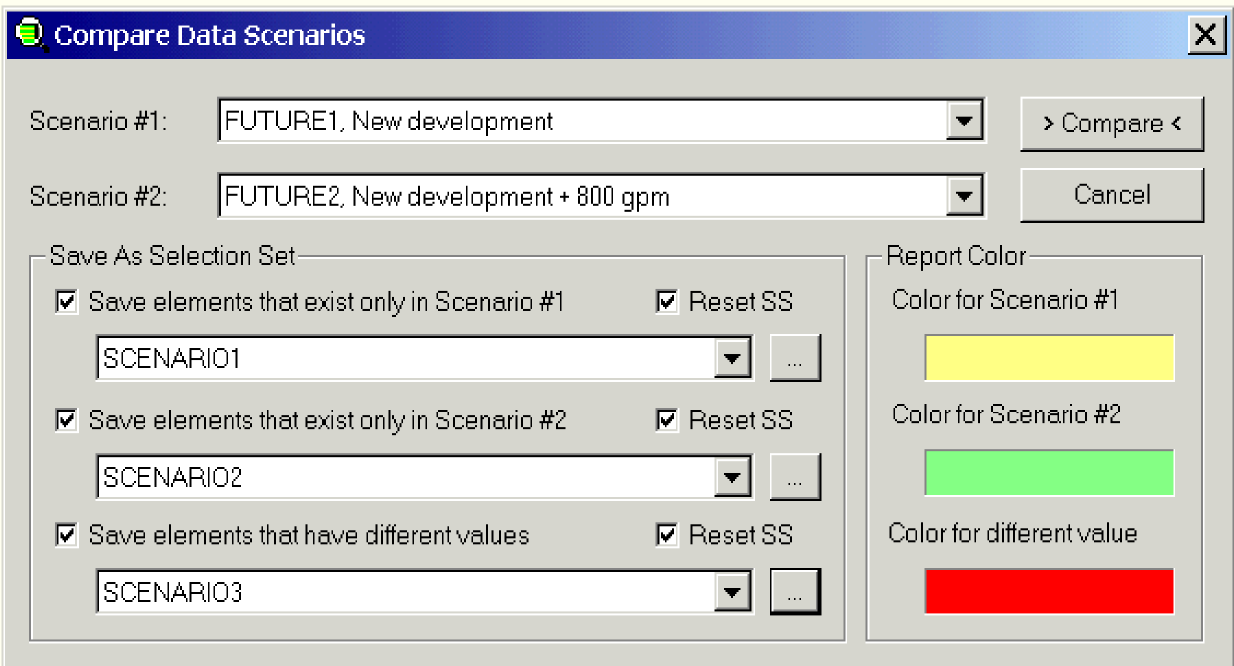


Other Related Topics - Active Scenario, Compare Scenarios, Compare Scenario Methodology, Comparison Report, Data Sets Methodology, Facility Sets Methodology, General Options Methodology, Scenario Manager Data Sets, Scenario Manager Facility Sets, Scenario Manager General, Scenario Manager Main Dialog Box, Scenario Methodology

Compare Data Scenarios

InfoWater's Compare Scenario command allows the user to monitor the differences between any two scenarios in a model.

To compare two scenarios, from the **Scenario** menu, select **Compare Scenarios**. The following dialog box is displayed. Click on any portion below to learn more.



Other Related Topics - Active Scenario, Compare Scenario Methodology, Comparison Report, Data Sets Methodology, Dataset Manager, Facility Sets Methodology, General Options Methodology, Scenario Manager Data Sets, Scenario Manager Facility Sets, Scenario Manager General, Scenario Manager Main Dialog Box, Scenario Methodology

Domains and Facilities

So what is the difference between a Domain and a Facility Set (the Domain Manager and the Facility Manager)?

Domain

A domain is a *temporarily* selected subset of network components. The domain can be used for a variety of purposes including group editing operations, mapping, contouring, etc. Domains are created and edited through the Domain Manager. Domains may also be used by InfoWater in conjunction with the Annotation, Contour and Map display features of InfoWater.

In essence, the Domain Manager is used as a quick editing feature for database and mapping purposes while the Facility Manager is used for modeling specific components prior to a simulation run. Only the Facility Manager controls the *Active* InfoWater network.

Facility Set

A facility set is used to create and maintain an active facility set (which may include the entire pipe and node network or any subset). The active facility set defines the network components in a current model that will be considered during a scenario and/or simulation run. Only the active facility set is displayed on the screen. Inactive components are hidden from the user until they become active again. Preferences may be altered by changing the **Inactive Element Color** specified under your **InfoWater Control Center ->** **InfoWater** button **-> Tools menu -> Project Preferences ->** **Display Settings** tab.

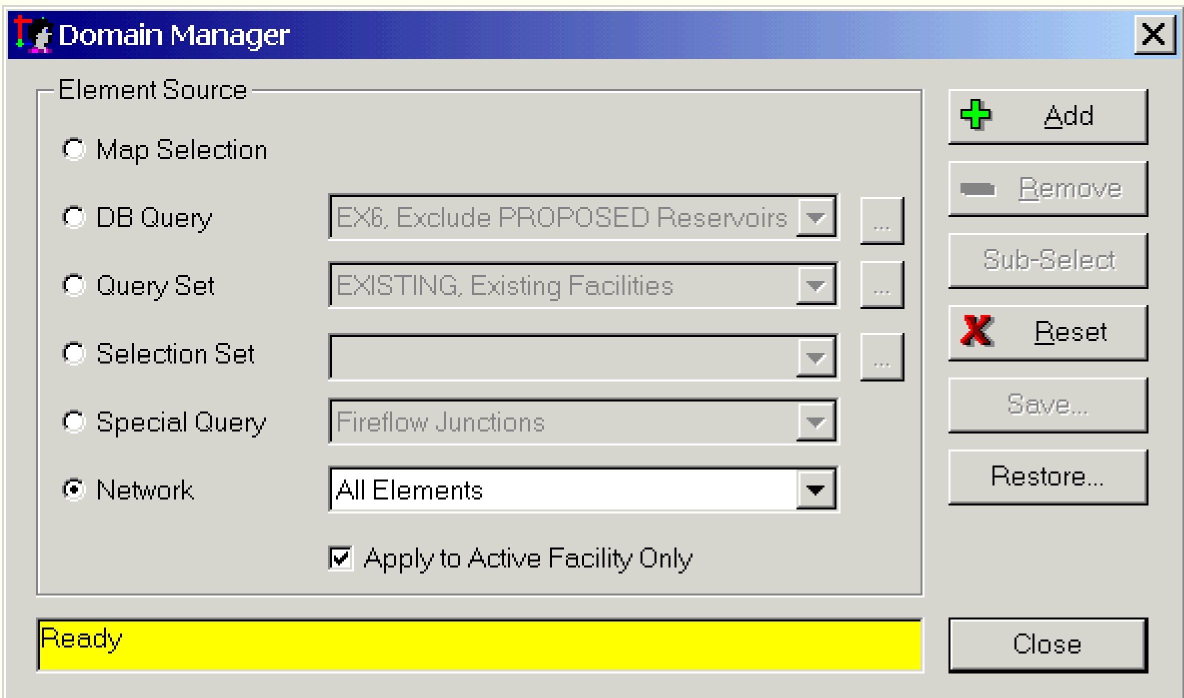
Facility sets are created and edited through the Facility Manager.

Other Related Topics - Domain Manager, Domain Methodology, Facility Methodology, Facility Manager.

Domain Manager

A domain is a *temporarily* selected subset of network components. The domain can be used for a variety of purposes including group editing operations, mapping, contouring, etc. For more information on the Domain Creating/Editing Methodology, click here.

From the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Domain Manager** in order to activate the Domain Manager as seen below. Click on any portion below to learn more.



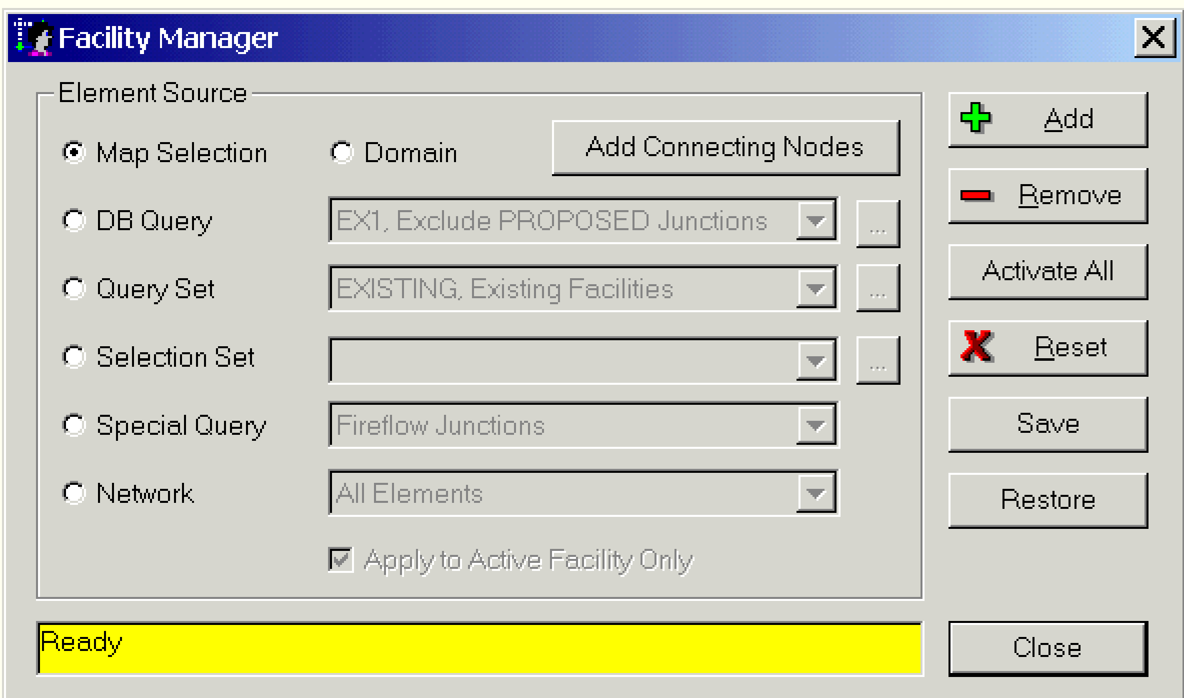
Note: The Run Manager disregards any selected domain during a model run. If you wish to model a subset of your network, use the Facility Manager to create a facility set rather than a domain.

Other Related Topics - Domain Methodology, Facility Methodology, Facility vs Domain, Facility Manager

Facility Manager

The Facility Manager is used to create and maintain the active facility set. The active facility set defines the network components in a current model that will be considered during the next simulation run(s). Facility sets can also be associated with a scenario via the Scenario Manager. To assign a facility set to a scenario, click here. For more information on the Facility Creating/Editing Methodology, click here.

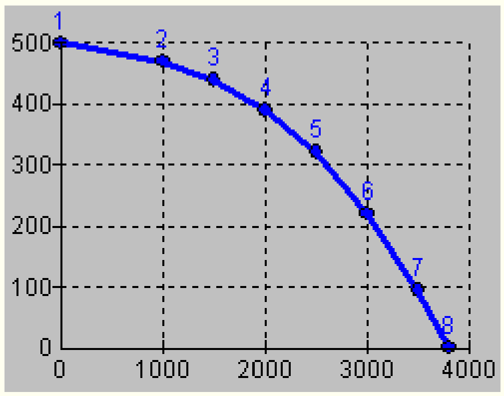
From the **InfoWater Control Center -> InfoWater** button -> **Tools** menu, select **Facility Manager** in order to activate the Facility Manager as seen below. Click on any portion below to learn more.



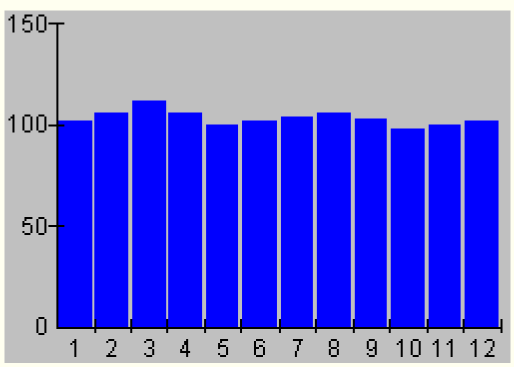
Other Related Topics - Domain Manager, Domain Methodology, Facility Methodology, Facility vs Domain.

Curves and Patterns

Curves - Used by InfoWater to relate one known variable vs. another (ex. Flow vs. Head for a pump). Curves are assigned to data elements through the **Attribute Browser** or through the **DB Editor** (database tables).



Patterns - A known variable vs. time (ex. Energy Rate vs. Time). Because patterns are time dependent, they only apply to **EPS** modeling. Patterns are assigned to data elements through the **Attribute Browser** or the **DB Editor** (database tables).



Other Related Topics- Curve Dialog Box, Curves, Pattern Dialog Box, Patterns.

Curves

Curves are objects that contain data pairs representing a relationship between two quantities. A curve is used by InfoWater in comparing one value against another (ex. Head vs. Flow). Curves are assigned to data elements through the **Attribute Browser.**

How do I...

- Create a Curve?
- Edit a Curve?
- Delete a Curve?
- Assign a Curve to an Entity?

An InfoWater model can utilize the following types of curves:

- Volume vs. Depth (Tanks - Variable-Area)
- % Efficiency vs. Flow (Pumps - Energy Cost Analysis)
- Headloss vs. Flow (Control Valves - General Purpose, Float)
- Minor Loss vs. % Open (Control Valves - Motorized Throttled)
- Head vs. Flow (Pumps - Multiple Point)
- NPSH vs. Flow (Pumps - Cavitation)
- Demand vs. Pressure (Junctions - Pressure Dependent Demand)

Each curve is identified by a unique identifier within InfoWater. It is advised that when defining curves, a detailed description is also provided to assist in curve selection.

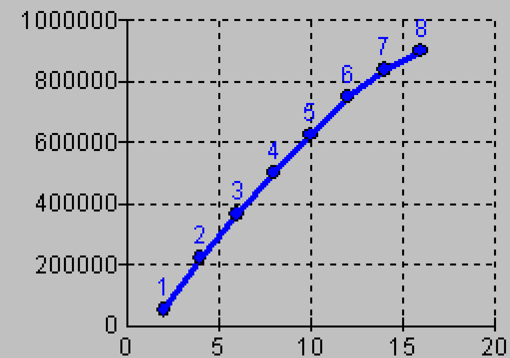
Types of Curves

InfoWater utilizes the following types of curves:

Volume vs. Depth

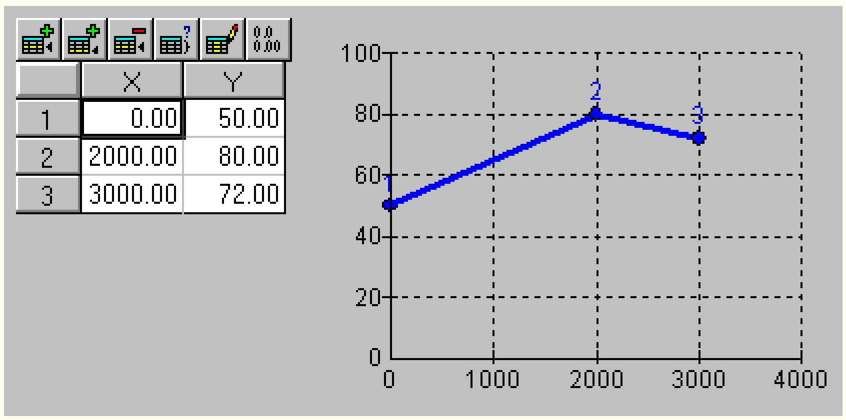
The curve for variable-area tanks is now defined as the volume of the tank versus its corresponding depth. It is very important to note that the volume must be defined in cubic feet (ft3) or cubic meters (m3) and not in any other unit (such as gallons or MG). This curve now relates to having the volume of the tank on the (Y-axis) with the corresponding depth on the (X-axis). A sample table and curve are provided below:

	X	Y
1	2	50000
2	4	220000
3	6	365000
4	8	500000
5	10	625000
6	12	750000
7	14	840000
8	16	900000



% Efficiency vs. Flow

Required for an Energy Management analysis, pump efficiency may be specified as a constant or as varying over a specified flow range. For pump efficiency curves, the curve consists of a collection of points defining the relationship between efficiency as a percentage (Y-axis) and flow rate in flow units (X-axis). A sample table and curve are provided below:

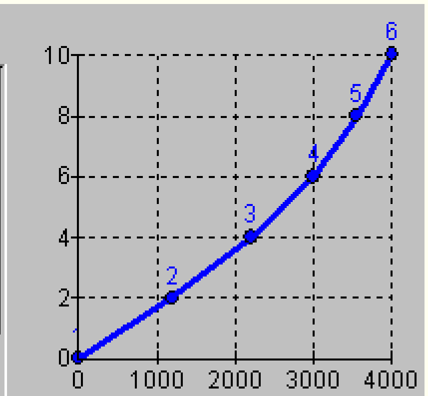


	X	Y
1	0.00	50.00
2	2000.00	80.00
3	3000.00	72.00

Headloss vs. Flow

Used for General Purpose Valves and Float Valves, the curve consists of a collection of points defining the headloss across the valve (Y-axis) and flow rate in flow units (X-axis). A sample table and curve are provided below:

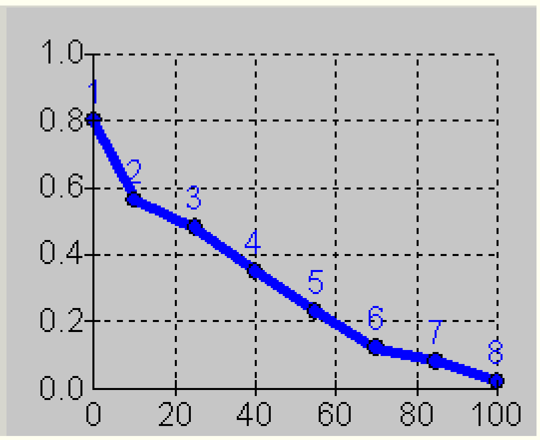
	X	Y
1	0.00	0.00
2	1200.00	2.00
3	2200.00	4.00
4	3000.00	6.00
5	3550.00	8.00
6	4000.00	10.00



Minor Loss vs. % Open

For Motorized Throttled Valves (MTVs), a Minor Loss Coefficient Curve consists of a collection of points defining the minor loss coefficient *K* (Y-axis) as a function of the percentage (degree) opening setting (X-axis). It provides the capability to model valves with unique headloss characteristics such as cone and butterfly valves. A sample table and curve are provided below:

	X	Y
1	0.00	0.80
2	10.00	0.56
3	25.00	0.48
4	40.00	0.35
5	55.00	0.23
6	70.00	0.12
7	85.00	0.08
8	100.00	0.02

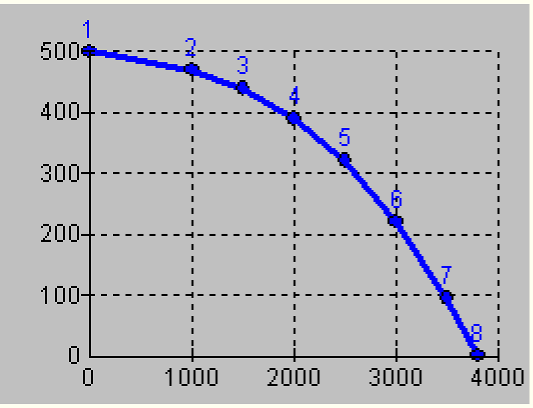


Head vs. Flow

A Pump Curve represents the relationship between the head and flow rate that a pump can deliver at its nominal speed setting. Head is the head gain imparted to the water by the pump and is plotted on the vertical (Y) axis of the curve. Flow rate is plotted on the horizontal (X) axis. A valid pump curve must have decreasing head with increasing flow.

When four or more data points are provided for the pump, the resulting head-flow curve is called a Multi-Point Pump Curve. It will be expressed as a piecewise linear curve of up to the last data point entered. This provides an accurate representation of the pump operation over the flow range specified. For pumps with multiple-point curves, the curve consists of a collection of points defining the head (Y-axis) as a function of flow rate (X-axis). A sample table and curve are provided below:

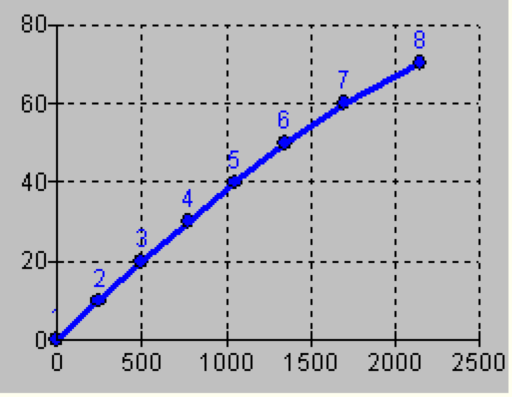
	X	Y
1	0.00	500.00
2	1000.00	470.00
3	1500.00	440.00
4	2000.00	390.00
5	2500.00	320.00
6	3000.00	220.00
7	3500.00	95.00
8	3800.00	0.00



NPSH vs. Flow

The required NPSH is evaluated from the multiple point curve (NPSH required vs flow) supplied by the user. This curve is based on performance tests and is normally available from the pump manufacturer. NPSH is defined as the total head at the suction side of the pump minus the vapor pressure head of the liquid being transported. A sample table and curve are provided below:

	X	Y
1	0.00	0.00
2	250.00	10.00
3	500.00	20.00
4	775.00	30.00
5	1050.00	40.00
6	1350.00	50.00
7	1700.00	60.00
8	2150.00	70.00

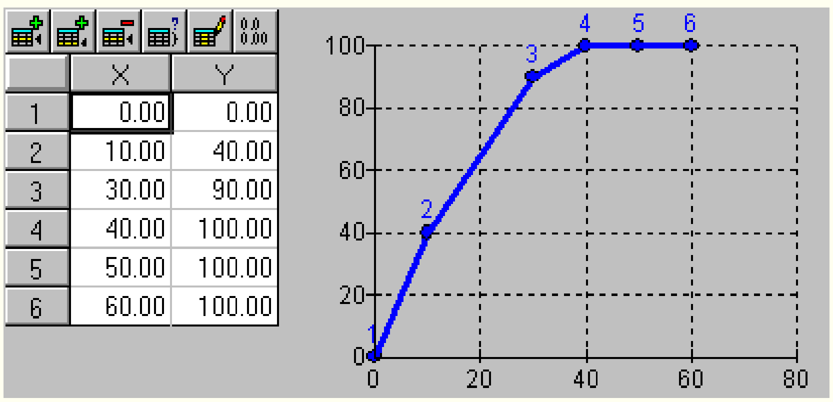


NOTE - If the required NPSH curve for a specific pump is not supplied by the user then InfoWater will only compute the available NPSH and the cavitation index will not be reported.

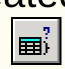
NOTE - If the suction piping diameter for a specific pump is not supplied by the user, then InfoWater will not compute the available NPSH.

Demand vs. Pressure

In certain situations, the user may wish to create a hydraulic model that more accurately reflects "real-world" conditions. As pressure drops, so too does demand. Knowing this, the user can assign a demand vs. pressure curve for selected junctions in the system. The Pressure-Demand Curve is used to describe how the demand flow rate (Y-axis) varies as a function of the pressure (X-axis) available at the node. The demand flow rate (Y-axis) is expressed as a percentage of the base demand with 100 equating to 100% of the base demand. The Demand vs. Pressure curve provides the capability to model situations where pressure-dependent demands are significant in a water distribution system :



Create a Curve

To create a curve, from the **View** menu select **Table of Contents -> Operation** tab, select and highlight **Curve** and right-click on your mouse and choose the **New** button. At this point the user is asked to provide a name and description for the new curve. When created, the curve dialog box will appear. Click on the **Set Rows** icon  to specify the number of curve points. For instance to plot the above curve (Demand vs Pressure) 6 rows were specified. Finally enter the desired X, Y coordinates of the new curve.

Edit a Curve?

From the curve dialog box, the user can change any X,Y value as necessary to reflect the relationship between the two values (ex. more flow at the same head). The user can also edit the graph directly by selecting either the X or Y field to the left and then clicking on the graph. With the X or Y field highlighted, the user can adjust the graph accordingly.

Delete a Curve?

To delete a curve, click on the appropriate curve (This curve should be listed under **Curve** in your **Table of Contents -> Operation** tab) right-click on your mouse and choose **Delete**.

Assign a Curve to an Entity

For example, to assign a curve to a pump, using the **Select** tool  on your **Attribute Browser**, highlight the pump and Under the **Modeling** section of the **Attribute Browser** click on the **Curve** cell. Using the down arrow button select the desired curve.

Other Related Topics - Curve Dialog Box**,** Curves and Patterns**,** Pattern Dialog Box**,** Patterns**.**

Patterns

A pattern is a graph (assigned to an element) - used to represent temporal (time-dependent) variations within the system. Whereas a curve is used to compare one value against another (Head vs. Flow), a pattern is always a known variable vs. time. Because patterns vary over time, they are to used for **Extended Period Simulation** analysis.

How do I...

- Create a Pattern?
- Edit a Pattern?
- Delete a Pattern?

An InfoWater model can utilize the following types of patterns:

- Water consumption vs. time (Junctions)
- Water quality vs. time (Junctions, Tanks and Reservoirs)
- Hydraulic grade vs. time (Reservoirs)
- Energy rate vs. time (Pumps)

Each pattern consists of a collection of multipliers that are applied over a sequence of consecutive time periods during the course of the simulation.

Pattern time steps are controlled with the Simulation Time command.

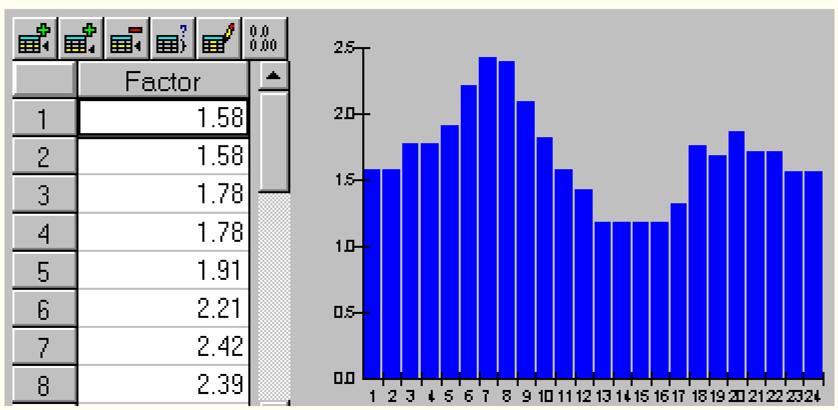
Types of Patterns

InfoWater utilizes the following types of patterns:

Patterns for Nodes

Demand Patterns (for Junctions only) - These are applied by selecting a junction node, then viewing the list of available patterns in the **Pattern** Cell under the **Modeling** section of the **Attribute Browser** . From here the user then selects one of the patterns .

Water Quality Patterns (for Junctions, Tanks and Reservoirs) - To assign a water quality pattern to a group of nodes (Junctions, Tanks and Reservoirs only), use the Group Editing (**Domain** or **Selection**) option under the **InfoWater button -> Edit** Pull down menu. Once these elements are selected, the user is provided the Group Editing dialog box where the option to create a Time Pattern for the water quality is provided under the Quality Source tab.

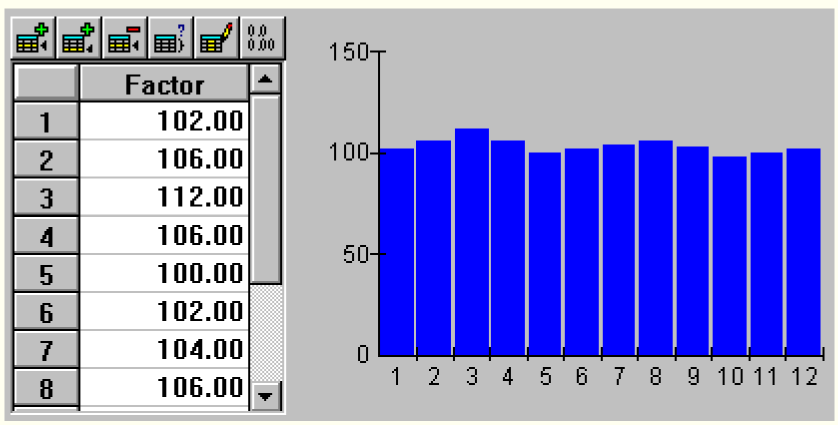


Note: This diurnal water demand pattern has 24 time steps. Under the Simulation Time option under the **Operation** tab of the **Table of Contents**, the user would specify a Pattern Timestep of 1-hour for a 24-hour simulation. If this pattern had 12 time steps, the user would specify a Pattern Timestep of 2-hours for a 24-hour simulation. If the duration of a pattern (say 24 time steps) is less than the duration of the simulation (say 5 days), then the pattern is repeated as necessary.

Patterns for Reservoirs

Supposing a water system has a reservoir with a variation in elevation head or pressure head throughout the day - a pattern can be assigned to that reservoir. In every other case for a pattern (except a variable head reservoir), the pattern is used as a multiplier to the base data. With a variable head reservoir, the pattern itself represents the water surface elevation of the reservoir over a given time period (see graph).

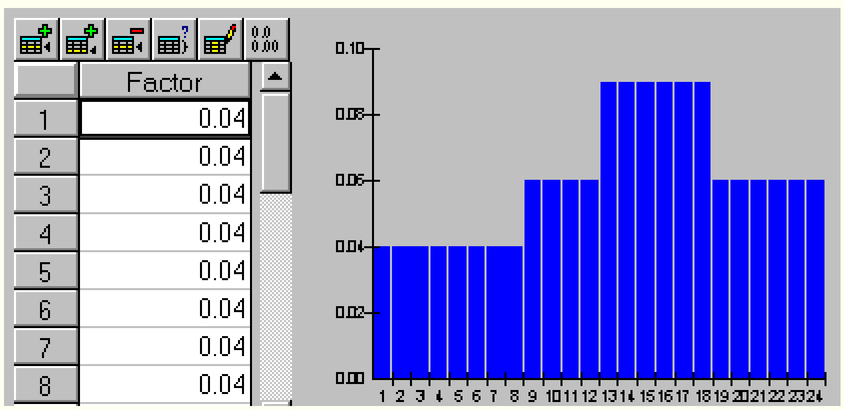
To assign a pattern to a reservoir, highlight the reservoir and view its attributes under the **Modeling** section of the **Attribute Browser**. Under the modeling section, select the **Pattern** drop down box and choose the appropriate pattern to assign to the reservoir.



Patterns for Pumps

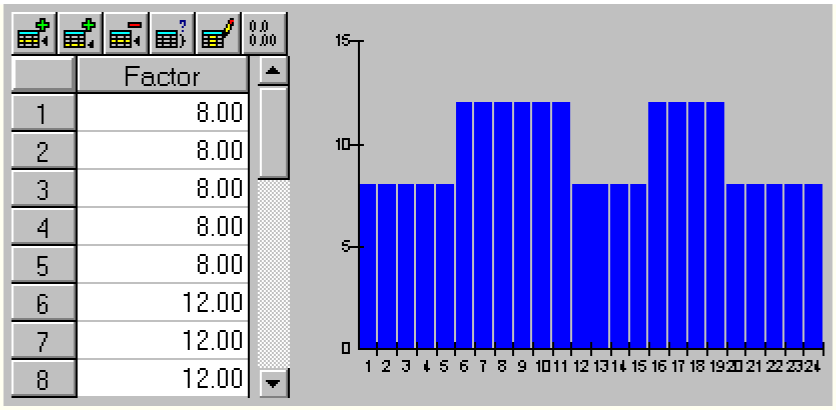
Energy Rate Pattern

When performing an Energy Management simulation, the user can specify an energy consumption pattern (which represents the cost of electricity over a 24-hour period) and a demand charge pattern (representing energy penalties). To assign an energy pattern to a pump, select the pump and then choose the **Tools** icon  under the **Attribute Browser** window. Select the **Energy Pattern** option which enables the user to specify the appropriate pattern for the energy analysis.



Demand Charge Patterns (Optional)

Demand (or capacity) charging rates are specified as time-series patterns in InfoWater. Each pattern factor defines a one-time charge for maximum power usage within each billing period ($/max. kW). A “billing period” is defined as the time span with identical charging factors in a demand charge pattern. A billing period may encompass several non-contiguous spans of time:



In the example above, two billing periods are specified. The first billing period runs from midnight to 6:00 AM, from noon to 4:00 PM, and from 8:00 PM to midnight. The second billing period runs from 6:00 AM to noon and from 4:00 PM to 8:00 PM.

Each pattern factor represents the demand charge at a given hydraulic timestep. The X-axis represents time period and the Y-axis represents the demand charge.

Pattern Representation Options

Between time-steps, two options are available for representing usage factors between regular fixed periods in time:

- *Stepwise* – InfoWater assumes a constant usage factor from one fixed period of time to the next.
- *Continuous* – InfoWater linearly interpolates variations in the pattern factor between fixed time periods.

To change the type of demand pattern used during a simulation, go to the **Operation** tab of the **View** menu **-> Table of Contents** dialog box, select **Simulation Options** and double-click on the **Base Simulation Options** command (if you have different simulation options listed then choose the one corresponding to the appropriate scenario), then go to the **Demand** Tab and change the **Default Demand Type**. Click here to see the Simulation Options dialog box.

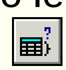
Multiple Demand Patterns

With InfoWater, the user can easily and efficiently model multiple demand patterns for any or all junction nodes. By assigning multiple demand patterns to a junction node, the user can model the unique diurnal demand patterns associated with various demand classifications such as residential planned community (RPC), commercial (C-1, C-2, etc.), industrial (I-1, I-2, etc.), and so on.

For each junction node, the user can assign numerous baseline demands (e.g., one for each land use classification) and optionally, demand patterns containing multipliers can be applied to each baseline demand. Multiple baseline demands and demand patterns can be assigned by using the select tool, selecting the Junction and under the **Attribute Browser** window selecting the Modeling tab. Here the user can populate up to **10** demands and patterns for each analysis junction node.

When running a simulation, InfoWater will combine all of the demands associated with a junction node in an additive fashion (at each hydraulic timestep in an extended period simulation) to accurately model the total demand exerted on the system at each junction node.

Create a Pattern

To create a pattern, highlight **Pattern** under the **Operation** tab of the **Table of Contents** dialog box. Right-click on your mouse and choose the **New** icon. From here the user enters a unique ID and description for the pattern and selects OK. The Pattern dialog box is now presented. Click here to learn more about this box and its icons. Click on the **Set Rows** icon  to specify the number of pattern multipliers. For instance to plot the Energy Rate Pattern above 24 multipliers were specified. Finally enter the desired multiplier values.

Edit a Pattern

By choosing the **Pattern** option under the **Operation** tab under the **Table of Contents** dialog box, the user is able to see all patterns currently created in InfoWater. To edit a pattern, double click on the desired pattern (or right-mouse click to edit) and the pattern dialog box will be enabled.

Delete a Pattern

By choosing the **Pattern** option under the **Operation** tab of the **Table of Contents** dialog box, the user is able to see all patterns currently created in InfoWater. To delete a pattern, right-mouse click and select delete or hit the delete key on your keyboard. At this point the user will be prompted to confirm deletion of the pattern.

Other Related Topics - Curve Dialog Box**,** Curves**,** Curves and Patterns**,** Pattern Dialog Box**.**

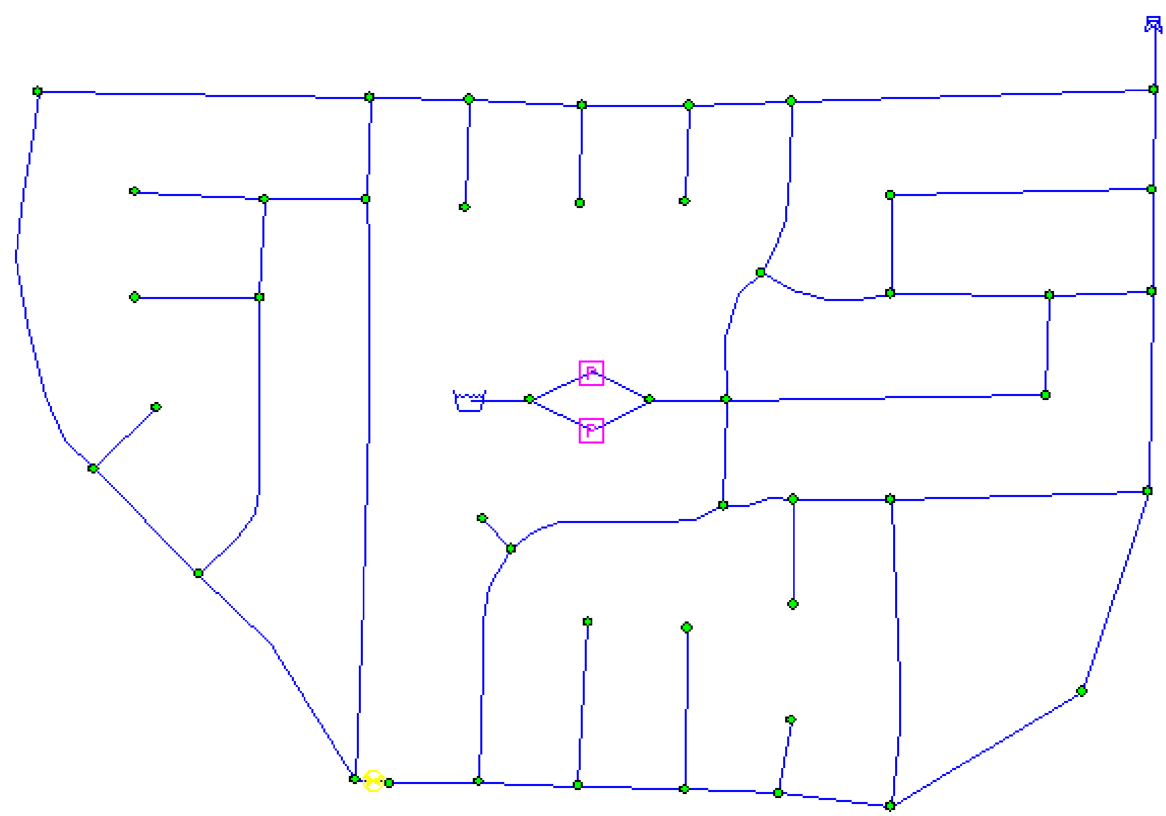
Printing and Presentation

The user has the option of adding text anywhere in the InfoWater project for presentation purposes. Adding a legend for plotting is done through the Print Preview function. Click on the following to learn more:

- To make a plot of the InfoWater project, use the Print Preview function.
- For reports, clicking the print icon on the Output Report Manager will display the Print Grid dialog box.
- For graphs, clicking the print icon on the Output Report Manager will display the Print Graph dialog box.
- You may also choose to print graphs through the System option.

Print Preview

Print Preview may be used to preview your entire InfoWater project. Use the InfoWater **File** menu **-> Print Preview** command to review your InfoWater project drawing and then click on **File -> Print** to print it.

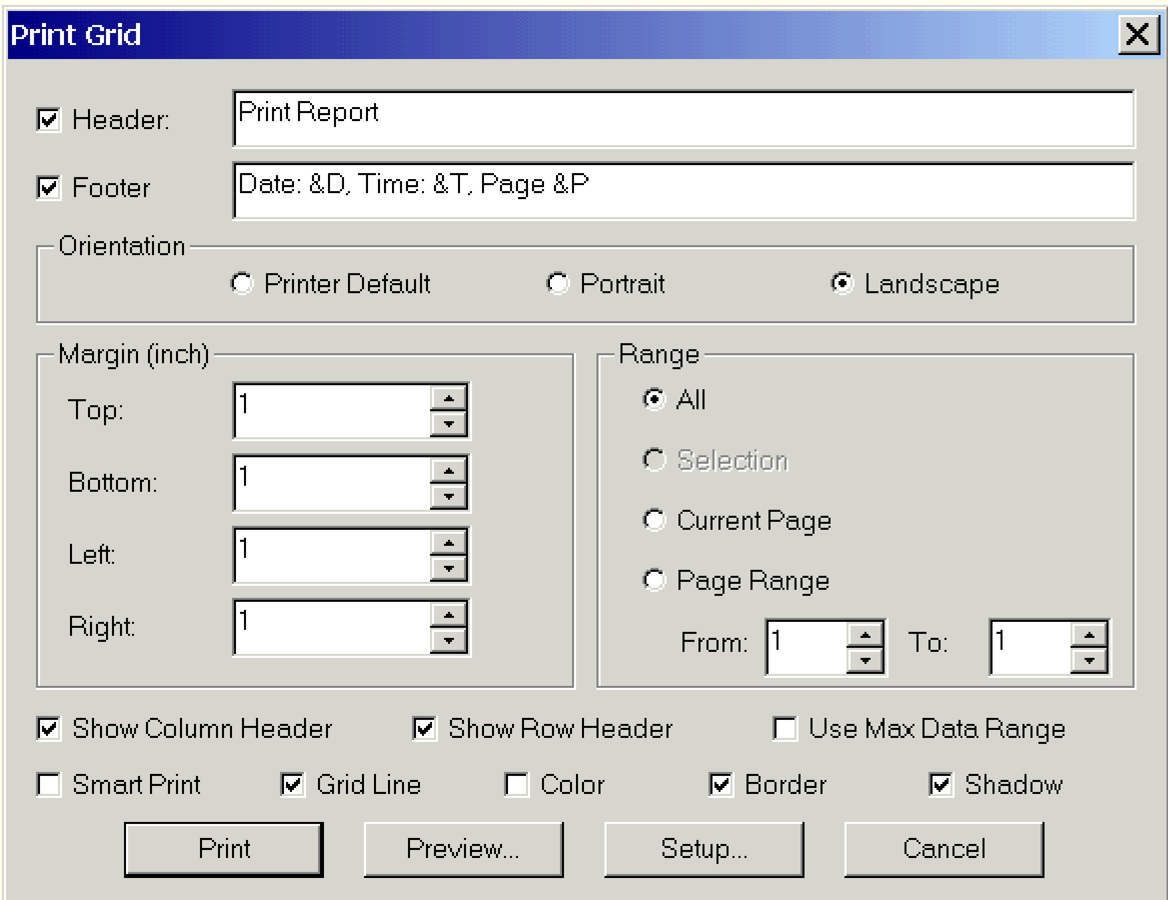


Other Related Topics - Graph Settings Methodology, Print Graph, Print Grid, Printing and Presentation, System Button

Print Grid

The print grid is available from the print icon found on any output report, customized report or query report. The options for the print grid are shown below the following graphic.

Click on any section to learn more.

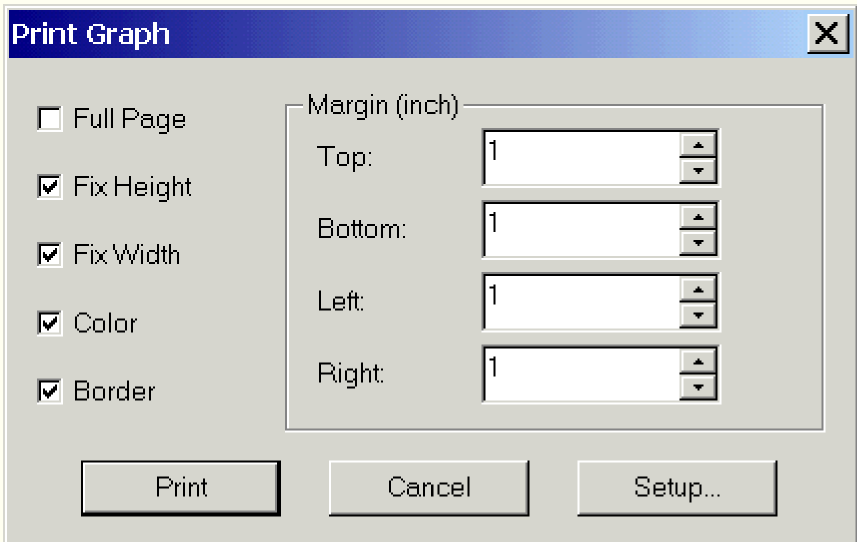


Other Related Topics - Graph Settings Methodology, Print Graph, Printing and Presentation, System Button, Print Preview

Print Graph

The print graph dialog box is available from the print icon found on any output graph.

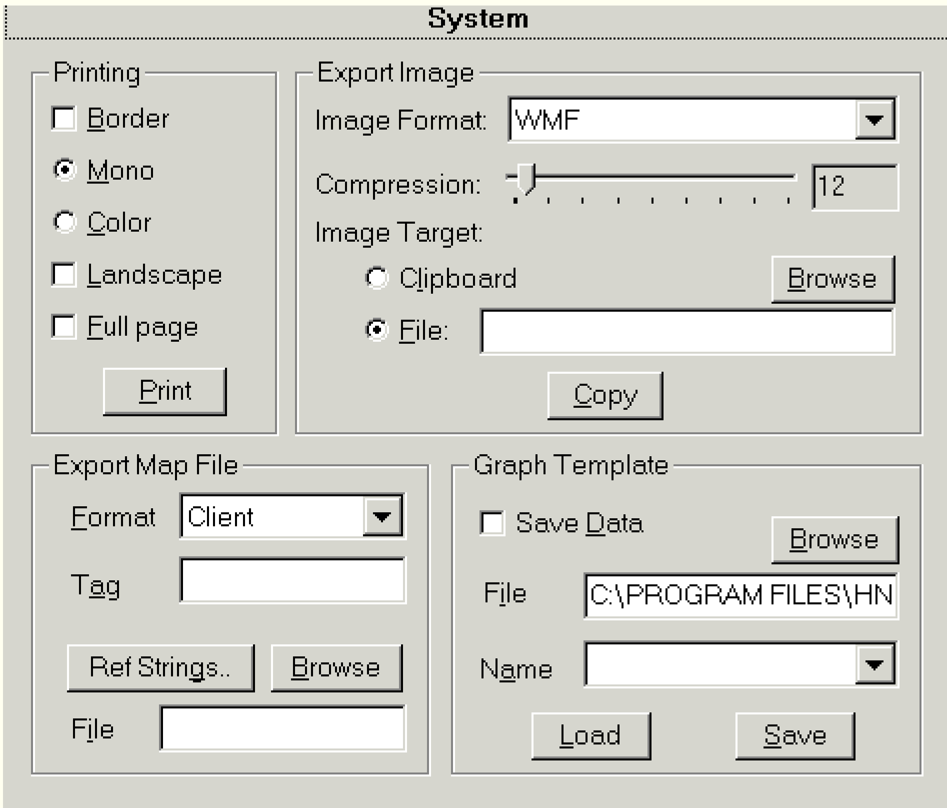
Click on any portion to learn more.



Other Related Topics - Graph Settings Methodology, Print Grid, Printing and Presentation, System Button, Print Preview

System Icon

By selecting the **System** icon from the **Output Report Manager** dialog box, the dialog box below appears. This dialog box may be used to specify your Print options, Graph Export and Copy options and your Graph template option. Click here to learn more about the **Graph Template** Setting option and how it can be used in InfoWater.



Other Related Topics - Graph Settings Methodology, Print Graph, Print Grid, Printing and Presentation, Print Preview

InfoWater Theory

InfoWater represents the state-of-the-art in water supply and distribution systems analysis. The program provides scores of cutting edge simulation capabilities for performing a wide variety of essential modeling tasks in record time. This section provides a comprehensive description of the theory and methodology behind the hydraulic and water quality modules used by InfoWater. InfoWater utilizes an enhanced version of the EPANET analysis engine as developed and distributed by the U.S. Environmental Protection Agency, Risk Reduction Engineering Laboratory (EPANET 2000).

This section describes how InfoWater models the physical objects that constitute a distribution system as well as its operational parameters and the computational methods used to simulate hydraulic and water quality transport behavior.

Capabilities

- **Building Distribution System Models** - InfoWater was designed to greatly facilitate and automate many of the data compilation and management duties in network modeling. Through its powerful and intuitive menu-driven interface, the user can construct an accurate representation of the distribution system, including a map of the system, attributes required for analysis, and other user-defined attributes.
- **Creating Hydraulic and Water Quality Simulations** - Upon building an appropriate representation of the distribution system, the user may perform several different types of hydraulic and water quality network simulations.
- **Analyzing Model Results** - Upon completion of each simulation, the user can view, query, and display analysis results using sophisticated graphical presentation tools including

dynamic map annotation/labeling, color-coding, graphing, contouring, profiling, customizable tabular reporting, and vivid VCR-style animation to produce truly compelling results.

- **Integrating Information with Other Applications** - InfoWater is compatible with other Windows, GIS and CAD-based applications. Data is easily exchanged with other applications including presentation applications, spreadsheets, relational databases, mapping, and geographic information systems.

Methodology

This section describes how InfoWater models the physical objects that constitute a distribution system as well as its operational parameters and the computational methods used to simulate hydraulic and water quality transport behavior. To learn more choose any of the options below:

- **Physical Components**
- **Non Physical Components**
- **Hydraulic Simulation Model**
- **Water Quality Simulation Model**
- **Fireflow Simulation Model**
- **System Head Curves**
- **Energy management Model**
- **SCADA Interface**

References

InfoWater Capabilities

In the past few years, advances in infrastructure management technology have been occurring at an accelerated pace. This technology is now being realized within the context of water distribution system modeling. It has allowed water distribution system modeling applications to move from a traditionally non-graphic, non-intuitive environment to one that embraces today's standards for easy-to-use, fully interactive graphical productivity software.

InfoWater provides

four major capabilities for the modeling and analysis of water distribution systems:

- **Building distribution system models** - InfoWater was designed to greatly facilitate and automate many of the data compilation and management duties in network modeling.

  Through its powerful and intuitive menu-driven interface, the user can construct an accurate representation of the distribution system, including a map of the system, attributes required for analysis, and other user-defined attributes.

- **Creating hydraulic and water quality simulations** - Upon building an appropriate representation of the distribution system, the user may perform several different types of hydraulic and water quality network simulations.

- **Analyzing model results** - Upon completion of each simulation, the user can view, query, and display analysis results using sophisticated graphical presentation tools including dynamic map annotation/labeling, color-coding, graphing, contouring, profiling, customizable tabular reporting, and vivid VCR-style animation to produce truly compelling results.

- **Integrating information with other applications** - InfoWater is compatible with other Windows and CAD and GIS-based applications. Data is easily exchanged with other applications including presentation applications, spreadsheets, relational databases, mapping, and geographic information systems.

InfoWater is developed

to run entirely within the ArcGIS for Windows environment. InfoWater tools are accessed using customized ArcMap commands, menus, and toolboxes, and the graphical user interfaces for these tools are based exclusively on ArcMap and Windows interface standards.

Methodology

InfoWater represents the state-of-the-art in water supply and distribution systems analysis. The program provides scores of cutting edge simulation capabilities for performing a wide variety of essential modeling tasks in record time. This section provides a comprehensive description of the theory and methodology behind the hydraulic and water quality modules used by InfoWater. InfoWater utilizes an enhanced version of the EPANET analysis engine as developed and distributed by the U.S. Environmental Protection Agency, Risk Reduction Engineering Laboratory (EPANET 2000).

This section describes how InfoWater models the physical objects that constitute a distribution system as well as its operational parameters and the computational methods used to simulate hydraulic and water quality transport behavior. To learn more choose any of the options below:

- **Physical Components**
- **Non Physical Components**
- **Hydraulic Simulation Model**
- **Water Quality Simulation Model**
- **Fireflow Simulation Model**
- **System Head Curves**
- **Energy management Model**
- **SCADA Interface**

NOTE: In InfoWater both pumps and control valves are treated as nodes, which is different from the link terminology used by EPANET.

Frequently Asked Questions

This section addresses some of the common questions that you might have pertaining to the InfoWater model creation, data elements, result analysis and project maintenance.

How do I **start an InfoWater session** ?

Click on the InfoWater icon on your Desk top to start InfoWater or alternatively launch InfoWater from your **Start -> Programs -> InfoWater Version X -> InfoWater**. Choose from the three options specified. Choose A new empty map to open a new project, A template to launch your template project or the An existing map: command to open an existing map. If you choose this option then choose the project that you want to open. Finally initialize your InfoWater project by clicking on the InfoWater initialize icon on your InfoWater Edit Network toolbar.

How do I specify a coordinate system for an InfoWater project?

Refer to the section Create New InfoWater Project to learn how to specify a coordinate system for an InfoWater project.

What does an **InfoWater project** consist of ?

An InfoWater project consists of a drawing file and a database folder. For instance a project named Model will contain a *Model.mxd* (ArcMap drawing file) and a *Model.IWDB* (InfoWater Database). The mxd file contains your network map while the IWDB folder contains all the data associated with your model.

After a successful model simulation another folder called *Model.Out* is created. This folder contains all the output data related to the InfoWater project Model. It is important to note that the Database folder (*Model.IWDB*), the drawing file (*Model.mxd*) and the Out folder

(*Model.Out*) need to reside within the same parent folder such as say **C:\Data**.

What are the different **Data Elements** in InfoWater and how do I create and modify them ?

Refer to the section on Data Elements

What are **Junction Nodes** & how do I create them ?

Refer to the section on Junction Nodes to learn more.

What are the different types of **Pumps** in InfoWater ? How do I create and edit them ?

Refer to the section Pumps to learn more.

How are **Storage** nodes modeled in InfoWater ?

Refer to the section on Storage Nodes for details.

What are **Valves** ? How will they be represented in my Model ?

Refer to the section on Valves for more details

How are **Pipes** modeled in InfoWater ? Tell me more about them.

Refer to the section on Pipes for details.

How do I **digitize my model quickly**?

Use the Network creation process.

How do I **select a sub set of my model** to display results, modify data etc ?

Use the Domain feature of InfoWater to select a sub set of your model for display and data edit purposes. The Domain Methodology section explains all the different stages in the Domain Creation process.

How do I **activate a sub set** of my model for analysis? What are **Facility Sets** ? Can you explain the Facility Set creation process ?

Facility sets may be used to activate a portion of your mode for your network analysis. Refer to the section on the Facility Methodology to learn about creating Facility Sets.

How are Facility Sets different from Domains ?

Facility vs Domain section of the help file highlights the difference between facility sets and domains.

How do I **edit data related to a selected group** of elements ?

Use the Group Edit option to modify data related to a group of elements in InfoWater. Refer to the section on Group Editing Methodology to learn more about the process and how it can be used in InfoWater.

Teach me more about **Curves**.

Refer to the section on Curves

Teach me more about **Patterns**.

Refer to the section on Patterns

I want to create **Database Queries**. Please explain the procedure.

The DB Query Methodology section explains the process and how you can use it to query your InfoWater Database.

Can you explain the process of creating **Query Sets** and how they are different from DB Queries ?

Refer to the section on the Query Sets Methodology to learn more.

Can I save a domain in InfoWater? How do I create **Selection Sets** and how can I use them in InfoWater ?

Use Selection Sets to save domains for later retrieval. A step by step process for creating selection sets is included in the Selection Set Methodology section. Also advantages and applications of Selection Sets in InfoWater is explained.

How do I create **Output Relates** ? What are the uses of Output Relates ?

Refer to the section on the Output Relate Methodology to learn the Output Relate creation process.

How can I use an **Output Relate** with my **DB Query** ? Can I query my Output results ?

Refer to the section Query Output relate Methodology for step by step instructions on creating Output Relates and using them.

I want to create a **Customized Report** but I don't know how to create it.

The Customized Report methodology section shows a detailed schema for creating Customized Reports.

How do I create a **Query Report** in InfoWater ?

Refer to the section on the Query Report Methodology to learn more.

Can you explain the process to create **Query Summation Reports** in InfoWater ?

The Query Summation Report Methodology contains instructions to create Query Summation Reports in InfoWater.

How do I look at my InfoWater **Reports** ?

Refer to the Output Report Methodology section to learn more.

How do I **Compare Data** in InfoWater ?

The How to Compare Output Data section contains the different methods and tools available in InfoWater to compare data.

What is a **Graph Settings Template** File ? How do I use it to create graph templates ?

Refer to the section Graph Settings Methodology. This section will explain the process of creating templates and loading them in InfoWater.

I want to set **Initial Controls** on my Data Elements. How do I go about it ?

Use the Initial Status Methodology section to learn more about setting initial status on your different InfoWater data elements.

What are **Rule Based Controls** and how do I create them ?

The details and the creation process is explained in Rule Based Control Methodology section of the InfoWater Help file.

Can you help me create and assign **Simple Controls** to my project ?

Refer to the section on the Simple Controls Methodology to learn more about creating simple controls and using them in InfoWater.

How do I conduct an **Energy Management Simulation** ?

The Energy Management Modeling section will walk you through the different steps required to perform an Energy management simulation in InfoWater.

What is an **Extended Period Simulation** and How do I run an EPS model in InfoWater ?

The EPS Modeling section will help you create and run an EPS model.

Can InfoWater run **Fire Flows** and if so how ?

For detailed step by step fire flow model creation and run process refer to the section on Fire Flow Methodology.

What is a **Hydrant Curve** and how do I conduct a Hydrant Curve analysis ?

Refer to the Hydrant Curve Methodology section to learn more.

Can I run a **Pressure Dependent Demand** analysis in InfoWater ? Can you give me some more information on that ?

The Pressure Dependent Demand Methodology section of the InfoWater help file will walk you through the different steps necessary to run a Pressure Dependent Demand analysis.

I want to **run all my scenarios together**. How do I do it ?

The Batch Run Manager can be used to run all your scenarios at the same time. Refer to the section on the Batch Run Methodology to learn more.

How do I **run** simulations in InfoWater ?

The Run Manager Methodology section will give you step by step instructions on "running" a model in InfoWater.

I want to use the **Real Time Connection** command but don't know how to do it ?

The Real Time Data Connection Methodology explains the procedure and uses of the Real Time Data Connection command in InfoWater.

How do I conduct a **SCADA** analysis?

Refer to the section on SCADA Integration to learn more.

Please explain the steps necessary to run a **Steady State Model** in InfoWater ?

The Steady State Modeling Methodology explains the different steps required to run a steady state model in InfoWater.

Can I create **System Curves** in InfoWater ?

Refer to the section on the System Curve Methodology to learn more.

What are **Variable Speed Pumps** and how do I model them in InfoWater ?

The Variable Speed Pumps Methodology section will walk you through the process of creating VSPs and using then in InfoWater.

Can you help me run a **Water Quality Simulation** in InfoWater ?

The Water Quality Simulation Methodology section will walk you through the process of running a Water Quality Model.

How do I **Compare Scenarios** in InfoWater ?

Refer to the section on Compare Scenarios Methodology to learn more.

How do I create **Data Sets** in InfoWater ?

The Data Sets Methodology section explains the data set creation process in detail.

What are **Facility Sets** and how do I use them in InfoWater ?

Refer to the section on the Facility Sets Methodology to learn more.

Can I customize the **Simulation Options** for my scenarios and if so how ?

The General Options Methodology section contains step by step instructions to help you customize your Simulation options.

How do I create **Scenarios** in InfoWater ?

Refer to the section on the Scenario Creation Methodology to learn more.

Can I create a model from my **Polylines** ?

Use the Convert Polyline Methodology section to learn the procedure for creating pipes and nodes from your AutoCAD polylines.

How do I export/Import to/from an **EPANET** file ?

The EPANET v2.x Methodology section contains step by step instructions for you to Export or import to or From an EPANET file.

What are **Generate Files** and how do I bring them in to my InfoWater project ?

The Generate File Methodology section contains step by step instructions for you to Export or import to or From a Generate file.

I would like to use the InfoWater **ODBC** command but don't know how ?

The ODBC Methodology section will help you understand and use the Open Database Connection tool of InfoWater to import or export to a third party data source.

How can I create **animations** in InfoWater ?

Refer to the Animation Viewer Methodology section to learn more. This section contains detailed instructions to help you create animations and view them in InfoWater.

How do I **color code my Data Elements** in InfoWater ?

Use the Map Display Command to color code your Data Elements. For step by step instructions refer to the Map Display Methodology section of the InfoWater help file.

Can I create **annotations** in InfoWater ? Can you help me create them ?

Refer to the section on Annotation Methodology to learn the annotation creation process.

Can I create **contours** in InfoWater ? Can you help me create them ?

The Contour Methodology section contains step by step instructions that will walk you through the contour creation process.

I would like to work with **Geodatabases**. Can you explain the procedure ?

Refer to the section on Working with ESRI Geodatabases to learn more.

How do I model a **groundwater pumping well**?

Represent the well as a reservoir whose head equals the piezometric head of the groundwater aquifer. Then connect your pump from the reservoir to the rest of the network. You can add piping ahead of the pump to represent local losses around the pump. If you know the rate at which the well is pumping then an alternate approach is to replace the well - pump combination with a junction assigned a negative demand equal to the pumping rate. A time pattern can also be assigned to the demand if the pumping rate varies over time.

How do I **size a pump** to **meet a specific flow**?

Set the status of the pump to CLOSED. At the suction (inlet) node of the pump add a demand equal to the required pump flow and place a negative demand of the same magnitude at the discharge node. After analyzing the network, the difference in heads between the two nodes is what the pump needs to deliver.

How do I **size a pump** to **meet a specific head**?

Replace the pump with a Pressure Breaker Valve oriented in the opposite direction. Convert the design head to an equivalent pressure and use this as the setting for the valve. After running the analysis the flow through the valve becomes the pump's design flow.

How can I **enforce a specific schedule of source flows** into the network from my reservoirs?

Replace the reservoirs with junctions that have negative demands equal to the schedule of source flows. (Make sure there is at least one tank or remaining reservoir in the network, otherwise InfoWater will issue an error message.)

How do I model a **reduced pressure backflow prevention valve**?

Use a General Purpose Valve (GPV) with a headloss curve that shows increasing head loss with decreasing flow. Information from the valve manufacturer should provide help in constructing the curve.

Place a check valve (i.e., a short length of pipe whose status is set to CV) in series with the valve to restrict the direction of flow.

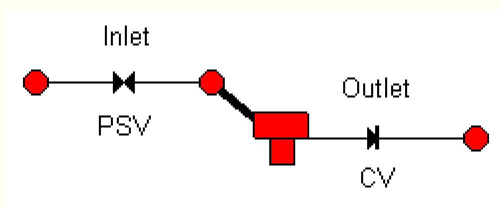
How do I model a **pressurized pneumatic tank**?

If the pressure variation in the tank is negligible, use a very short, very wide cylindrical tank whose elevation is set close to the pressure head rating of the tank. Select the tank dimensions so that changes in volume produce only very small changes in water surface elevation.

If the pressure head developed in the tank ranges between H1 and H2, with corresponding volumes V1 and V2, then use a cylindrical tank whose cross-sectional area equals (V2-V1)/(H2-H1).

How do I model a **tank inlet that discharges above the water surface**?

Use the configuration shown below:



The tank's inlet consists of a Pressure Sustaining Valve followed by a short length of large diameter pipe. The pressure setting of the PSV should be 0, and the elevation of its end nodes should equal the elevation at which the true pipe connects to the tank. Use a Check Valve on the tank's outlet line to prevent reverse flow through it. There should be another small length of pipe (a dummy pipe) separating the tank and check valve.

How do I determine **initial conditions** for a **water quality analysis**?

If simulating existing conditions monitored as part of a calibration study, assign measured values to the nodes where measurements were made and interpolate (by eye) to assign values to other locations. It is highly recommended that storage tanks and source locations be included in the set of locations where measurements are made.

To simulate future conditions start with arbitrary initial values (except at the tanks) and run the analysis for a number of repeating demand pattern cycles so that the water quality results begin to repeat in a periodic fashion as well. The number of such cycles can be reduced if good initial estimates are made for the water quality in the tanks. For example, if modeling water age the initial value could be set to the tank's average residence time, which is approximately equal to the fraction of its volume it exchanges each day.

How do I estimate values of the **bulk** and **wall reaction coefficients**?

Bulk reaction coefficients can be estimated by performing a bottle test in the laboratory. Wall reaction rates cannot be measured directly. They must be back-fitted against calibration data collected from field studies (e.g., using trial and error to determine coefficient values that produce simulation results that best match field observations). Plastic pipe and relatively new lined iron pipe are not expected to exert any significant wall demand for disinfectants such as chlorine and chloramines.

How can I model a **chlorine booster station**?

Place the booster station at a junction node with zero or positive demand or at a tank. For the selected node in the Source Concentration Data dialog box, set Source Type to SETPOINT BOOSTER and set Baseline Source Strength to the chlorine concentration that water leaving the node will be boosted to. Alternatively, if the booster station will use flow-paced addition of chlorine then set Source Type to FLOW PACED BOOSTER and

Baseline Source Strength to the concentration that will be added to the concentration leaving the node. Specify a time pattern ID in the Time Pattern field if you wish to vary the boosting level with time.

How would I model **THM growth** in a network?

THM growth can be modeled using first-order saturation kinetics. Select Options - Reactions from the Data Browser. Set the bulk reaction order to 1 and the limiting concentration to the maximum THM level that the water can produce, given a long enough holding time. Set the bulk reaction coefficient to a positive number reflective of the rate of THM production (e.g., 0.7 divided by the THM doubling time). Estimates of the reaction coefficient and the limiting concentration can be obtained from laboratory testing. The reaction coefficient will increase with increasing water temperature. Initial concentrations at all network nodes should at least equal the THM concentration entering the network from its source node.

Error Messages & Warnings

After a model run, errors or warnings may be encountered. When you run a simulation in InfoWater, the simulation may complete successfully. In these cases the Run Status Indicator (stop light) on the Run Manager dialog box shows green. In other cases, the simulation may complete, but several warnings are issued. In other cases, one or more errors are present in the model input data preventing InfoWater from properly compiling required data to run a simulation or several errors occur during the simulation.

If the simulation completed with warnings: **Warning Messages**

If the simulation failed with errors: **Error Codes**

If a simulation fails with warnings or errors, you may want to have InfoWater generate a standard output report as it runs the simulation. This report lists important simulation options and parameters and can report on system performance at each reporting timestep. This report can be used to help you debug any problems. Standard output report options are specified with the Standard Report Manager command prior to running the simulation.

Contact Us

You can reach MWH Soft, Inc. by phone between 8:00 A.M. and 5:00 P.M. Pacific Standard Time, Monday through Friday. InfoWater technical support can be reached at:

Phone: (626) 568-6868

Fax: (626) 568-6870

Email: MWHSoft@mwhsoft.com : Support@mwhsoft.com : Sales@mwhsoft.com

Internet: http://www.mwhsoft.com

Or write us at MWH Soft, Inc., 300 North Lake Avenue, Suite 1200, Pasadena, California 91101 USA. We will seriously consider your suggestions for future versions of InfoWater.

For international support, please contact your local MWH Soft agent.

We will occasionally create interim updates that contain fixes and/or new features. To get to our home page, point your web browser to our internet address listed above.

Note - Do not forget to renew your Annual Maintenance Agreement (Subscription Program) to take full advantage of future enhancements, product upgrades and product updates.

Domain Create/Edit Methodology

The Domain Manager is used to create, edit and delete elements contained within a domain.

How Do I...

- Create a Domain?
- Clear a Domain?

Using Domains in InfoWater

Domains in InfoWater may be used for the following:

- For assigning values globally "on the fly" through the Group Editing on Domain option for conducting Fire-Flows, Water quality analysis etc.
- Domains may also be created to select a sub-set of the network for contouring, annotating etc.
- Restricting the database records that are available for editing using the Database Editor.
- Creating Selection Sets (see Selection Sets).
- Restricting model results to identify network components that are chosen as a part of a domain

Methodology

Create Domain

To create a domain, do the following:

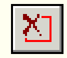
- Launch the **InfoWater Control Center** from the **View** menu **-> Toolbars** command.
- Select the **Domain Manager** command from the **InfoWater Control Center -> InfoWater button -> Tools** menu.
- At the Domain Manager dialog box, choose the Domain Creation Process you want to use from the 6 different options available namely Map Selection, DB Query, Query Set, Selection Set, Special Query and/or Network . You may use more than one (or in fact all of the different methods if you so choose) method to create the domain
- Click on the **Add** button to add the elements to your domain.
- At this point, only those elements selected will turn red.
- By editing the database tables related to these elements, the user will only see the domain affected database records and not the entire database (as long as the domain option is checked under the DBEditor).

Clear a Domain

To clear a domain, do the following:

- From the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Domain Manager**.
- With the dialog box open, click on the **Reset** button. Doing this will clear the selected domain and return the elements to their

normal color.

- Alternatively you may click on the **Clear Domain** icon  on your **InfoWater Edit Network** toolbar.

Other Related Topics - Domain Manager**,** Facility Methodology**,** Facility vs Domain**,** Facility Manager

---

Minor Losses

Minor losses (also called local losses) are caused by the added turbulence that occurs at bends and fittings. The importance of including such losses will depend on the layout of the network and the degree of accuracy required. In water distribution systems, minor losses are typically small compared to pipe friction losses. However, these losses can be significant in heating and cooling systems.

InfoWater allows each pipe and valve to have a minor loss coefficient associated with it. It computes the resulting headloss from the following formula:

$$h_L = \frac{0.0252 K q^2}{d^4}$$

where h_L is the headloss in feet, K is a minor loss coefficient, q is flow rate in cfs, and d is diameter in ft. The table below gives values of K for several kinds of components.

Minor Loss Coefficients for Common Components

Component	Loss Coefficient
Globe valve, fully open	10.0
Angle valve, fully open	5.0
Swing check valve, fully open	2.5

Gate valve, fully open	0.2
Butterfly valve, fully open	0.3
Diaphragm valve, fully open	2.3
Short-radius elbow	0.9
Medium-radius elbow	0.8
Long-radius elbow	0.6
Standard tee, 90^{o} Turn	1.8
45° elbow	0.4
Closed return bend	2.2
Standard tee - flow through run	0.6
Standard tee - flow through branch	1.8
Square entrance	0.5
Exit	1.0

Variable Speed Pump Analysis

VSP/FPP Analysis

In instances where a variable speed pump exists, it may be desirable to determine pump speed variation over time. Knowing the speed variation is required in order to conduct an energy analysis for the pump during an EPS simulation. It is also important for creating a more realistic hydraulic model.

Where pumps do not need to be run at full speed then energy costs can be reduced by controlling the pump motors with variable speed drives (a.k.a., variable frequency drive). In addition, where constant speed motors cannot adequately provide a necessary range of pumping requirements, then variable speed motors should be considered. The use of variable speed drive enables the pumped flow to match varying water demands, such as in a closed distribution system. With a variable speed pump, the pump (characteristic) curve, rather than the system curve is altered to obtain the necessary flow.

Before running a VSP/FPP Analysis, it is important to note the following:

- Variable Speed Pumps cannot be placed in parallel. Parallel VSP will create a hydraulically infeasible situation since the number of known target pressures will be less than the number of variable speed parameters that need to be adjusted (a one to one relation must always be maintained for hydraulic feasibility).
- The target junction node may be located at any geographical location in your model. In other words a variable speed pump can maintain a specified pressure at any junction in your system. However, each junction can be controlled by only one variable speed pump i.e., every variable speed pump needs to have a unique target junction specified.

- All standard controls defined for variable speed pumps will be omitted. Rule based controls cannot be used for a VSP. The variable speed pump can be only controlled by the user specified discharge pressure.

- A variable speed pump can only be described by points of operating data. Therefore, a constant power input pump is not applicable.

How a VSP/FPP Analysis Works

During a VSP/FPP Analysis, InfoWater allows the user to specify a target pressure to be maintained at a target junction node along with the minimum and maximum allowable speeds for each variable speed pump in the system. The lower and upper pump speed limits should represent a wide variety of operating conditions and should include the range where the pump is expected to operate. During the analysis, the program automatically computes the pump speed required to meet the targeted pressure at the target junction node. If no solution can be found within the user specified pump speed range, then InfoWater will set the pump speed to the corresponding upper or lower speed limit. A warning message will also be issued indicating that no solution is available within the operating range specified. This closely describes actual variable speed pumping operation.

Note: If the **Target Node ID** entry is left blank (no entry), then InfoWater will assume that the pump discharge node is that the target node and the desired pressure will be maintained at that location.

Methodology

To run a Variable Speed Pump analysis do the following:

- To conduct a variable speed pump analysis, from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Run Manager** to display the Run Manager dialog box. From the Standard tab, select the VSP/FPP Analysis button to launch the Variable Speed Pump dialog box.
- The Data Tab allows the user to input VSP Model parameters. Enter the Pump ID for which the VSP Analysis is to be conducted. More than one pump can be analyzed during a VSP simulation.
- Specify the minimum pump speed to be utilized during the VSP Analysis. This is the minimum value that will be employed by InfoWater.
- The maximum pump speed to be utilized during the VSP Analysis. This is the maximum value that will be employed by InfoWater.
- Specify the Target Node ID. This is an optional field. Specify the target junction node here where the desired target pressure is to be maintained. The speed of the pump for which a VSP analysis is being carried out will be varied to maintain the specified target pressure at this junction. This target junction may be located anywhere in your system and has to be unique for each VSP pump when multiple VSP analysis is run. If the **Target Node ID** entry is left blank (no entry), then InfoWater will assume that the pump discharge node is that the target node and the desired pressure will be maintained at that location.
- Specify the target pressure to be maintained at the target node ID during the VSP Analysis. InfoWater will calculate the pump

speed between the min. and max. values to maintain the specified pressure at the target junction node.

- For an EPS, the target pressure can be specified to vary during the simulation period as a time pattern. In that case, the time pattern ID should be specified in the Pressure Pattern ID field.

- Once the above data are specified, click the **RUN** button to initialize the VSP Analysis. The results are then displayed as shown in the Report dialog box. The red color indicates that the pump is off while the green represents normal pump operation and the corresponding results (speed setting, discharge flow and discharge pressure) are shown in the associated columns.

- Review the results from the run and for an EPS use the export tab to export the VSP Analysis results.

- For an Extended Period Simulation (EPS), InfoWater automatically computes the pump speed pattern to maintain the desired targeted pressure (which can be defined as a time varying pattern) at the target junction node. This pattern can also be exported for use with the standard variable speed pump (manual) operation.

- Once the speed pattern has been created and exported to the selected pump, a Standard hydraulic simulation via the Run Manager will then need to follow the VSP Analysis in order to adequately determine pump behavior and pump operating costs.

Other Related Topics - Running a Model, Variable Speed Pump Dialog Box, Variable Speed Pump Export tab, Variable Speed Pump Report tab

---

Simple Controls

A simple control allows the user to modify the status of elements (pipe, pump or valve) during a hydraulic simulation - depending on the state of some other element in the system as specified by the user. For example, actual field operations may consist of having a pump turn on and off depending on the high and low water levels in a storage tank. To model this type of situation, the user would place a simple control on the element in question. Simple controls apply to both steady state as well as extended period simulation models. To build more advanced logic statements based on if, then, else premises, use a Rule Based Control.

Methodology

To create Simple Controls do the following:

- Select the appropriate element in your InfoWater model by clicking on it. Controls may be established only for Pipes, pumps and Valves.
- From the **Attribute Browser** window select the **Tools** icon  .
- Click on the **Control** option to launch the Simple Controls dialog box.
- Specify the appropriate Type, Method and the values and click on **Insert** to create a New Control.
- Repeat the above steps to create more controls. use the **Update** button to change or alter an existing Control.
- Click on the Move Up/ Down icons to change the order of the controls.

Other Related Topics - Controls Overview**,** Initial Status Methodology**,** Pipe Initial Status**,** Pump Initial Status**,** Rule Based Control Dialog Box**,** Rule Based Control Methodology**,** Simple Control Dialog Box**,** Valve Initial Status

---

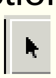
Rule Based Logic Controls

Rule Based Logic Controls are used by InfoWater to perform a given action during a hydraulic simulation when a user specified condition is met. However, unlike Simple Controls, rule based controls allow for the creation of multiple conditions to be satisfied before an action is performed.

The basic building blocks for rule based logic statements are multiple premises (conditions) encountered in the distribution system (if node 11 has a pressure below 25 psi and tank 12 has a level below 12 feet) and their resulting action statements (then turn pump 200 on).

Methodology

To create a Logical Control do the following:

- Launch the **InfoWater Control Center** (if it is not available) from the ArcMap **View** menu -> **Toolbars** command.
- To create logical controls in a simulation, from the **InfoWater Control Center -> InfoWater** button **-> Edit** menu, select **Rule-Based Control**.
- Enter an ID and an optional Description for your Logic Control (InfoWater IDs are 20 characters, no spaces and no funny characters and the Description is 60 Characters)
- On your Rule Data Edit dialog box, under the Rule Clause Data section, under the Data Type methodology create Premises first. Premises are conditions that when met will trigger the action.
- To create a Premise, choose the Premise radio button, then under Object specify the premise Object type (such as a Junction or Reservoir etc) and the Object ID. Click on the Select icon  to graphically select the Object from your InfoWater Project.
- Specify the Data Comparison Attribute and the Relationship and the Value that you want to compare against (Pressure > 20 for say Junction type Object)
- Click on the Insert button to Insert this Premise into the Rule Statement section of the Rule Data Edit box.
- Repeat the above steps to insert as many premises you want for the control.
- Once the Premises (conditions) have been established, create an Action Statement by choosing the Action radio button.

- Then specify the Action Clause (Then, Else, And) and choose the Object and the Object ID (just like for Premises)
- Choose the Object Action Attribute, Relation and the Status and click on Insert to add it to the Control.
- Create as many actions as you want by repeating the above steps. Use the **Update** button to change or alter an existing Premise or Action in the Control.
- When all the premises and actions are in place, click on the  button to save and exit from the Controls Dialog box.
- Make sure that you set the Logic Controls **On** prior to running your model through the **Table of Contents -> Operation** tab **-> Simulation Options** dialog box by checking the **Rule Control** command. InfoWater will read the Controls only when this option is checked.
- Repeat the above steps to create more controls.

Other Related Topics - Controls Overview**,** Initial Status Methodology**,** Pipe Initial Status**,** Pump Initial Status**,** Rule Based Control Dialog Box**,** Simple Control Dialog Box**,** Simple Controls Methodology**,** Valve Initial Status

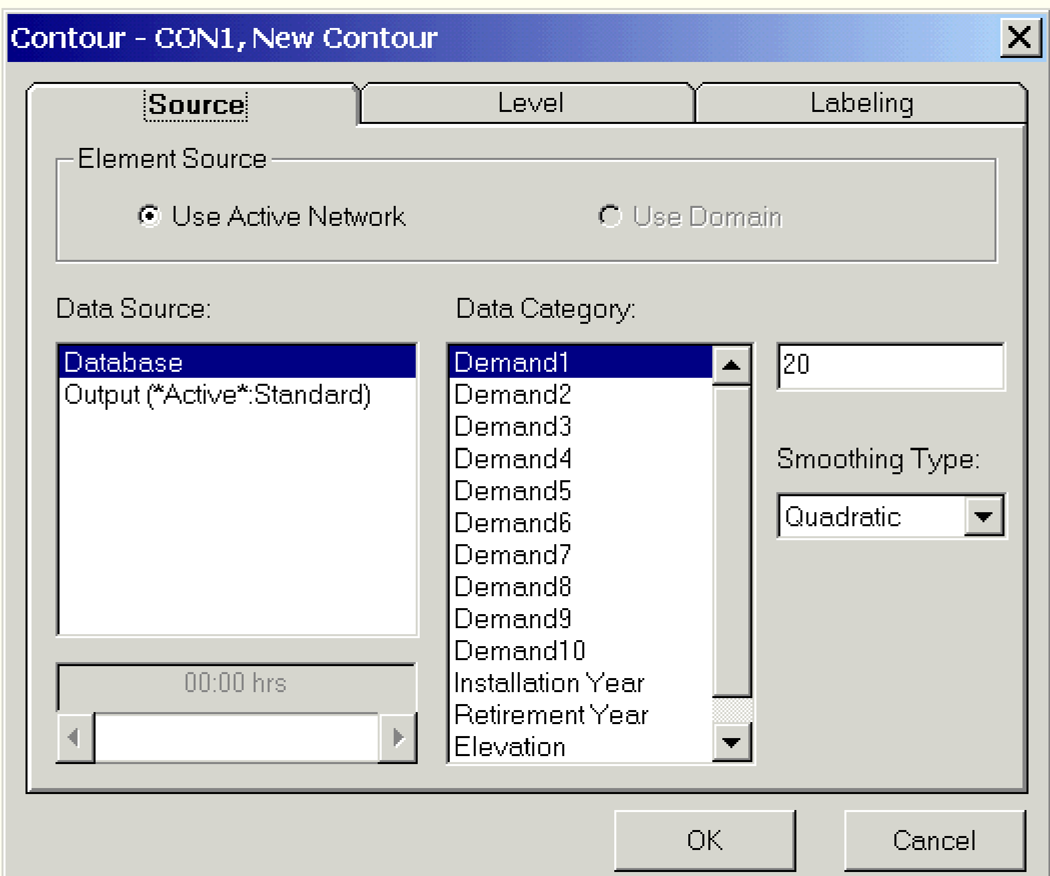
---

Contour Dialog Box

The Contour section of the **Table of Contents** is used to generate, edit and delete contours. Contours can be displayed at any time before or after a simulation run and are saved as a permanent part of the InfoWater projects in layers.

Contours can be generated for any numeric **Junction**-based model input field or simulation result variable and also can be used to visualize the impacts of a hydraulic model run. By generating contours, the user is able to immediately discern where problem areas exist within an existing or proposed system. Click here to learn about the Contour creation process.

Click on any section below to learn more.



Other Related Topics - Annotation Dialog Box, Annotation Methodology, Contour-Labelling, Contour-Level, Contour Methodology, Contour Options, Operation Data, Table of Contents

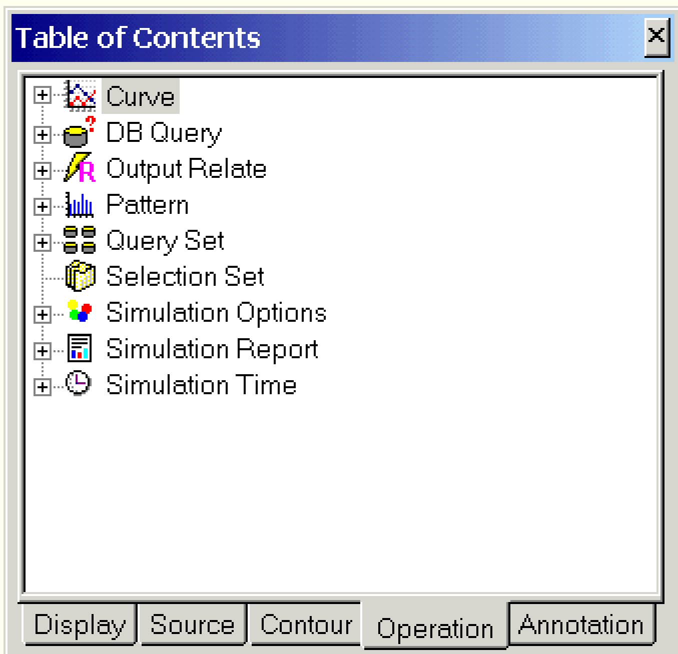
Operation Data

This section of the InfoWater Table

of Contents provides you with tools to create, edit and operate

on the different operational features of InfoWater.

Click on any section below to learn more:

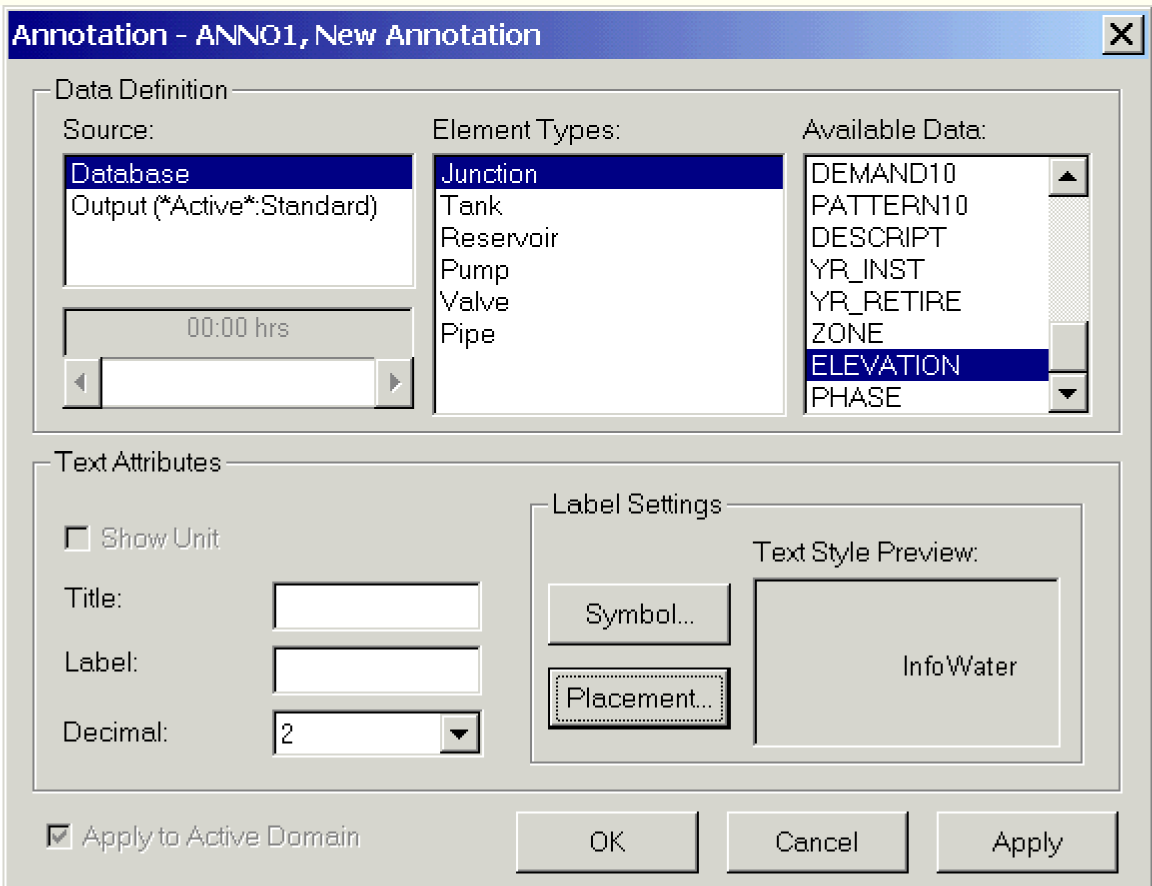


Other Related Topics - Annotation Dialog Box, Annotation Methodology, Contour-Labelling, Contour-Level, Contour Dialog Box, Contour Methodology, Contour Options, Table of Contents

Annotation Dialog Box

The annotation feature allows you to create annotations for viewing in the InfoWater project. Such annotations can consist of any input and/or output data as provided in the InfoWater project and model databases. To learn more about the process of creating annotations click here.

Click on any section below to learn more.



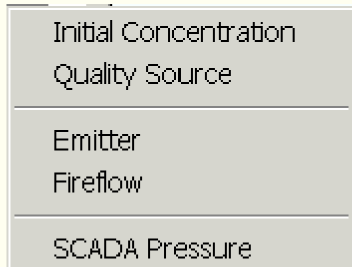
Other Related Topics - Annotation Methodology, Contour-Labelling, Contour-Level, Contour Dialog Box, Contour Methodology, Contour Options, Operation Data, Table of Contents

Tools

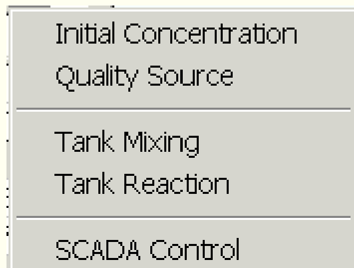
Use this command to access certain advanced tools provided by InfoWater to manipulate data related to the currently active element. Click on the Down Arrow icon  to launch the **Tools** menu. The tools menu changes depending on the type of element presently selected as shown below.

Click on any portion below to learn more:

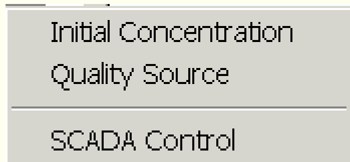
Junction



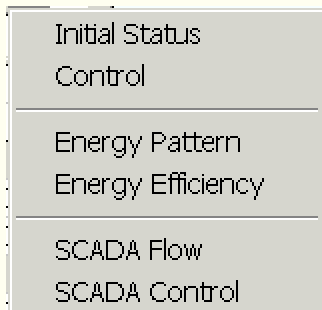
Tank



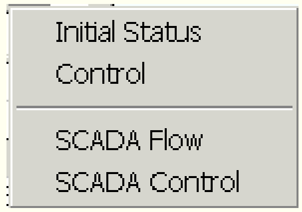
Reservoir



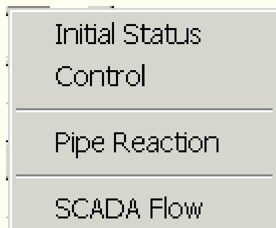
Pump



Valve



Pipe

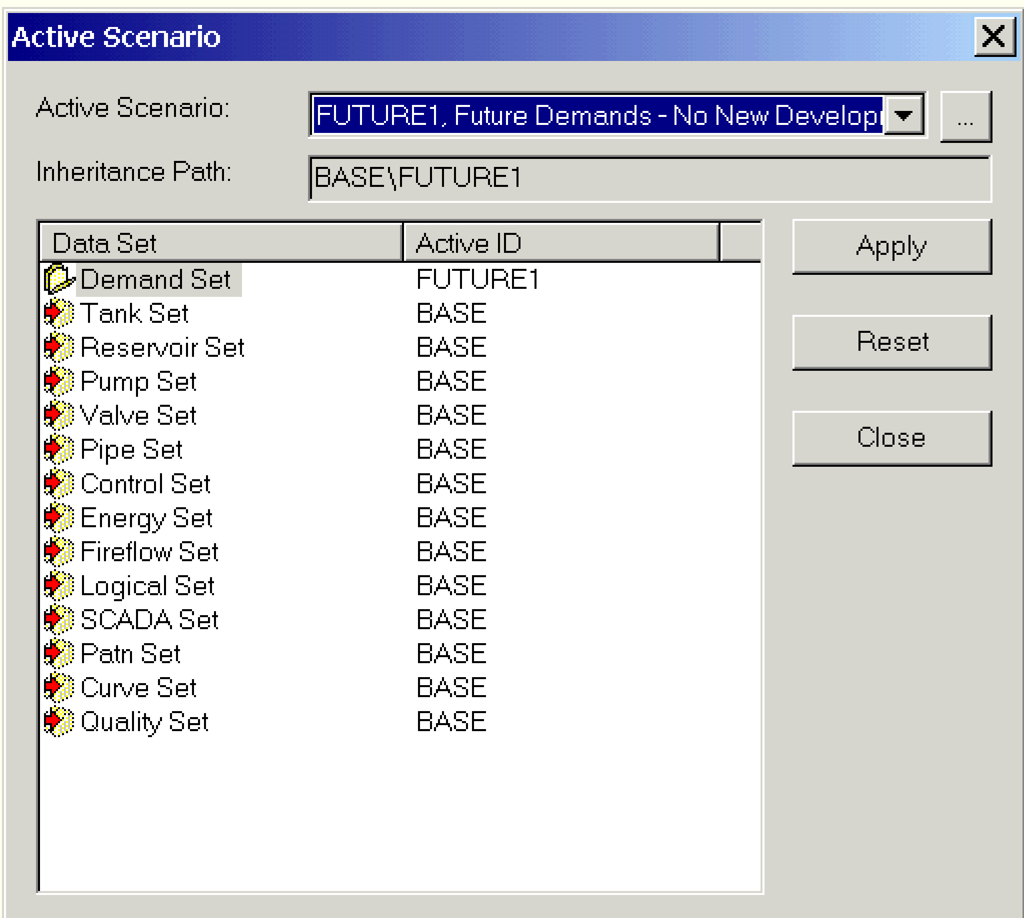


Other Related Topics - Attribute Browser

Edit Active Scenario

The Active Scenario dialog box allows the user to select any of the scenarios created in an InfoWater project and make it active. Once a scenario is made active, each of the three facets that comprise a scenario (facility sets, data sets, simulation sets) also become active.

Click on any section for more information.



Other Related Topics - Compare Scenarios, Compare Scenario Methodology, Comparison Report, Data Sets Methodology, Dataset Manager, Facility Sets Methodology, General Options Methodology, Scenario Manager Data Sets, Scenario Manager Facility Sets, Scenario Manager General, Scenario Manager Main Dialog Box, Scenario Methodology

Run Manager Methodology

The Run Manager is used to perform simulations and to manage simulation output sources (results). Five individual simulation types are available from the Run Manager; Standard simulations (Hydraulics-Steady State & Extended Period Simulations, Water Quality, Energy management), Fireflow simulations, SCADA, Hydrant Curves, and System head curves.

Methodology

- Activate the Scenario that you want to run from your **InfoWater Control Center - Active Scenario** select box.

- Launch the Run Manager dialog box by clicking on **Run Manager** from your **InfoWater Control Center -> InfoWater** button **-> Tools** pull down menu.

- Choose the appropriate tab. Use the Standard tab for Steady State, Extended Period Simulations, Water Quality and Energy management Analysis. For Fireflow simulations, SCADA, Hydrant Curves, and System head curves choose the appropriate tabs.

- Choose the options required for each of the analyses and click on the **Run** icon to run the simulation.

- Check the Run Manager **Run Status** indicator and confirm a green light. If you get a Red light or a Yellow light click on the **Report** icon next to your **Run** icon on the Run Manager dialog box to trouble shoot. Click here for more information on the Simulation Report customization and creation process to aid in better trouble shooting and verification of the problem areas in your network.

- After a successful simulation review your results using the InfoWater Output Report manager.

Other Related Topics - Running a Model, **Run Manager**, Batch Run Manager, Batch Run Methodology

---

Batch Simulation Methodology

The Batch Simulation Manager is used to run models for numerous user-selected scenarios in a single operation. This command is especially useful where several simulations are simultaneously required for a large model. With the Batch Simulation Manager, the user can select the desired scenarios and run each model in a "batch" process. The Batch Simulation Manager is only available for standard simulations and not available for Fireflow, SCADA, etc. model runs.

Methodology

Once all your scenarios are in place, use the batch simulation manager to run all your desired scenarios at one time.

- Launch the Batch Simulation Manager dialog box by clicking on **Batch Simulation** under the **InfoWater Control Center -> InfoWater** button **-> Tools** menu.
- Choose the Error handling options. You may wish to continue with the simulation regardless of the errors or you may wish to choose the stop option.
- Also specify the Execution Order. This will determine the order in which all the selected scenarios will be run.
- From the Scenario Selection section choose the scenarios that you want to run by clicking on them. You may choose as many scenarios as you wish. You may also choose all the scenarios by clicking on the Select All button.
- Change the order of the scenarios listed in the Batch Simulation dialog box using the **Move up**/**Move Down** buttons.
- Finally click on the **Run** button to run all your selected model simulations.

Note - InfoWater will activate each of your Scenarios one at a time and run them. The Scenario listed last in your list of selected scenarios (depending on your Execution Order) will be run last and will be saved as the **Active* Scenario.

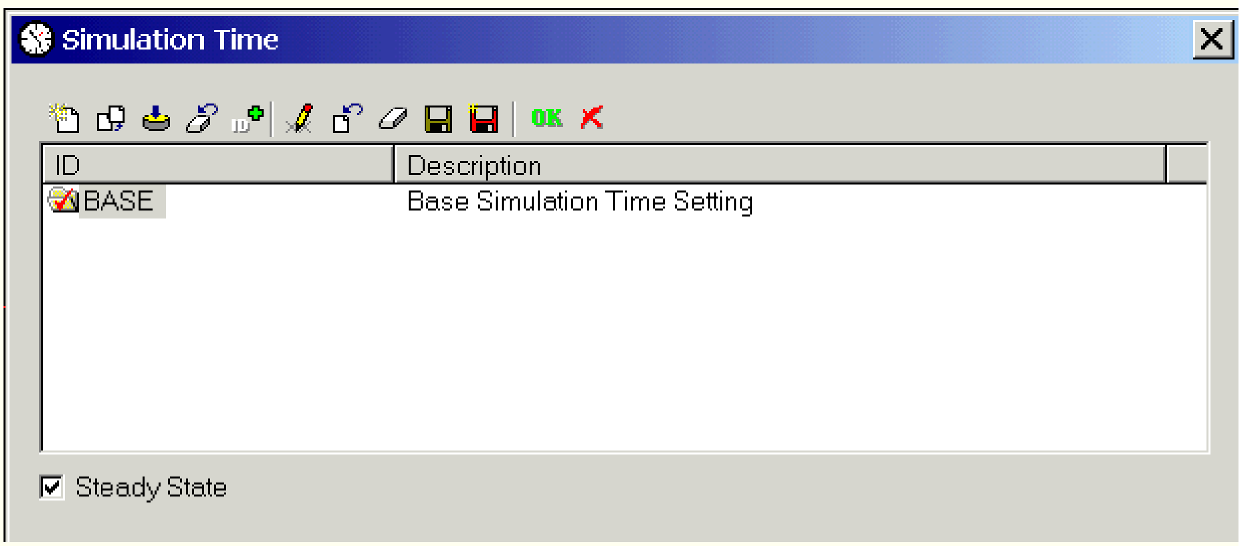
Other Related Topics - Running a Model, **Batch Run Manager**, Run Manager, Run Manager Methodology

Steady State

A steady state simulation is a single, instantaneous "snap-shot" of a water distribution system. Flows are based on system demands and pressures are dependent upon pump curves and initial storage reservoir parameters.

Click here to learn more about steady state modeling.

From the **Table of Contents -> Operation** tab choose **Simulation Time**. Once there, double click the BASE simulation time to see the dialog box below. Click on the Steady State option to place a check in the box. Once this is done, close the dialog box by clicking the **OK** button.

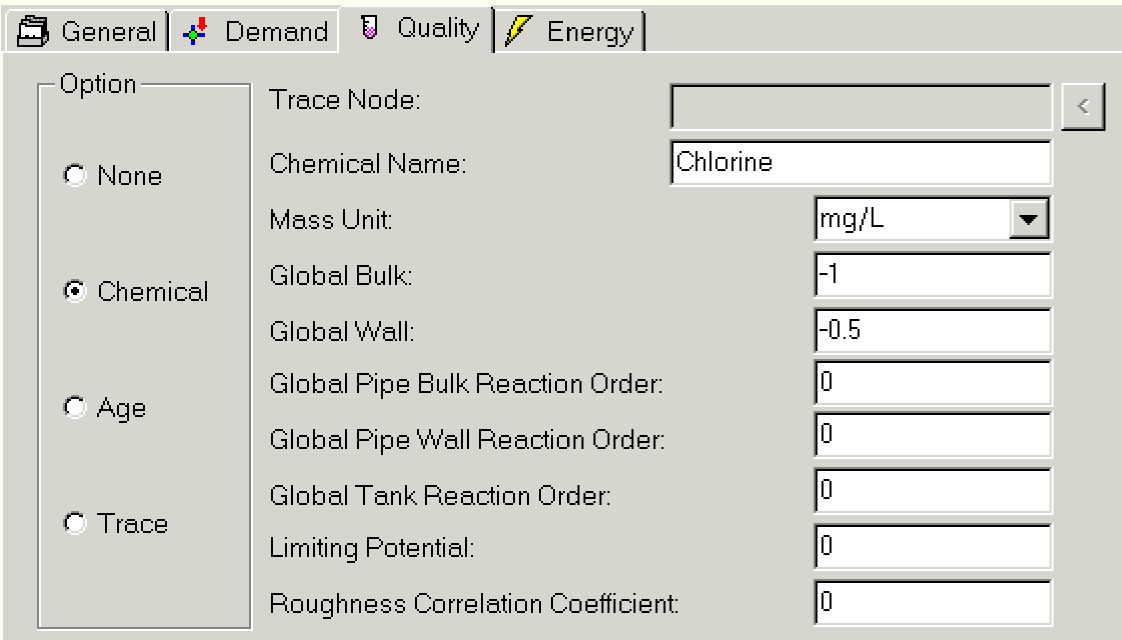


Other Related Topics - Running a Model, Steady State Modeling Methodology

Quality

Use this to choose the kind of Water Quality analysis you want to run. Also specify the different parameters for the analysis run.

Click on any section for more information.



Other Related Topics - Running a Model**,** Simulation Options Dialog Box**,** Simulation Options - Demand, Simulation options - Energy, Simulation Options Methodology, Simulation Report, Simulation Report Customization Tab, Simulation Report Methodology, Simulation Time, Simulation Time Methodology

Water Quality Simulation

Several other important model input data additional to the network components must be specified to define a water quality model run. These include the following:

- *Hydraulic Model Parameters*
- *Simulation Options*
- *Extended Period Simulation Timestep*
- *Simulation Report Definition*

(Note: InfoWater only performs a water quality analysis in conjunction with an extended period simulation hydraulic analysis.)

Water Quality Growth or Decay

When performing a constituent growth or decay analysis (non-conservative substances), the following model inputs must also be specified:

- **Water quality sources** - Nodes representing sources of a water quality constituent into the network and optionally, patterns defining how the constituent injection rate changes throughout the simulation. At least one water quality source node must be identified. Use the **Quality Source** command under the **Tools** icon  on the **Attribute Browser** window to identify water quality sources or use the Group Editing command to assign quality sources to a set of junctions, tanks and/or reservoirs. InfoWater can model the following types of sources:

- A concentration source fixes the concentration of any external inflow entering the network at a node, such as flow from a reservoir or from a negative demand placed at a junction.
- A mass booster source adds a fixed mass flow to that entering the node from other points in the network.
- A flow paced booster source adds a fixed concentration to that resulting from the mixing of all inflow to the node from other points in the network.
- A setpoint booster source fixes the concentration of any flow leaving the node (as long as the concentration resulting from all inflow to the node is below the setpoint).

The concentration-type source is best used for nodes that represent source water supplies or treatment works (e.g., reservoirs or nodes assigned a negative demand). The booster-type source is best used to model direct injection of a tracer or additional disinfectant into the network or to model a contaminant intrusion.

- **Reaction rate coefficients** - Define how the constituent grows or decays as it reacts with the pipe wall, in tanks, and in the bulk flow. At least one reaction rate coefficient must be defined. Use the **Pipe Reaction** command under the **Tools** icon  on the **Attribute Browser** window to set a reaction rate coefficient for an individual element or use the Group Editing command to assign reaction rates to a set of pipes or tanks. Set the global reaction rate using the Quality tab under Simulation Options.
- **Initial water quality** - Defines the initial constituent concentration at selected or all network nodes. This information is optional. If not specified, InfoWater assumes an initial constituent concentration of 0.0. Use the **Initial Concentration** command under the **Tools** icon  on the **Attribute Browser**

window to set water quality values for nodes within the system or use the Group Editing command to initial water quality values to a set of nodes or tanks.

Water Quality Timestep

In addition to the timesteps set for an extended period simulation, the following timesteps may be set for water quality analysis:

- **Quality timestep** - The timestep that will be used to track water quality changes through the network. If not supplied, InfoWater uses an internally computed timestep based on the smallest time of travel through any pipe in the network (the “minimum travel time”). Use the Simulation Time option to set the quality timestep.

Methodology

To run a Water Quality analysis, perform the following procedure:

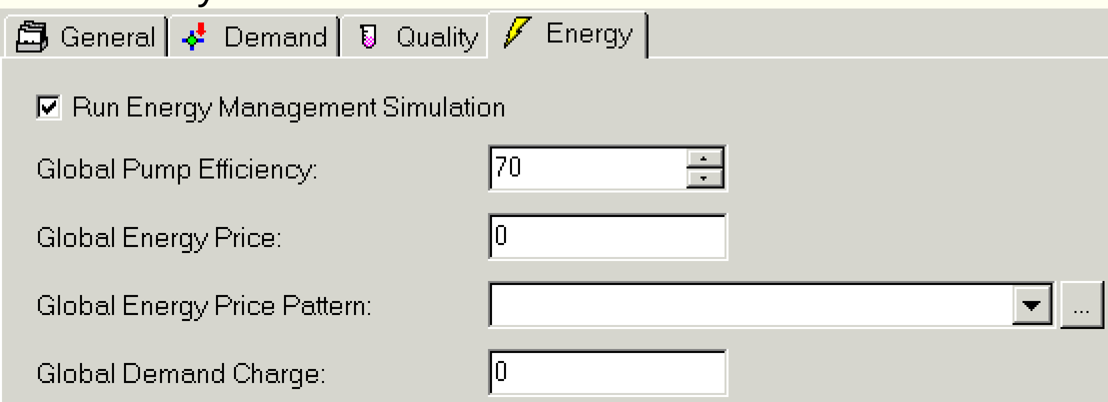
- Set the options described above in this section under the Simulation Options dialog box,
- Assign your options to the appropriate scenario (via the Scenario Manager),
- Supply chemical concentrations at the subject tanks and nodes.
- Select the Run icon from the Run Manager to run the model.
- See the Output Report Manager to view results. To learn more about output reports, click here.

Other Related Topics - Running a Model, Water Quality

Energy

Specify the global energy settings. This will be used when running an Energy Management Simulation.

Click on any section for more details:



Note - In order to run an Energy Management Simulation you would need to check the Run Management Simulation check box.

Other Related Topics - Running a Model**,** Simulation Options Dialog Box**,** Simulation Options - Demand, Simulation Options -Quality, Simulation Options Methodology, Simulation Report, Simulation Report Customization Tab, Simulation Report Methodology, Simulation Time, Simulation Time Methodology

---

,

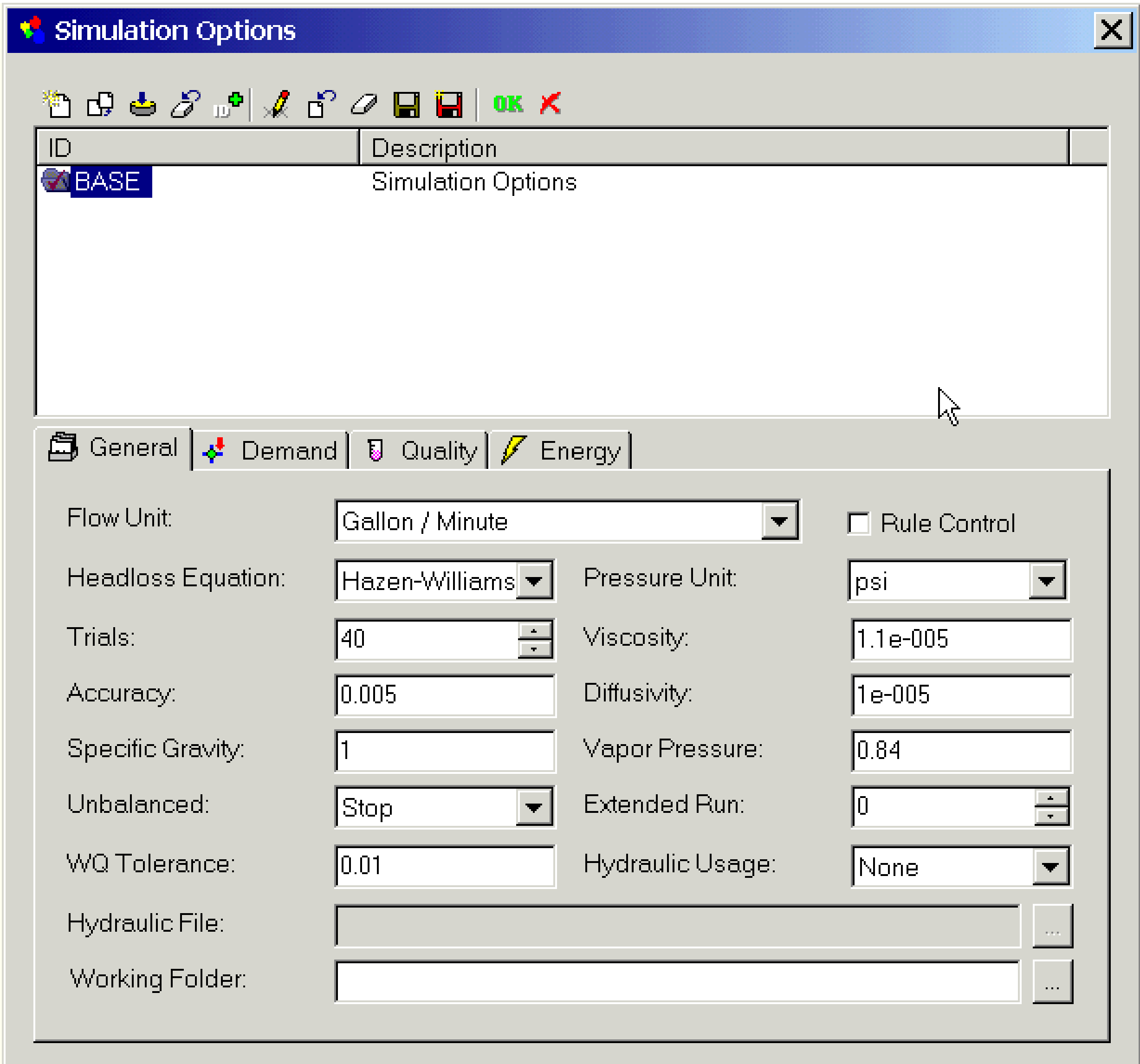
Simulation Options Dialog Box

The Simulation Options dialog box is used to adjust some facet of a model prior to a model run. Simulation options are usually altered to be associated with a given scenario.

For example, suppose the user wishes to specify a global bulk chlorine decay of -0.5 for one model run and have another model run with a decay of -0.7 specified. This can be accomplished through the Simulation Options where one option can be tied to one scenario while another option is specified for a different scenario.

To create a simulation option, from the Edit pull down menu select Simulation Options.

Click on any icon or tab below to learn more.



Other Related Topics - Running a Model, Simulation Options - Demand, Simulation options - Energy, Simulation Options -Quality, Simulation Options Methodology, Simulation Report, Simulation Report Customization Tab, Simulation Report Methodology, Simulation Time, Simulation Time Methodology

---

Energy Management

In today's energy conscious environment, the ability to accurately model the energy consumption of your system is critical. Building an energy simulation lets you further optimize your power consumption by providing the data required for modifying system operations, negotiating electricity contracts and targeting an optimal rate structure with your energy provider. You will not know how much money you are losing or how efficient your system can become until you leverage the true power of InfoWater.

Click here to learn more about Energy Management Modeling.

Other Related Topics - Energy Management Modeling, Running a Model

---

Fire Flow Methodology

Fire flow computations can be carried out for a steady-state simulation or for any time step during an *EPS* run. This allows direct evaluation of system performance under a wide range of demand loading and operating conditions (e.g., maximum hour, seasonal change, etc).

How do I...

- Assign Fire Flow Demands?
- Create Fire Flow Sets?
- Run a Fire Flow Simulation?

Properties of InfoWater Fire Flow Analysis

Via the Run Manager, the following fire flow analysis is provided by InfoWater:

- **Fire Flow** – InfoWater calculates fire flows based on a fire flow occurring at a single junction node. Fire flow results are calculated for each junction node in the fire flow set (i.e., nodes assigned fire flow demands) one node at a time. Fire flows are calculated for a single simulation time period.

Several important properties of the InfoWater fire flow analysis tool include:

- *Hydrant Locations*
- *Fire Flow Distribution*
- *Fire Flow Demands*

- *Running a Fire Flow Simulation*
- *Reviewing Fire Flow Simulation Results*
- *Alternatives Management*

Note: A properly calibrated hydraulic model of the water distribution system is required for fire flow modeling applications.

Assigning Fire Flow Demands

Fire flow demands at any or all junction nodes must be specified prior to running a fire flow simulation. InfoWater will perform a fire flow simulation for all junctions that have been assigned fire flow demands at the time a fire flow simulation is initiated. Junctions assigned a fire flow demand are part of the **fire flow set** and can be edited via the DBEditor.

To Assign a Fire Flow Demand...

- *Single Junction*: From the **Attribute Browser** window select the **Tools** icon and click on Fire Flow to assign a fire flow demand for a single junction node.
- *Group of Junctions*: Use the Group Editing (on Domain or Selection) command under the **InfoWater Control Center -> InfoWater** button **-> Edit** menu to assign or modify fire flow demands for groups of selected junction nodes. Click here for more information on the Group Editing feature of InfoWater.

Creating Fire Flow Sets

Alternative fire flow scenario sets are supported via the Scenario Manager. The user can create numerous fire flow sets that can be saved and then activated into the current scenario for easy, comparative analysis.

To create a fire flow set, from the **InfoWater Control Center -> InfoWater** button **-> Scenario** menu, select **Fire Flow Set**. At this point, the user can clone the *active* set to a new set and name the new set. Once this is done, the user can activate the new set by assigning the new set to a scenario through the Scenario Manager and then run the fire flow model.

Running a Fire Flow Simulation

To run a fire flow simulation,

- First assign Fire-flow data to one or more Junctions using any of the methods specified above.
- From the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Run Manager**.
- Once the Run Manager dialog box is initiated, choose the Fireflow tab, set the Fireflow Options, and click on the **Run** button on the dialog box.
- Upon successful completion of a fire flow simulation, the signal light on the Run Manager will turn green. To view the results of a fire flow simulation, click the **OK** button to close the Run Manager dialog box and from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select the Output Report Manager.
- Click the **New** button and select the current *Active*.Fireflow output source.

- Under the **Report** tab you will see that two different fire flow reports are generated. The first is the standard fire flow results while the second report displays a fireflow design report. For each report, one result record is displayed for each junction node in the fire flow set (i.e., nodes assigned fire flow demands at the time the simulation was initiated). Click here to learn more about the results generated for a fireflow simulation.

Other Related Topics - Running a Model, Fire Flow Dialog Box

SCADA Integration

The intent of the SCADA module is to update InfoWater with the most recent SCADA data and to monitor your hydraulic system as opposed to controlling your system through InfoWater.

Based on a pre-defined ASCII output file from the SCADA system, a hydraulic simulation can then be executed by InfoWater manually upon request or automatically on a real-time basis (based on a user-specified time period, ex., every 30 minutes). The new or most recent SCADA values would then replace the existing boundary data in the active InfoWater project prior to a hydraulic simulation.

To learn more about SCADA Integration, click on any of the following subjects.

Keys to Success - SCADA Output Format

Example SCADA Input File

Implementing SCADA Data

- SCADA System Data Edit (Inflow and Outflow)
- SCADA Configuration
- SCADA Measurement

Running a SCADA Simulation

Keys to Success - SCADA Output Format

Communication between the SCADA system and InfoWater occurs with a predefined ASCII file (ex. C:\SCADA\run1.sda) which can reside anywhere on the computer system. The key to successful data transfer is the format of the SCADA output file. It must adhere to a specific format as follows:

- [TIME] *time* - actual clock (military) time with SCADA readings; the time will be used as the reference time for the InfoWater model simulation.
- [TANK DATA]
  MeterID - meter reference name from SCADA
  level - tank water level; InfoWater will update the corresponding tank water level.
- [PUMP DATA]
  pumpID - meter reference name from SCADA
  status - On, True, Up, or 1 if pump is open. Off, False, Down or 0 if pump is closed
  setting - measured flow value
  speed - optional pump speed value
- [VALVE DATA]
  valveID - meter reference name from SCADA
  status - Open or Closed
  setting - measured valve setting (pressure for PRV's, PSV's, and PBV's; flow rate for FCV's; minor loss coefficient for TCV's and degree open for MTV's).
- [PRESSURE MEASUREMENTS]
  meterID - meter reference name from SCADA
  setting - measured node pressure
- [PIPE FLOW MEASUREMENTS]
  meterID - meter reference name from SCADA

setting - measured pipe flow value

- [PUMP FLOW MEASUREMENTS]
  meterID - meter reference name from SCADA
  setting - measured pump flow value
- [VALVE FLOW MEASUREMENTS]
  meterID - meter reference name from SCADA
  setting - measured valve flow value
- [SYSTEM FLOW MEASUREMENTS]
  meterID - meter reference name from SCADA
  setting - measured inflow and outflow value (in flow unit) system flow value
- [END]
  Signals the end of the SCADA input file.

Note: The only mandatory sections are [TIME] and [END] which must respectively begin and end the file. The order of the other sections is not important. All sections must be separated by at least one blank line. Each section can contain one or more lines of data, however, blank lines are not allowed within a section. Data items can appear in any column of a line. Meter ID and data items in any line are separated with a colon (:), while data items on the same line are separated by spaces.

Example SCADA Measurement Input File:

[Time]
Time : 18:00

[Tank Data]
tank2 : 90.0

[Pump Data]
pump 9 : 1100.0 950.0
pump 10 : Off

[Valve Data]
valve 12 : Open
valve 12 : 65

[Pressure Measurements]
junction11 : 125.0
junction23 : 90.0

[Pipe Flow Measurements]
pipe10 : 40.5
pipe12 : 100.2
pipe122 : 135.0

[End]

Implementing SCADA Data

There exist three different ways of applying SCADA data to an InfoWater project.

1. To apply an overall approach which consists of specifying inflow and outflow data. This applies only if you are having water delivered to your system or are delivering water to another nearby water system. Of course, these turnouts need to be hooked up to your SCADA system. Click here to learn about applying inflow and outflow SCADA data.

2. Applying SCADA data to those elements whose settings/status are updated from the SCADA system. Such elements consist only of storage tanks, reservoirs, pumps and valves. Click here

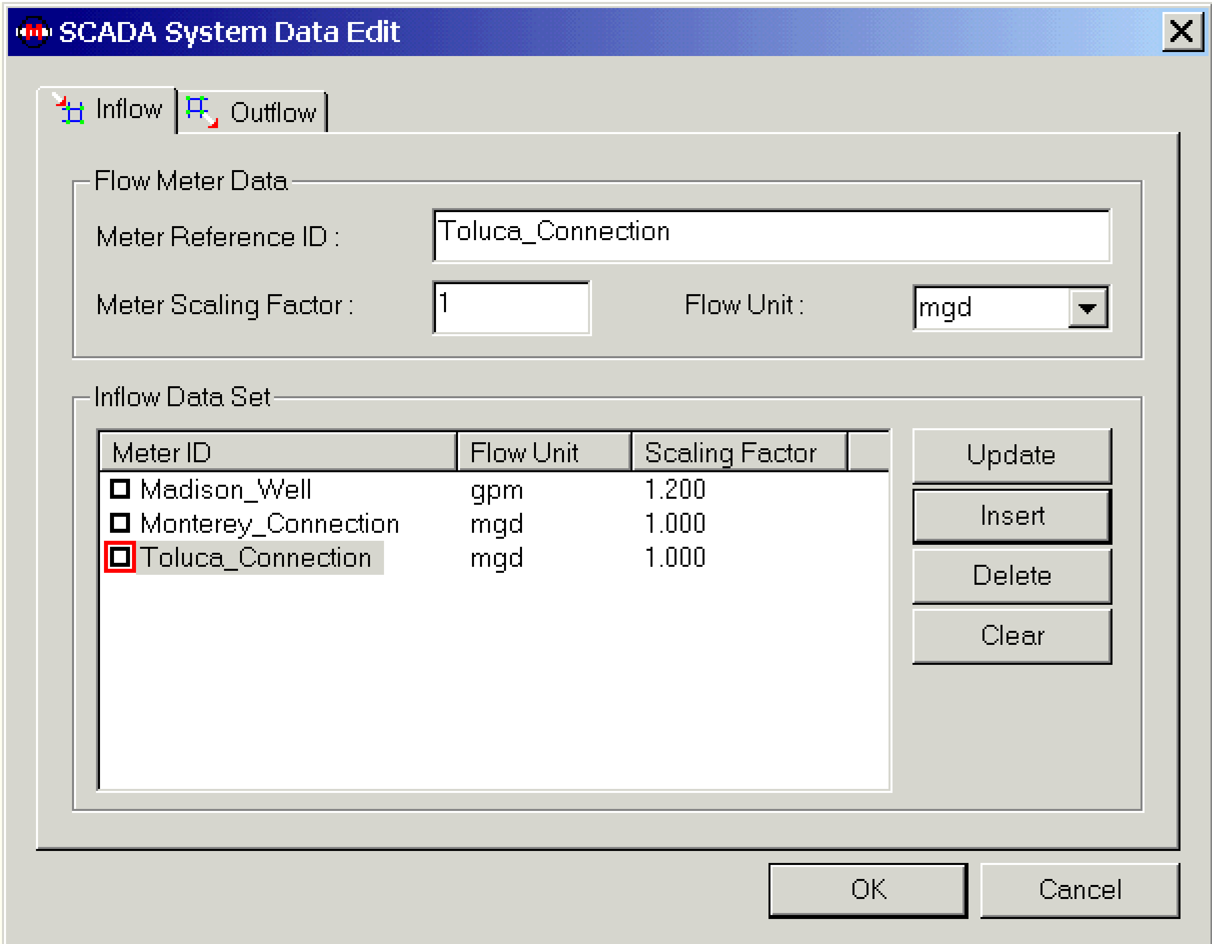
to learn more about applying SCADA Configuration data to these InfoWater elements.

3. Applying SCADA date to those elements for which pressure/flow levels are imported and alarm levels are set by the user. Such elements consist of junction nodes, pipes, pump or valves. Tanks do not apply (unless your tank is under pressure). Click here to learn more about applying SCADA Measurement data to these InfoWater elements.

SCADA System Data Edit

Some water authorities purchase their water from a provider while others serve as a provider to other agencies. If these turnout locations are hooked up to your SCADA system, use the SCADA Interface feature to monitor these locations.

To use the SCADA Interface feature, from the **InfoWater Control Center -> InfoWater** button **-> Edit** menu, select **SCADA Interface**. Doing this will initialize the SCADA System Data Edit dialog box shown below.



SCADA Configuration

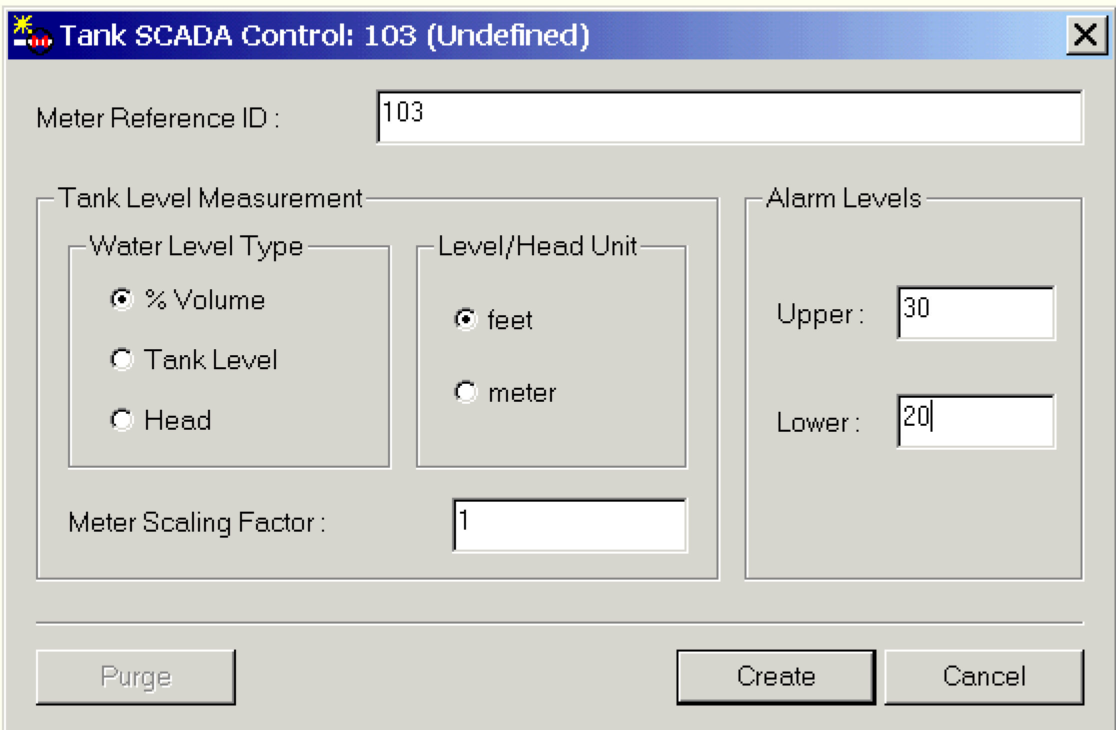
Configuration data is required only for those network elements whose settings or status are to be updated from the SCADA system and/or a comparison between modeled and measured data is desirable. Network elements which can be selected for SCADA configuration are storage tanks and pumps.

To enable this feature, select either a pump or storage tank then choose the SCADA Configuration icon under the Attribute Browser. The following represent the SCADA Configuration dialog boxes for tanks and pumps.

Tank Data

The SCADA configuration dialog box for tanks is shown below. Configuration is set for tank water levels and is used to define tank measurement data in the *[TANK DATA]* section of the SCADA Measurement Input File.

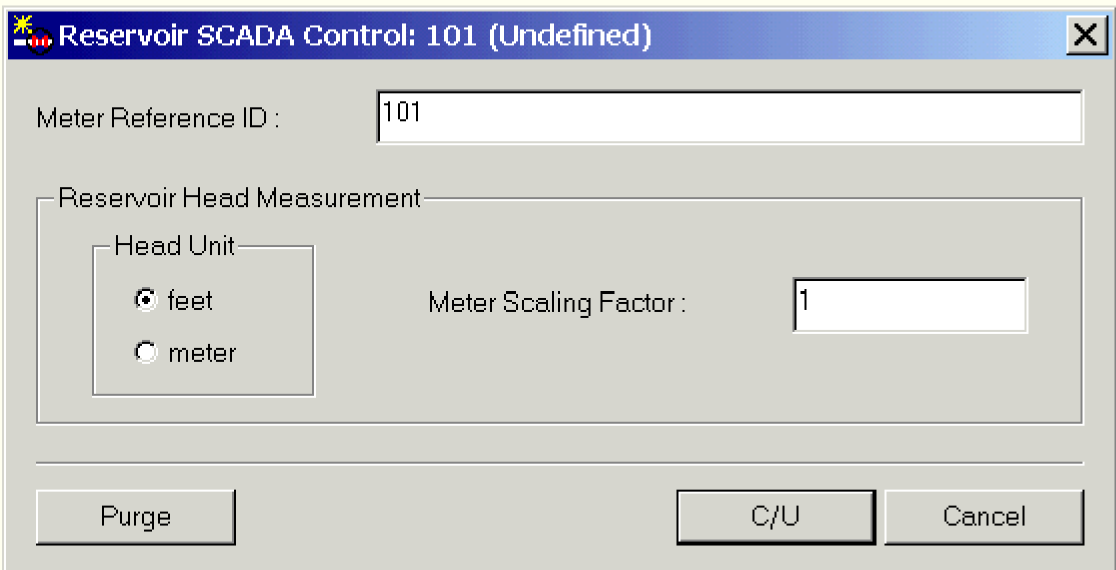
Click on any section to learn more.



Reservoir Data

The SCADA configuration dialog box for reservoirs is shown below. Configuration is set for tank water levels and is used to define tank measurement data in the *[TANK DATA]* section of the SCADA Measurement Input File.

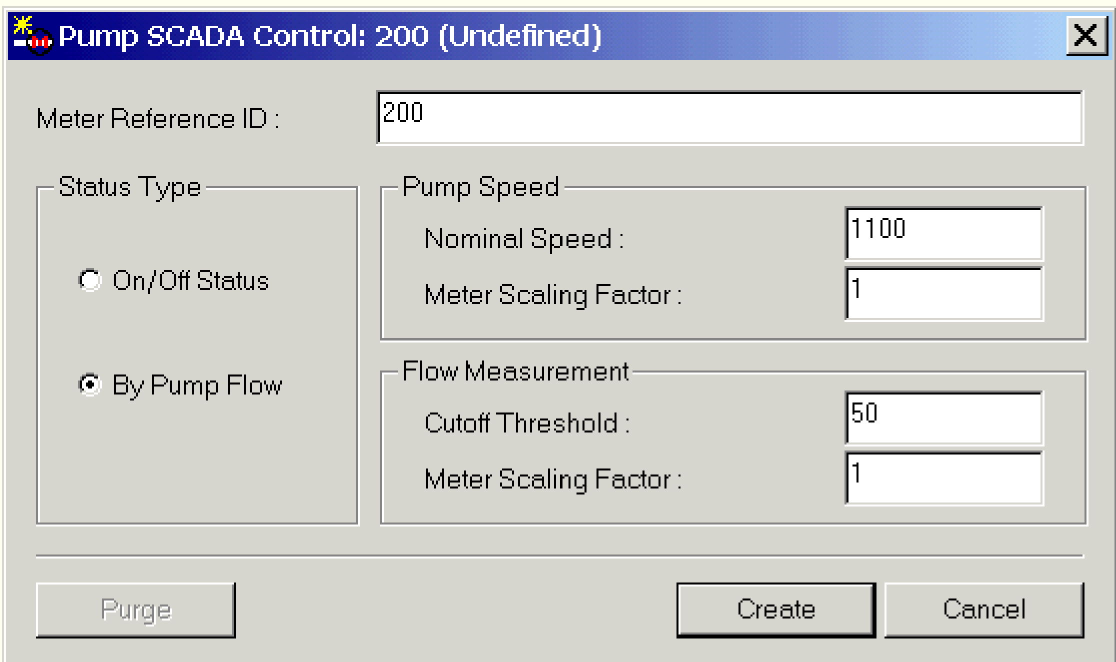
Click on any section to learn more.



Pump Data

The SCADA configuration dialog box for pumps is shown below. Configuration is defined for pump ON/OFF status and optionally for the pump operating speed. This configuration is used to define pump measurement data in the *[PUMP DATA]* section of the SCADA Measurement Input File.

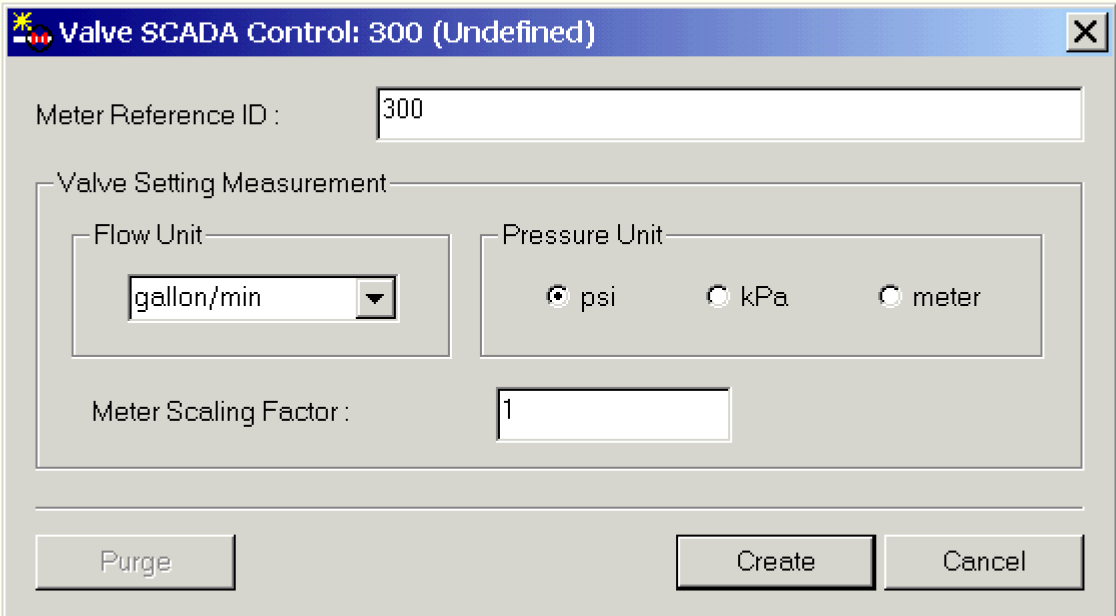
Click on any section below to learn more.



Valve Data

The SCADA configuration dialog box for valves is shown below. Configuration is defined for valve settings (pressure and/or flow).

Click on any section below to learn more.



SCADA Measurement

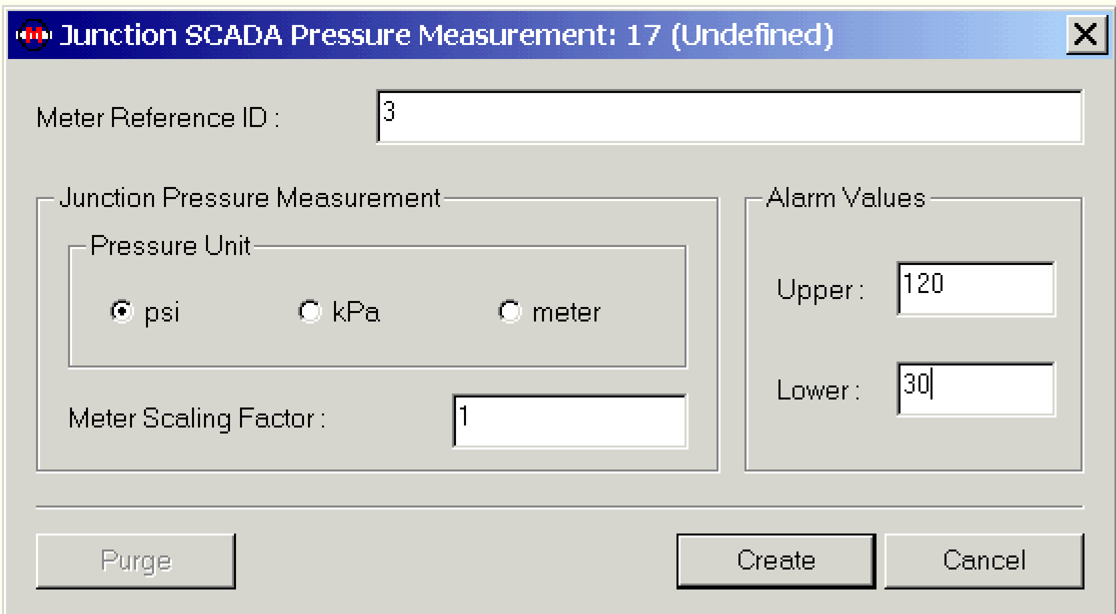
Prior to running the SCADA interface simulator, InfoWater must first be configured to provide a matching link between the network elements and the SCADA meter references; to define the format and units of measurement data, and to optionally specify the lower and upper limits for alarm settings.

To enable this feature, choose the SCADA Measurement icon under the Attribute Browser. The following represent the SCADA

Measurement dialog boxes for each of the two InfoWater data elements types.

Junction Node Pressure Measurement Data

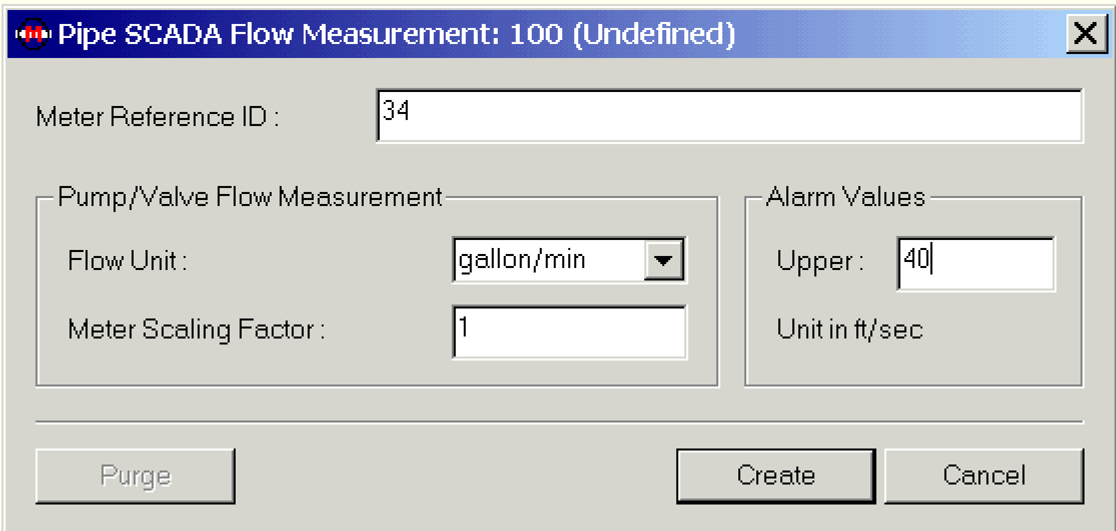
The Junction SCADA Pressure Measurement dialog box is shown below. Configuration is required for all the junctions where a comparison between modeled and measured pressure values is desired.



Note: Junction node demands are updated according to the SCADA time (specified in the *[TIME]* section of the SCADA Measurement Input file) based on the multiplication (usage) factor in the DEMAND Pattern definition. For example, if the Pattern Start Time is 6:00 AM and the SCADA measurement time is 13:00, then all junction demands will be multiplied by the 8th factor (assuming a one hour pattern time step) of their associated pattern IDs.

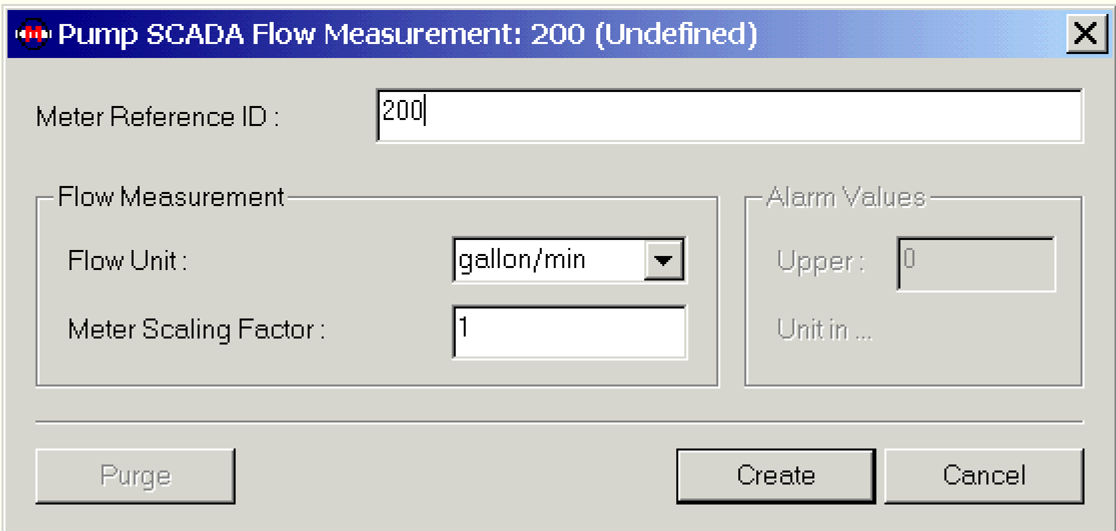
Pipe Flow Measurement Data

The Pipe SCADA Flow Measurement dialog box is shown below. Configuration is required for all those pipes where a comparison between modeled and measured flows is desired.



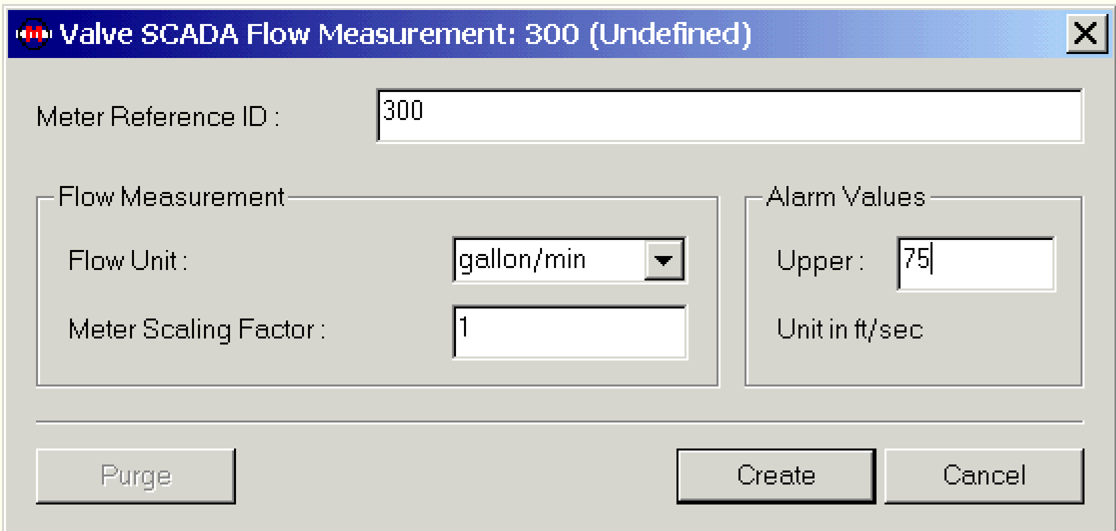
Pump Flow Measurement Data

The Pump SCADA Flow Measurement dialog box is shown below. Configuration is required for all those pumps where a comparison between modeled and measured flows is desired.



Valve Flow Measurement Data

The Valve SCADA Flow Measurement dialog box is shown below. Configuration is required for all those control valves where a comparison between modeled and measured flows is desired.



One-Time SCADA Simulation Run

Choosing the SCADA tab from the Run Manager and then selecting the Run icon to run the model. This performs an immediate one-time read of the most recent SCADA data file and performs a one-time steady-state analysis of the network using information from that SCADA file.

The InfoWater Run Manager is used to specify the name of the SCADA input file to be processed by InfoWater. It is also used to define the start time of all patterns, specify the demand adjustment method, apply a demand adjustment factor and to specify if, and how frequent, automatic runs are desired.

Automatic Run

Choosing the Automatic Run option at the bottom of the SCADA tab from the Run Manager dialog box. This performs a steady-state analysis of the network on a regular basis, first reading the SCADA data file as defined by the user, and then runs a steady-state simulation with the information in the latest version of the SCADA data file at whatever time interval specified by the user.

Note: When choosing this option, each SCADA simulation run creates a set of standard hydraulic simulation results that overwrites the previous results from earlier SCADA simulation runs. If you wish to save the results from a series of SCADA simulations, be sure to save the desired output sources (sets of simulation result data – in this case storing SCADA simulation results) for later retrieval. You can save output sources from the Run Manager.)

---

1. To stop an automatic SCADA run, open the Run Manager, choose the SCADA tab, and then choose the “Stop” button at the bottom of the Run Manager.
2. You may perform any desired task while the automatic SCADA run is initiated. You may modify the network (graphics and attributes), may change model data such as demands, etc. However, be careful not to delete those network components referenced as part of the SCADA run definition.
3. At the user-specified time interval, InfoWater will read the SCADA data file and will automatically initiate the simulation. At that time, normal InfoWater operations will be halted, allowing for the simulation engine to calculate results. InfoWater will use

the most current network definition (graphics, attributes, modeling parameters). Upon completion of the simulation, control of InfoWater is returned to the user.

4. You may use any of InfoWater's data review functions and capabilities to review and analyze the results of the SCADA simulation.

Other Related Topics - Running a Model**,** Real Time Data Connection Dialog Box**,** Real Time Data Connection Methodology**,** SCADA Interface**,** SCADA Modeling Dialog Box**,** SCADA System Data Edit

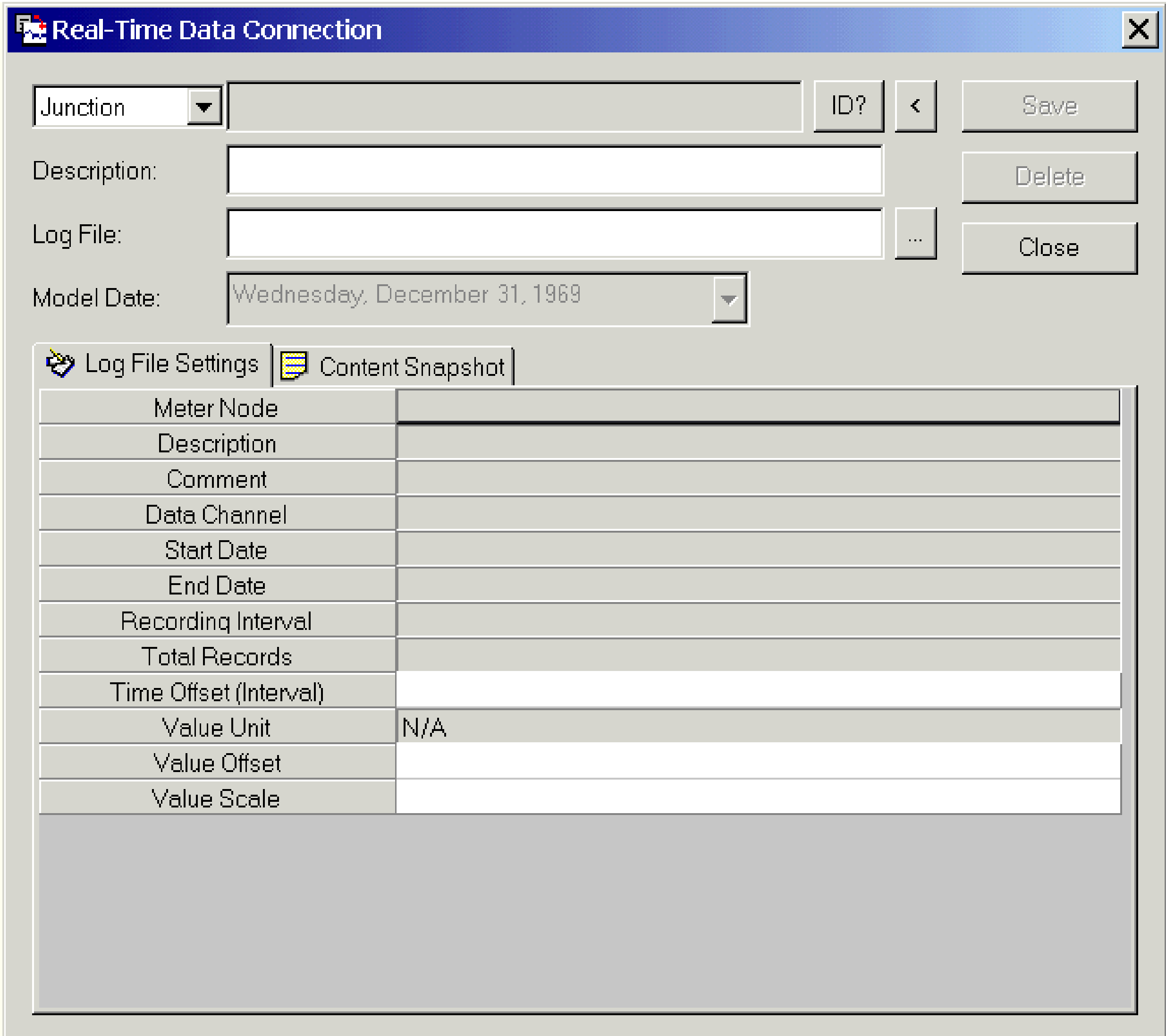
---

Real Time Data Connection Dialog Box

The Real-Time Data Connection allows the user to associate recorded data files (from pressure loggers or SCADA downloads) which can span a number of days. When conducting EPS simulations, you are able to compare a set of recorded data against simulated data. This feature is especially helpful during the calibration of a model, as it allows the user to select a particular day to compare a model's simulation results. An example may include a peak-demand day, to which a model is being calibrated. InfoWater stores the date selected when associating the data so that a particular day can be selected for all locations to assist in the calibration process.

To run the Real-Time Data Connection feature, from the **InfoWater Control Center -> InfoWater** button **-> Edit** menu, select **Real-Time Data Connection** to display the following dialog box. To learn more about the Real Time Data Connection methodology click here.

Click on any section below for more information.



Other Related Topics - Running a Model, Real Time Data Connection Methodology, SCADA Integration, SCADA Interface, SCADA Modeling Dialog Box, SCADA System Data Edit

Real Time Data Connection Methodology

The Real-Time Data Connection allows the user to associate recorded data files (from pressure loggers or SCADA downloads) which can span a number of days. When conducting EPS simulations, you are able to compare a set of recorded data against simulated data.

This feature is especially helpful during the calibration of a model, as it allows the user to select a particular day to compare a model's simulation results. An example may include a peak-demand day, to which a model is being calibrated. InfoWater stores the date selected when associating the data so that a particular day can be selected for all locations to assist in the calibration process.

Methodology

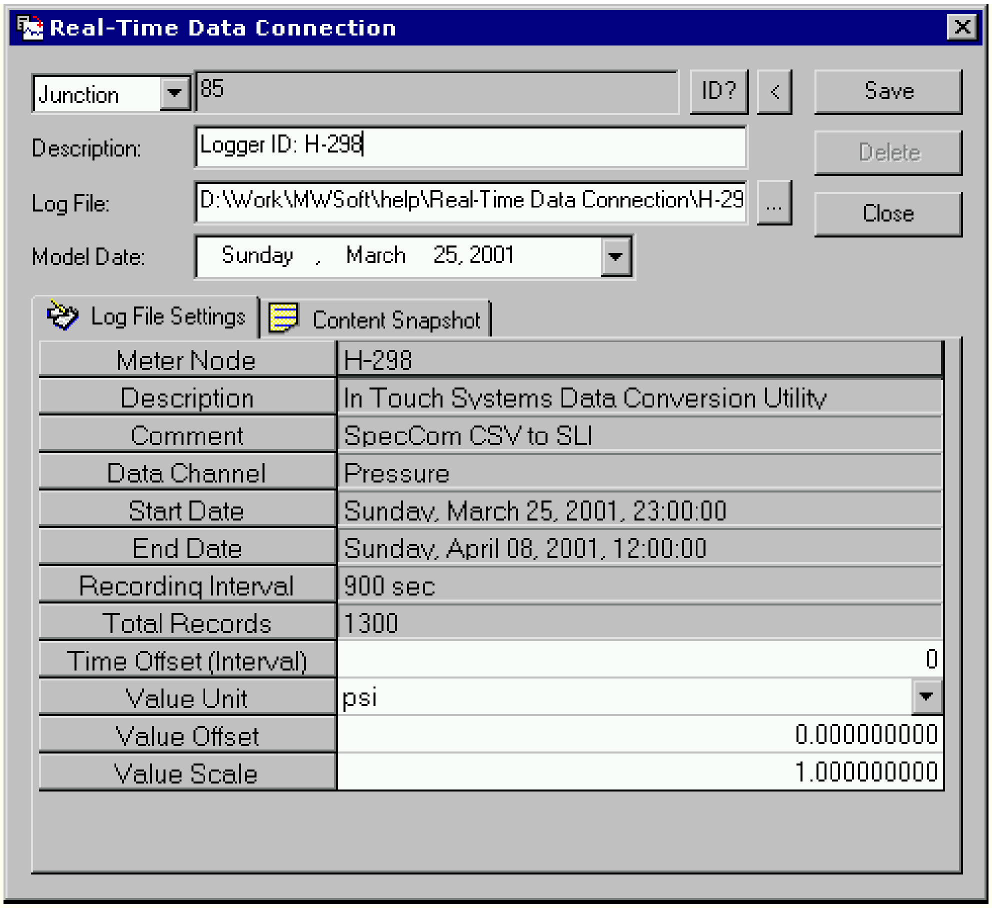
To run the Real-Time Data Connection feature of InfoWater do the following:

Associating Data with the real Time Data Connection Feature

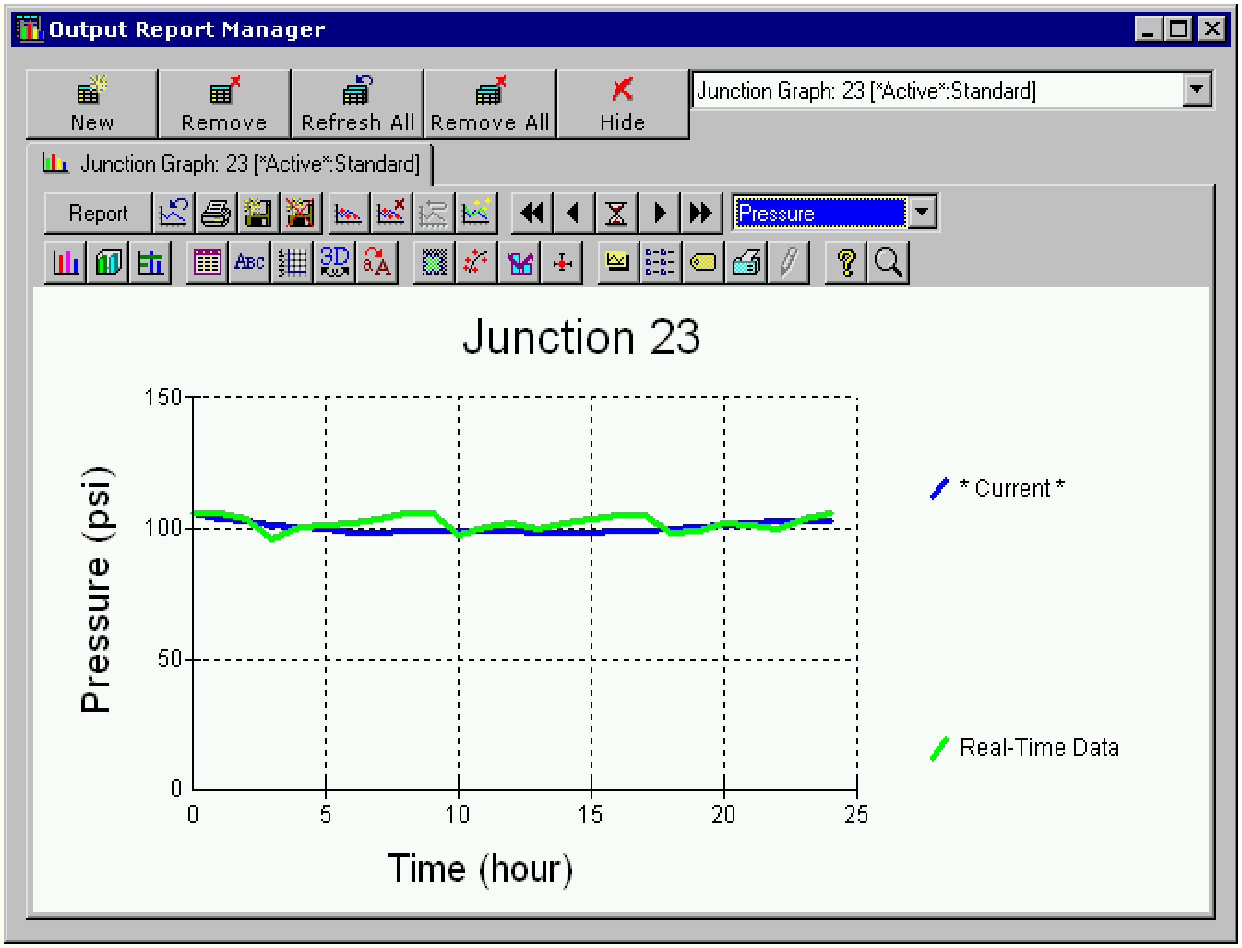
- From the **InfoWater Control Center -> InfoWater** button **-> Edit** menu, select **Real-Time Data Connection** to launch the Real Time Data Connection dialog box:
- Choose the **Model Element Type.**
- Select the **ID** of the network element you wish to assign data to and enter an optional **Description** for the data element. Enter the **Model Date** and specify the **Log File**. Finally enter all the **Log File Settings** in the Real Time Data Connection dialog box.

Comparing simulation data to recorded data:

- To use data that has been associated with a particular model element, open the Graph Manager.
- Create a graph, such as a Junction Graph for an element known to have Real-Time Data connected to it.
- If you have selected a Junction, then select the Pressure Graph and click on the Import Data button .
- If data has been associated through the Real-Time Data Connection, a prompt will appear asking if you wish to view the comparison data. Press the Yes button.
- You will be presented with the Real-Time Data Connection dialog. If you wish to view the data based on the settings contained in the file, select the Apply button. You can alter the Data Calibration settings, and then press Apply to view the data with these changes. Doing this will not alter the data contained in your data file.
- The data that is contained within the file for the date that has been specified will be imported into the graph. Refer to the figure below to see an example of this feature operating.



- The following figure illustrates an example of a graph that incorporates Real-Time data.



Other Related Topics - Running a Model, Real Time Data Connection Dialog Box, SCADA Integration, SCADA Interface, SCADA Modeling Dialog Box, SCADA System Data Edit

---

SCADA Interface

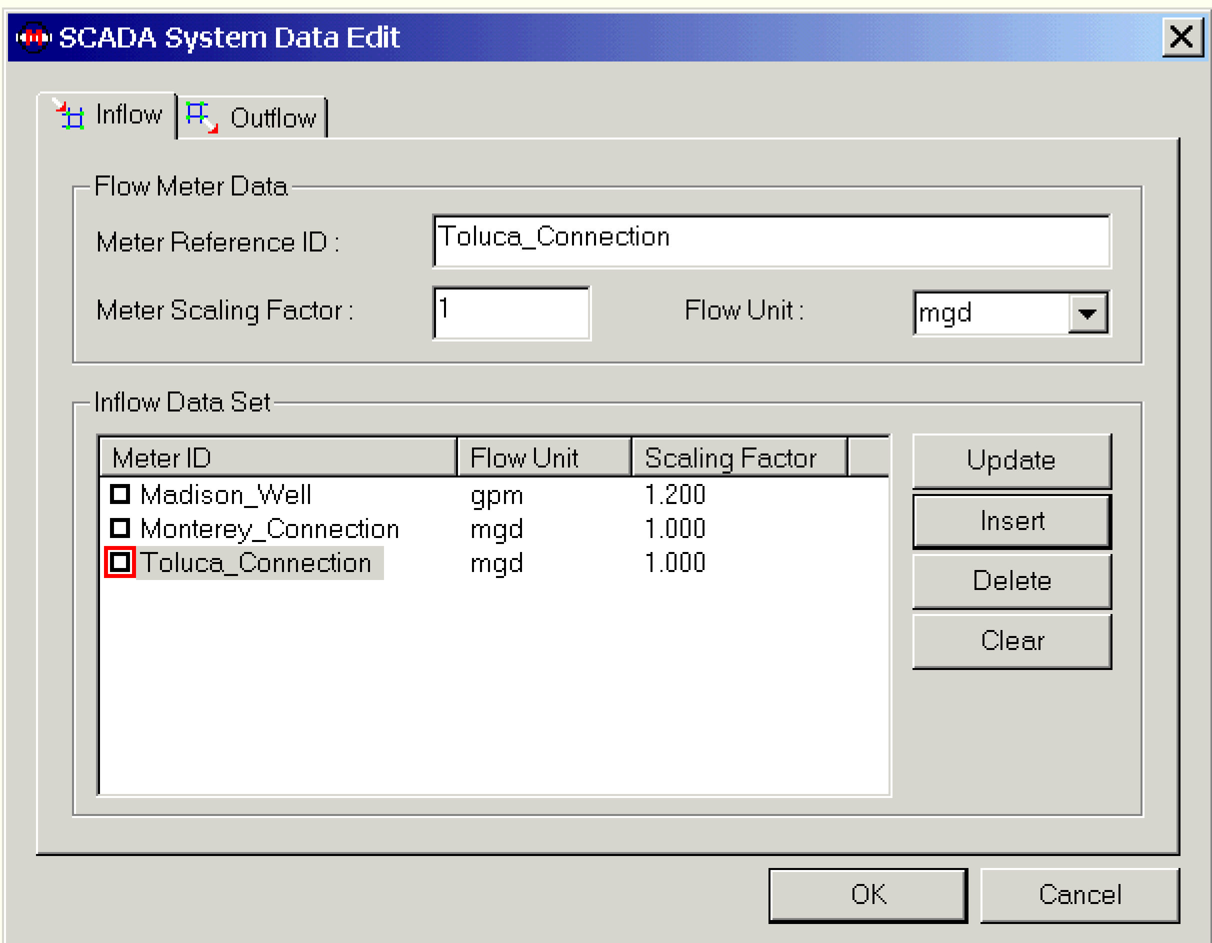
A Supervisory Control and Data Acquisition system (SCADA) stores information useful for network modeling applications. InfoWater provides a comprehensive SCADA interface module that allows for direct model updates from SCADA and thereafter generating comparisons between measured data and modeled results.

Click here to learn more about integrating SCADA information and modeling a SCADA simulation.

SCADA System Data Edit

Junction node demands are updated according to the SCADA time (specified in the [TIME] section of the SCADA Measurement Input file) based on the multiplication (usage) factor in the DEMAND Pattern definition. For example, if the Pattern Start Time is 6:00 AM and the SCADA measurement time is 13:00, then all junction demands will be multiplied by the 8th factor (assuming a one hour pattern time step) of their associated pattern IDs.

If the system inflow/outflow measurement option is chosen, the inflow/outflow measurement meters should be configured using the SCADA System Edit dialog box

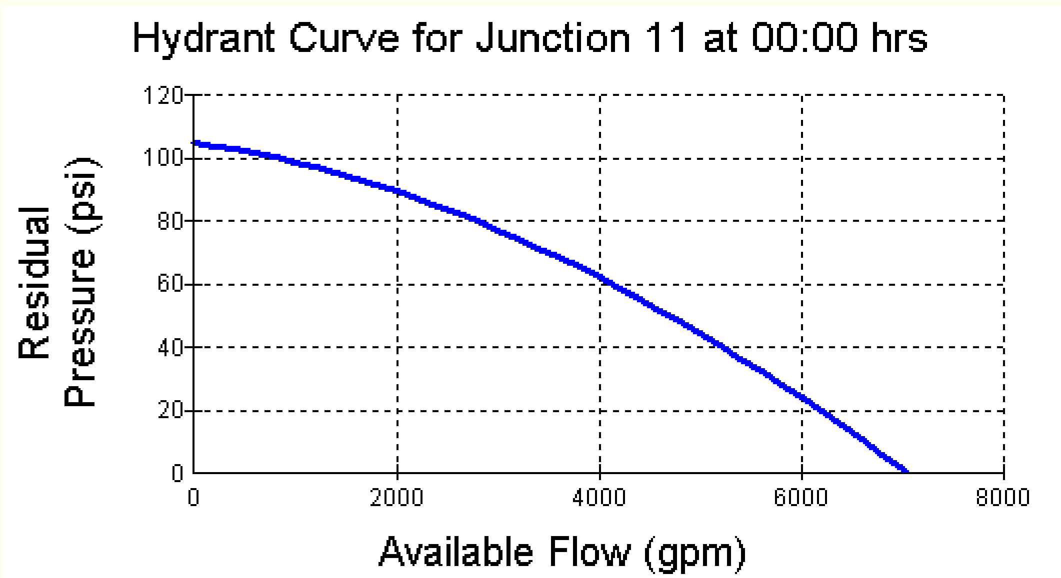


Other Related Topics - Running a Model**,** Real Time Data Connection Dialog Box**,** Real Time Data Connection Methodology**,** SCADA Integration**,** SCADA Interface**,** SCADA Modeling Dialog Box

Hydrant Curve

Hydrant curve graphs always show pressure on the Y-axis and flow on the X-axis. The range of flow displayed is from zero flow to the maximum flow at zero pressure. You can select the simulation timestep for which the hydrant curve will be calculated and displayed.

The figure below depicts an example fire hydrant curve (as seen after a hydrant curve simulation or through the graphing tool available within the Output Report Manager.)



Using the Hydrant Curve feature in InfoWater

Hydrant Curves can be used in InfoWater for various purposes :

- In conjunction with Fire-Flows, Hydrant Curves will provide an estimate of how much flow one can get from a Hydrant. From the graph above, one can say that the maximum available flow from the hydrant possible is 7,000 gpm.
- Also "on the fly" visualization can be achieved to see what the expected Hydrant flow would be when maintaining specific residual pressures at the Hydrant. For instance from the graph above one can predict a flow of around 6,000 gpm when the Hydrant maintains a pressure of 20 psi.
- It is a very powerful tool in Hydrant design and is recommended to be used in conjunction with the InfoWater Fire-Flow analysis.

Methodology

To run a Hydrant Curve analysis, do the following:

- From the **InfoWater Control Center -> InfoWater** button -> **Tools** launch the **Run Manager** dialog box.
- Click on the Hydrant Curve tab to conduct the Hydrant Curve analysis.
- Specify the **Target Junction ID** or click on the **Browse** button ... to graphically select your Hydrant from the Map. Note that Hydrants are modeled as Junctions in InfoWater and hence a Hydrant analysis may only be performed on Junctions.
- Specify the **Normal Hydrant flow** or Average Flow that you would expect from the hydrant.
- Specify the **Report Flow Increment**. InfoWater will calculate the hydrant residual pressure at each flow increment that you specify.
- Select the **Time** for the Hydrant Curve analysis for Extended Period Simulations by clicking on the Select Time button.
- Click on the **Run** icon to run the Hydrant Analysis.
- After a successful simulation, InfoWater will launch the Output Report manager and will display the Hydrant Curve automatically.
- Review your results and click on **Hide** on the Output report manager to close out of your Hydrant Curve results.

Other Related Topics - Running a Model, Hydrant Curve Dialog Box

System Curve

System head curves show the relationship between system head and flow capacity for a junction node at different time ranges. This relationship represents the variation in total dynamic head against which pumps will be required to operate under various flow conditions throughout the network. While the user can select any junction node to evaluate - this feature is provided to assist the engineer with determining the expected head curve for delivering a minimum fire flow to a given node that is furthest from a pumping facility.

Methodology

Follow the steps below to run a System Curve Analysis:

- To develop a system curve for a specific node, from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Run Manager** and select the **System Curve** tab.
- Choose the junction that you want to evaluate the System Curve for either by entering the **Target Junction ID** or by clicking on the Browse button you may graphically select the target junction .
- Specify the **Curve Type** and the **Maximum Flow** that you expect usually the highest fire demand expected at the junction. You may even specify the number of data points.
- You must specify the simulation time periods (up to three) at which the curves will be evaluated.
- Click on the **Run** icon  in your **Run Manager** dialog box to run the System Curve analysis.
- Your Model has now been run. After a successful run InfoWater will automatically launch the **Output Report manager** and display the System Curve. For more information on System Curve click here.

Other Related Topics - Running a Model, System Curve Dialog Box

---

Pressure Dependent Demand

Pressure dependent demand simulations allow the user to create a hydraulic model that more accurately reflects "real-world" conditions. In some water systems, as pressure drops, the ability to deliver the modeled demand also decreases with respect to a decrease in system pressure. Knowing this, the user can assign a demand vs. pressure curve for selected nodes in the system.

Methodology

To run a pressure dependent demand model, perform the following procedure:

- Using the Domain Manager, create a domain of the system junctions for which a pressure dependent demand model is desired. Once these nodes have been added to a domain, close the Domain Manager dialog box.
- From the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Run Manager**. Once the Run Manager dialog box is initiated, choose the Standard tab and place a check in the "Run Pressure Demand Simulation" check box.
- Specify the Characteristic Curve from the **Curve ID** drop down box. If you have not created a characteristic curve, click on the [...] icon to create one.
- Once a Characteristic Curve has been specified for those nodes in the present domain set, click on the **Run** icon at the top of the **Run Manager** dialog box.
- Close the Run Manager dialog box by clicking the **OK** button.
- (Optional) Open the **Domain Manager** and click the Reset button to clear the present domain from the InfoWater display.

Viewing the Results

To view the results of the Pressure Dependent Demand simulation:

- From the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Output Report Manager**.
- With the Output Report Manager dialog box open, select the **New** icon.
- From the Output Report & Graph dialog box, highlight *Active*.Standard as the Output Source and select the **Junction Report** from the Report tab. Click the **Open** button when complete.
- You will now see the output report for the nodes in the domain set with an additional data field showing (% Supply) as a percentage of the modeled demand. A value of 100% means that the system was able to deliver 100% of the modeled demand to that analysis node. A value of 50% means that the system was only able to deliver 50% of the modeled demand to that analysis node.

Note - Due to the hydraulic nature of a pressure demand analysis, the following nodes are not included in the model simulation:

- Nodes immediately downstream of tanks.
- Source nodes that are modeled with a negative value to simulate supply to the hydraulic system.

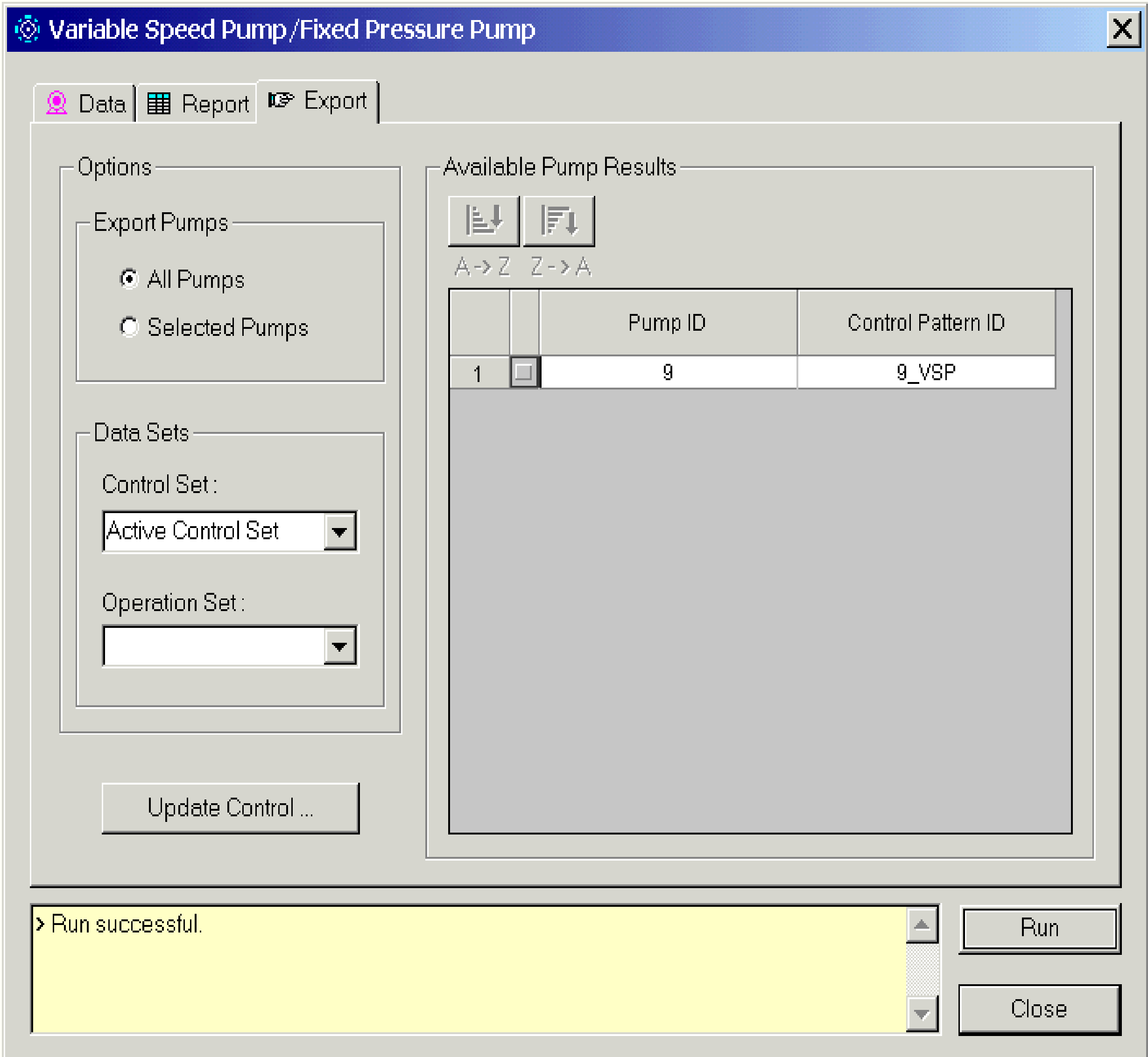
Other Related Topics - Running a Model, Pressure Dependent Dialog Box

Variable Speed Pump Export

For an Extended Period Simulation (EPS), InfoWater automatically computes the pump speed pattern to maintain the desired target pressure (which can be defined as a time varying pattern) at the **Target Junction Node**. This pattern can also be exported for use with the standard variable speed pump (manual) operation.

Once the speed pattern has been created and exported to the selected pump, a Standard hydraulic simulation via the Run Manager will then need to follow the VSP Analysis in order to adequately determine pump behavior and pump operating costs. To learn more about the VSP methodology click here.

Click on any portion below to learn more.

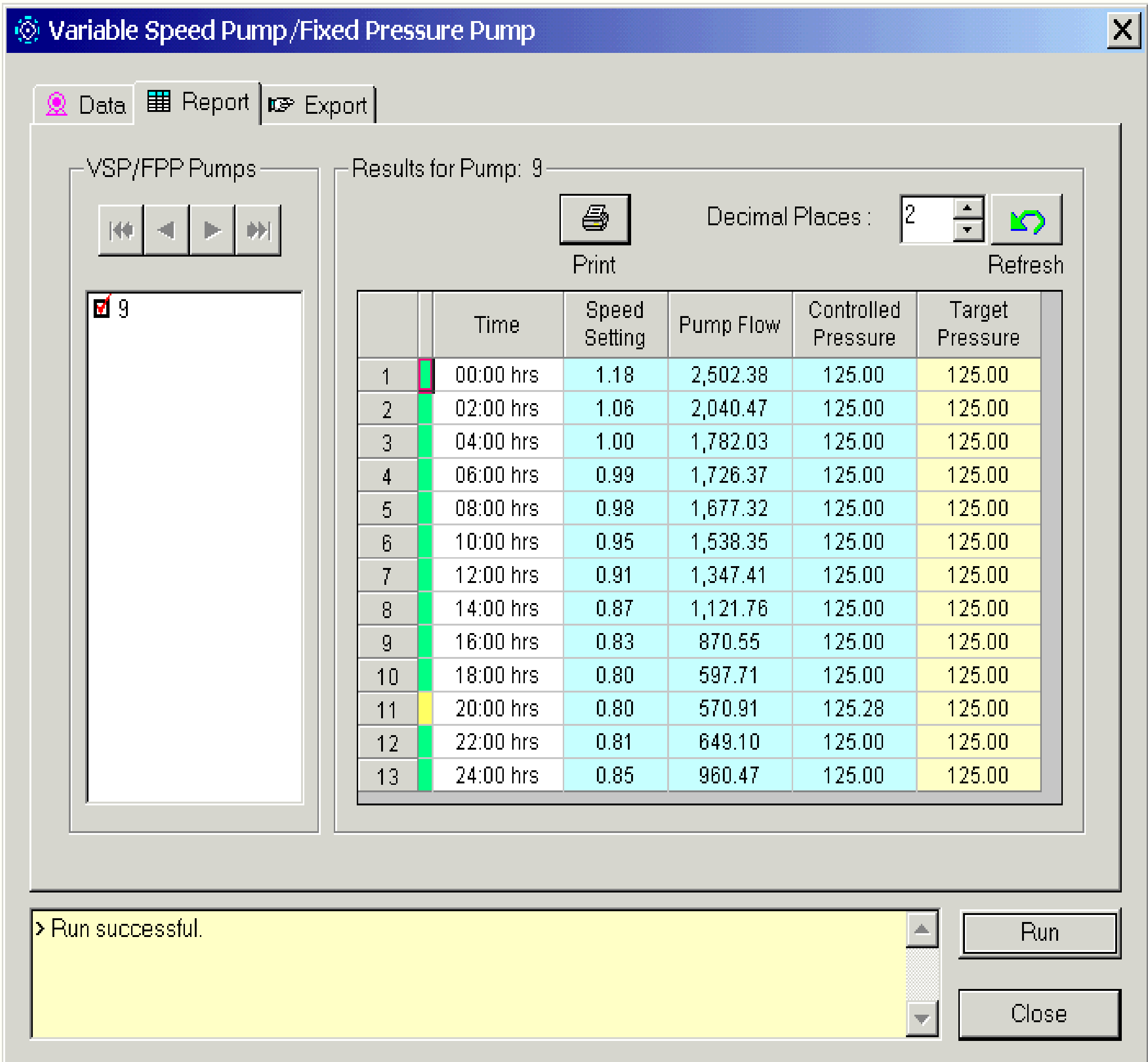


Other Related Topics - [Running a Model](), [Variable Speed Pump Dialog Box](), [Variable Speed Pump Report tab](), [Variable Speed Pumps Methodology]()

Variable Speed Pump Report

Once all the data are specified in the Data section of the Variable Speed Pump analysis, click the RUN button to initialize the VSP Analysis. The results are then displayed as shown in the dialog box below. The red color indicates that the pump is off while the green represents normal pump operation and the corresponding results (speed setting, pump flow and controlled pressure) are shown in the associated columns. To learn more about the VSP methodology click here.

Click on any portion below to learn more.



	Time	Speed Setting	Pump Flow	Controlled Pressure	Target Pressure
1	00:00 hrs	1.18	2,502.38	125.00	125.00
2	02:00 hrs	1.06	2,040.47	125.00	125.00
3	04:00 hrs	1.00	1,782.03	125.00	125.00
4	06:00 hrs	0.99	1,726.37	125.00	125.00
5	08:00 hrs	0.98	1,677.32	125.00	125.00
6	10:00 hrs	0.95	1,538.35	125.00	125.00
7	12:00 hrs	0.91	1,347.41	125.00	125.00
8	14:00 hrs	0.87	1,121.76	125.00	125.00
9	16:00 hrs	0.83	870.55	125.00	125.00
10	18:00 hrs	0.80	597.71	125.00	125.00
11	20:00 hrs	0.80	570.91	125.28	125.00
12	22:00 hrs	0.81	649.10	125.00	125.00
13	24:00 hrs	0.85	960.47	125.00	125.00

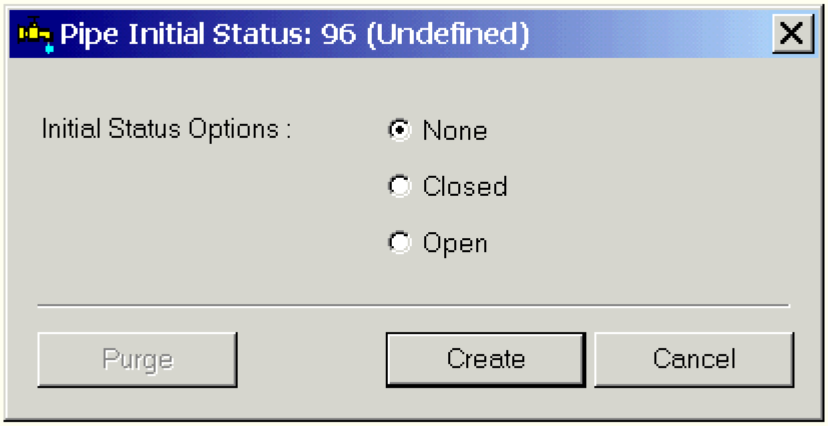
Other Related Topics - Running a Model, Variable Speed Pump Dialog Box, Variable Speed Pump Export tab, Variable Speed Pumps Methodology

---

Pipe Initial Status

Specify the Initial Status for Pipes over here. Select the pipe that you want to prescribe initial controls for on your InfoWater map by clicking on it and then launch this dialog box by choosing the **Initial Status** command from your **Attribute Browser -> Tools** icon  .

Click on any section below to learn more.

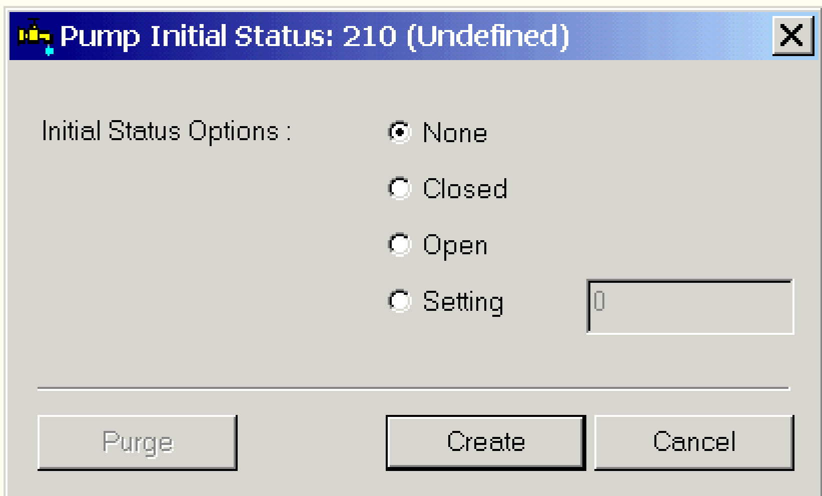


Other Related Topics - Controls Overview, Initial Status Methodology, Pump Initial Status, Rule Based Control Dialog Box, Rule Based Control Methodology, Simple Control Dialog Box, Simple Controls Methodology, Valve Initial Status

Pump Initial Status

Specify the Initial Status for pumps over here. Select the pump that you want to prescribe initial controls for on your InfoWater map by clicking on it and then launch this dialog box by choosing the **Initial Status** command from your **Attribute Browser -> Tools** icon  .

Click on any section below to learn more.

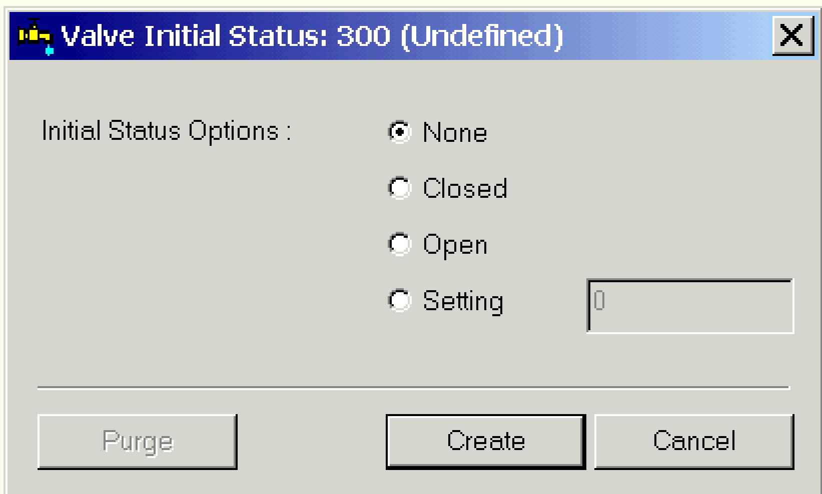


Other Related Topics - Controls Overview, Initial Status Methodology, Pipe Initial Status, Rule Based Control Dialog Box, Rule Based Control Methodology, Simple Control Dialog Box, Simple Controls Methodology, Valve Initial Status

Valve Initial Status

Specify the Initial Status for valves over here. Select the valve that you want to prescribe initial controls for on your InfoWater map by clicking on it and then launch this dialog box by choosing the **Initial Status** command from your **Attribute Browser -> Tools** icon  .

Click on any section below to learn more.



Other Related Topics - Controls Overview, Initial Status Methodology, Pipe Initial Status, Pump Initial Status, Rule Based Control Dialog Box, Rule Based Control Methodology, Simple Control Dialog Box, Simple Controls Methodology

Enable Logic Sets

Prior to a running a simulation that will include a logic set, ensure that the logic functions of InfoWater have been enabled. To do this, from the Operation tab under the View -> Table of Contents, click on the Simulation Options

folder and select the appropriate simulation options file. Once the Simulation Options dialog box is opened, click on the Rule Control box (checked) to enable the logic functionality of InfoWater.

Junction Report

Shows standard (hydraulic and water quality) simulation results at any simulation timestep for all junction nodes in tabular format. The node report displays one record for each node in the current InfoWater project. Junction node report columns include the node identifier, demand, elevation, grade, pressure, and water quality analysis variable.

The following variables are displayed on the Junction Report in the Output Report Manager for all or selected junctions. Click on any portion below for details:

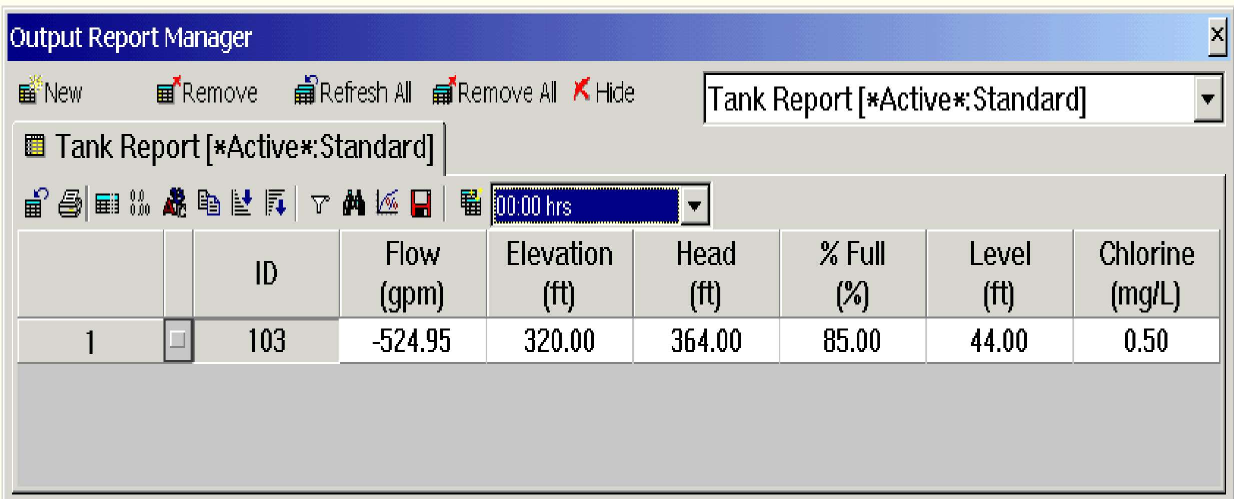
		ID	Demand (gpm)	Elevation (ft)	Head (ft)	Pressure (psi)	Chlorine (mg/L)
1	☐	1	0.00	95.00	99.96	2.15	0.50
2	☐	11	14.00	115.00	362.76	107.35	0.50
3	☐	13	14.00	127.00	362.80	102.17	0.50
4	☐	15	25.20	118.00	362.96	106.14	0.50
5	☐	17	56.00	150.00	363.50	92.51	0.50

Other Related Topics - Available Report Types, Demand Cost, Energy Cost, Energy Summary, Fire Flow, Fire Flow Design, Fire Flow Report Explanation, Pipe Report, Pump Report, Range Reports, Reservoir Report, SCADA Alarm, SCADA Flow, SCADA Pressure, Tank Report, Valve Report

Tank Report

Shows standard (hydraulic and water quality) simulation results at any simulation time step for all tanks in tabular format. This report displays one record for each tank in the current InfoWater project. Tank report columns include the node identifier, flow, elevation, head, percent full, level and water quality analysis results.

The following variables are displayed on the Tank Report in the Output Report Manager for all tanks. Click on any section for more information.



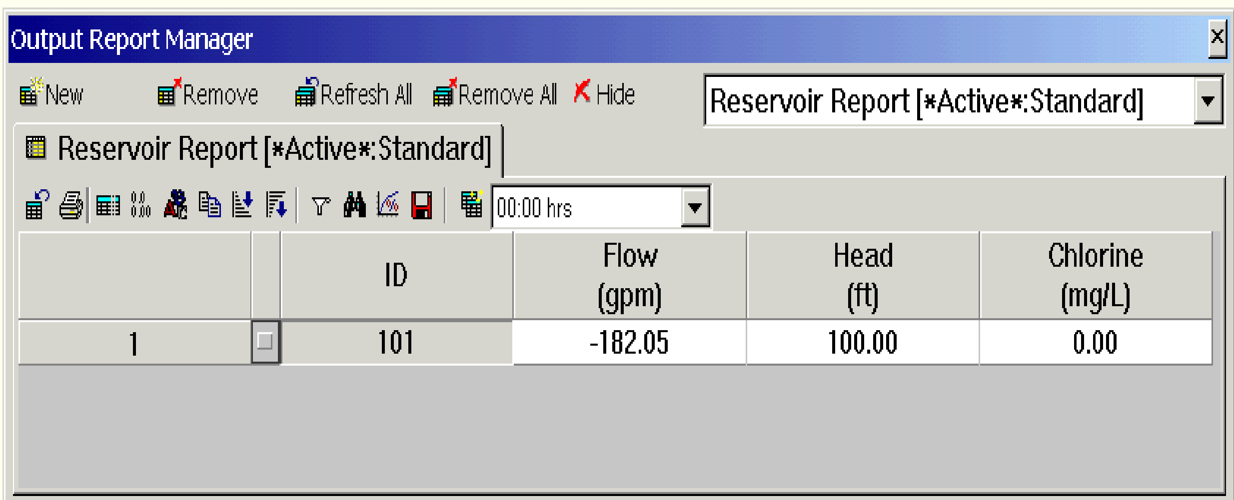
Other Related Topics - Available Report Types, Demand Cost, Energy Cost, Energy Summary, Fire Flow, Fire Flow Design, Fire Flow Report Explanation, Junction Report, Pipe Report, Pump Report, Range Reports, Reservoir Report, SCADA Alarm, SCADA Flow, SCADA Pressure, Valve Report

---

Reservoir Report

Shows standard (hydraulic and water quality) simulation results at any simulation time step for all reservoirs in tabular format. This report displays one record for each reservoir in the current InfoWater project. Reservoir report columns include the node identifier, flow, head and water quality analysis results.

The following variables are displayed on the Reservoir Report in the Output Report Manager for all tanks. Click on any section for more information.



Other Related Topics - Available Report Types, Demand Cost, Energy Cost, Energy Summary, Fire Flow, Fire Flow Design, Fire Flow Report Explanation, Junction Report, Pipe Report, Pump Report, Range Reports, SCADA Alarm, SCADA Flow, SCADA Pressure, Tank Report, Valve Report

Valve Report

Shows standard (hydraulic and water quality) simulation results at any simulation timestep for all control valves in tabular format. The valve report displays one record for each valve in the current InfoWater project. Valve report columns include the valve identifier, from and to nodes, diameter, flow, velocity, headloss, and water quality analysis variable.

The following variables are displayed on the Valve Report in the Output Report Manager for all or selected valves:

		ID	Diameter (in)	Elevation (ft)	Upstream Pressure	Downstream Pressure	Flow (gpm)	Velocity (ft/s)	Headloss (ft)	Chlorine (mg/L)
1		300	8.00	50.00	135.05	50.00	145.60	0.93	196.29	0.00

Other Related Topics - Available Report Types, Demand Cost, Energy Cost, Energy Summary, Fire Flow, Fire Flow Design, Fire Flow Report Explanation, Junction Report, Pipe Report, Pump Report, Range Reports, Reservoir Report, SCADA Alarm, SCADA Flow, SCADA Pressure, Tank Report

Pump Report

Shows standard (hydraulic and water quality) simulation results at any simulation timestep for all pumps in tabular format. The pump report displays one record for each pump in the current InfoWater project. Pump report columns include the pump identifier, from and to nodes, flow, headloss, and water quality analysis variable. Available net positive suction head (NPSH) and cavitation index are available for pumps that include an NPSH curve.

The following variables are displayed on the Pump Report in the Output Report Manager for all or selected pumps. Click on any portion for details:

		ID	Elevation (ft)	Upstream Pressure	Downstream Pressure	Flow (gpm)	Headloss (ft)	Avail.NPSH (ft)	Cavitation Index	Chlorine (mg/L)
1	☐	200	95.00	2.15	116.01	0.00	0.00	0.00	0.00	0.00
2	☐	210	95.00	2.15	116.01	182.05	262.77	38.04	0.00	0.00

Other Related Topics - Available Report Types, Demand Cost, Energy Cost, Energy Summary, Fire Flow, Fire Flow Design, Fire Flow Report Explanation, Junction Report, Pipe Report, Range Reports, Reservoir Report, SCADA Alarm, SCADA Flow, SCADA Pressure, Tank Report, Valve Report

Pipe Report

Shows standard (hydraulic and water quality) simulation results at any simulation timestep for all pipes in tabular format. The pipe report displays one record for each pipe in the current InfoWater project. Pipe report columns include the pipe identifier, from and to nodes, length, diameter, roughness, flow, velocity, headloss, headloss per 1000/feet, and water quality analysis variable. Total forward flow, total backward flow, and total net flow are available for pipes that include flow totalizers.

The following variables are displayed on the Pipe Report in the Output Report Manager for all or selected pipes. Click on any portion for more information.

	ID	From Node	To Node	Length (ft)	Diameter (in)	Roughness	Flow (gpm)	Velocity (ft/s)	Headloss (ft)	HL/1000 (ft/kft)	Total Forward	Total Reverse	Total Net Flow	Flow Reversal	Chlorin (mg/L)
1	10	11	9	415.00	8.00	125.00	37.87	0.24	0.02	0.05				0	0.50
2	100	13	15	685.20	8.00	125.00	-90.89	0.58	0.16	0.23				0	0.50
3	102	13	11	255.00	8.00	125.00	76.89	0.49	0.04	0.17				0	0.50
4	104	103	17	215.00	10.00	115.00	524.95	2.14	0.50	2.34				0	0.50

Other Related Topics - Available Report Types, Demand Cost, Energy Cost, Energy Summary, Fire Flow, Fire Flow Design, Fire Flow Report Explanation, Junction Report, Pump Report, Range Reports, Reservoir Report, SCADA Alarm, SCADA Flow, SCADA Pressure, Tank Report, Valve Report

Range Reports

The Range Report (for pipes, pumps, valves, junctions, tanks and reservoirs) displays the maximum, minimum and average values (and the difference between the maximum and minimum values) for the output variables during the entire extended period simulation period. One record is displayed for each component of the selected component type.

		ID	Max.Value (gpm)	Max.Time (hrs)	Min.Value (gpm)	Min.Time (hrs)	Average (gpm)	Difference (gpm)
1	☐	1	0.00	00:00	0.00	00:00	0.00	0.00
2	☐	11	22.40	01:00	5.60	20:00	14.22	16.80
3	☐	13	22.40	01:00	5.60	20:00	14.22	16.80
4	☐	15	40.32	01:00	10.08	20:00	25.60	30.24

Ranges allow the user to see, in a report format, the maximum and minimum values experienced at any element in the system over the EPS. For example, in a junction report, the user is able to get a print out of the maximum and minimum pressures experienced at any of the junction nodes. The range report is important when considering system fatigue.

Other Related Topics - Available Report Types, Demand Cost, Energy Cost, Energy Summary, Fire Flow, Fire Flow Design, Fire Flow Report Explanation, Junction Report, Pipe Report, Pump Report, Reservoir Report, SCADA Alarm, SCADA Flow, SCADA Pressure, Tank Report, Valve Report

Energy Cost

Displays results of an energy cost simulation for any time period. The Energy Cost report displays one record for each pump assigned energy data prior to running the energy management simulation. Results displayed for each pump include pump flow, head, useful power, efficiency, required power, and aggregated cost up to each simulation time period.

The following summary information is displayed on the Energy Cost Report in the Output Report Manager for all pumps assigned energy data. Click on any portion for more details.

		ID	Flow (gpm)	Head (ft)	Useful Power (hp)	Efficiency (%)	Required Power (kw)	Aggregated Cost ($)
1		200	0.00	0.00	0.00	0.00	0.00	0.00
2		210	182.05	262.77	12.08	73.57	12.25	0.00

Other Related Topics - Available Report Types, Demand Cost, Energy Summary, Fire Flow, Fire Flow Design, Fire Flow Report Explanation, Junction Report, Pipe Report, Pump Report, Range Reports, Reservoir Report, SCADA Alarm, SCADA Flow, SCADA Pressure, Tank Report, Valve Report

Energy Summary

Displays summary results of an energy cost analysis for the simulation period. One record for each pump assigned energy data prior to running the energy management simulation will be displayed in the Energy Summary report. Results displayed for each pump include average pump usage (as a percentage of time used over the simulation duration), average efficiency, average power, average power per unit, energy vs. volume ratio, total energy consumption cost, total demand cost, total operating cost, and total cost per unit volume.

The following summary information is displayed on the Energy Summary Report in the Output Report Manager for all pumps assigned energy data. Click on any portion for more details.

		ID	Usage (%)	Ave.Efficiency (%)	Ave.Powe r (kw)	Ave.Power/Flow (kw/gpm)	Energy/Volume (kw-hr/MG)	Energy Cost ($)	Demand Cost ($)	Total Cost ($)	Total Cost/Volume ($/MG)
1		200	71.61	78.17	22.63	0.06	1,000.06	37.75	55.67	93.42	171.99
2		210	100.00	78.27	15.39	0.06	1,000.79	22.69	27.34	50.03	135.55

Other Related Topics - Available Report Types, Demand Cost, Energy Cost, Fire Flow, Fire Flow Design, Fire Flow Report Explanation, Junction Report, Pipe Report, Pump Report, Range Reports, Reservoir Report, SCADA Alarm, SCADA Flow, SCADA Pressure, Tank Report, Valve Report

Demand Cost

The following variables are displayed on the Demand Cost Report in the Output Report Manager for all pumps assigned energy data and specifically, demand charge patterns.

Click on any portion below to learn more.

		ID	Charge Rate ($/max.kw)	Max.Power (kw)	Demand Charge ($)
1	☐	200	50.00	33.40	55.67
2	☐	210	50.00	16.40	27.34

Other Related Topics - Available Report Types, Energy Cost, Energy Summary, Fire Flow, Fire Flow Design, Fire Flow Report Explanation, Junction Report, Pipe Report, Pump Report, Range Reports, Reservoir Report, SCADA Alarm, SCADA Flow, SCADA Pressure, Tank Report, Valve Report

SCADA Flow

The SCADA Flow report displays the comparison between Measured and Modeled Flows. The Measured Flows are the flows that your SCADA system measured while the Modeled Flow is the value that the InfoWater model evaluated after running a SCADA analysis. The % difference will directly tell you how well your model is calibrated.

		ID	Measured Flow (gpm)	Modeled Flow (gpm)	% Difference
1	□	200	532.00	528.11	0.73
2	□	210	253.00	264.06	-4.37

Other Related Topics - Available Report Types, Demand Cost, Energy Cost, Energy Summary, Fire Flow, Fire Flow Design, Fire Flow Report Explanation, Junction Report, Pipe Report, Pump Report, Range Reports, Reservoir Report, SCADA Alarm, SCADA Pressure, Tank Report, Valve Report

SCADA Pressure

The SCADA Pressure Report displays the comparison between Measured and Modeled Pressures for SCADA elements. The **Measured Pressures** are the pressures that your SCADA system measured while the **Modeled Pressure** is the value that the InfoWater model evaluated after running a SCADA analysis. The **% Difference** will directly tell you how well your model is calibrated.

		ID	Measured Pressure (psi)	Modeled Pressure (psi)	% Difference
1		3	122.00	109.74	10.05

Other Related Topics - Available Report Types, Demand Cost, Energy Cost, Energy Summary, Fire Flow, Fire Flow Design, Fire Flow Report Explanation, Junction Report, Pipe Report, Pump Report, Range Reports, Reservoir Report, SCADA Alarm, SCADA Flow, Tank Report, Valve Report

SCADA Alarm

The SCADA Alarm table flags all the elements that display calculated values either greater than or less than the Upper Alarm values or Lower Alarm Values set during the SCADA initialization process. This provides you with a very powerful tool to monitor your system and locate problem areas in your network.

Click on any section below to learn more.

		Tanks > Upper Levels	Tanks < Lower Levels	Pressures > Upper Levels	Pressures < Lower Levels	Velocities > Upper Levels
1				3		

Other Related Topics - Available Report Types, Demand Cost, Energy Cost, Energy Summary, Fire Flow, Fire Flow Design, Fire Flow Report Explanation, Junction Report, Pipe Report, Pump Report, Range Reports, Reservoir Report, SCADA Flow, SCADA Pressure, Tank Report, Valve Report

Fire Flow

The first report is a standard fire flow simulation. This report includes static demand, static pressure, fire flow demand, residual pressure, available flow at the hydrant and pressure at the available flow. Click here for the Fire Flow Design Report. To learn more about conducting fire flows click here. To understand a little more about the Fire Flow reports click here.

The Fire flow report contains the same information, regardless if the Minimum Design Pressure design flow calculation is checked or not. Click on any portion for details.

		ID	Static Demand (gpm)	Static Pressure (psi)	Fire-Flow Demand (gpm)	Residual Pressure (psi)	Available Flow @Hydrant	Available Flow Pressure
1	☐	11	16.80	99.87	1,000.00	95.69	6,382.02	20.41
2	☐	13	16.80	94.69	1,000.00	90.14	5,592.59	20.31
3	☐	15	30.24	98.66	1,000.00	95.34	7,520.39	20.56
4	☐	19	100.80	96.65	1,000.00	90.50	4,933.36	20.23
5	☐	21	16.80	99.64	1,000.00	93.48	5,098.46	20.26

Other Related Topics - Available Report Types, Demand Cost, Energy Cost, Energy Summary, Fire Flow Design, Fire Flow Report Explanation, Junction Report, Pipe Report, Pump Report, Range Reports, Reservoir Report, SCADA Alarm, SCADA Flow, SCADA Pressure, Tank Report, Valve Report

Fire Flow Design Report

The Fireflow design report displays the final design flow. The contents of this report are different, depending on whether the Minimum Design Pressure at the bottom of the Fireflow tab of the Run Manager dialog box is checked or not checked. Click here for the Fire Flow Report. To learn more about conducting fire flows click here. To understand a little more about the Fire Flow reports click here.

With Minimum Design Pressure Unchecked (OFF)

When the minimum design pressure option is checked off, InfoWater will generate a fireflow design report that simply lists the critical node ID and its pressure during the standard fireflow simulation. The other fields are repeated from the standard fireflow report to aid the user in viewing the fireflow design report.

Click on any portion below for details:

		ID	Total Demand (gpm)	Critical Node ID	Critical Node Pressure (psi)	Available Flow @Hydrant (gpm)	Available Flow Pressure (psi)
1	☐	11	1,016.80	81	87.69	6,382.02	20.41
2	☐	13	1,016.80	81	87.86	5,592.59	20.31
3	☐	15	1,030.24	81	88.28	7,520.39	20.56
4	☐	19	1,100.80	81	87.78	4,933.36	20.23

With Minimum Design Pressure Checked (ON)

When the minimum design pressure option is checked on, InfoWater will generate a fireflow design report that determines the minimum pressures in the critical node searching range and returns a Design Flow to be used as a maximum available fire flow in order to maintain minimum pressures in the distribution system.

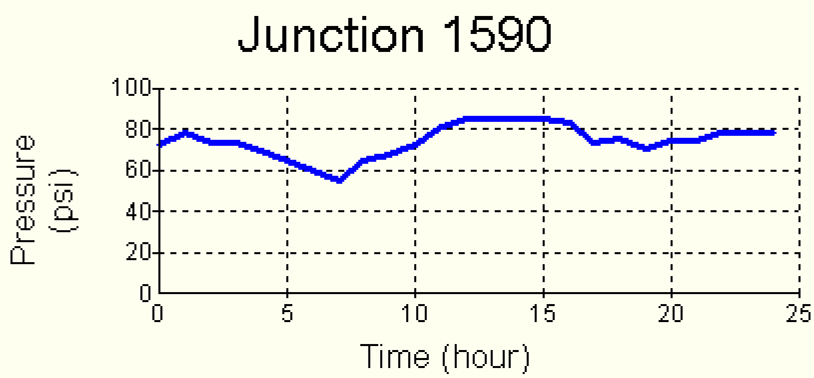
Click on any portion below for details:

		ID	Total Demand (gpm)	Critical Node 1 ID	Critical Node 1 Pressure	Adjusted Fire-Flow (gpm)	Available Flow @Hydrant	Critical Node 2 ID	Critical Node 2 Pressure	Adjusted Available Flow	Design Flow (gpm)
1	☐	11	1,016.80	81	87.69	7,206.23	6,382.02	11	20.41	6,401.00	6,401.00
2	☐	13	1,016.80	81	87.86	7,508.81	5,592.59	13	20.31	5,606.07	5,606.07
3	☐	15	1,030.24	81	88.28	8,128.25	7,520.39	15	20.56	7,551.85	7,551.85
4	☐	19	1,100.80	81	87.78	7,409.05	4,933.36	19	20.23	4,942.01	4,942.01

Other Related Topics - Available Report Types, Demand Cost, Energy Cost, Energy Summary, Fire Flow, Fire Flow Report Explanation, Junction Report, Pipe Report, Pump Report, Range Reports, Reservoir Report, SCADA Alarm, SCADA Flow, SCADA Pressure, Tank Report, Valve Report

Junction Graph

Displays simulation results for one junction node. The graph X-axis displays time in units defined with the Edit Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected junction node. Upon choosing this graph type you are prompted to choose a junction node. Upon selecting a node, the graph appears in the Output Report Manager window. Junction node graphs can display node demand, grade, pressure, or water quality analysis variable.

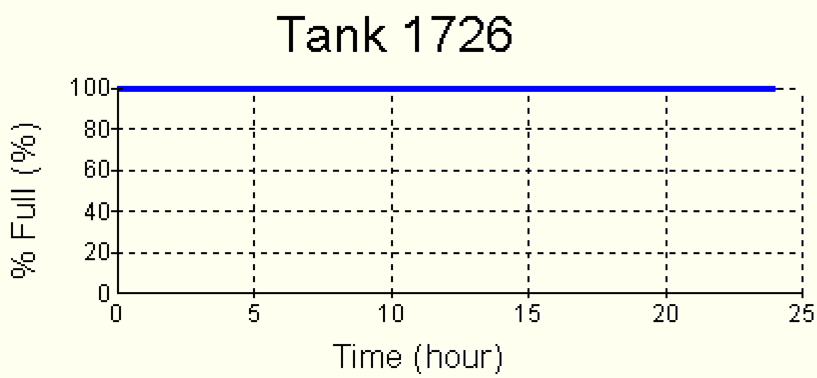


Note - To compare results for various junction, use the Junction Group Graph.

Other Related Topics - Available Graph Types, Average Reaction Rate, Energy Curve, Frequency, HGL Graph, Hydrant Curve, Junction Group Graph, Pipe Graph, Pipe Group Graph, Pump Curve, Pump Graph, Pump Group Graph, Reservoir Graph, Reservoir Group Graph, System Curve, System Demand, Tank Graph, Tank group Graph, Valve Graph, Valve Group Graph.

Tank Graph

Displays simulation results for one storage node. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected storage node. Upon choosing this graph type you are prompted to choose a storage node. Upon selecting a node, the graph appears in the Output Report Manager window. Tank node graphs can display node demand, head, pressure, percent full, or water quality analysis variable.

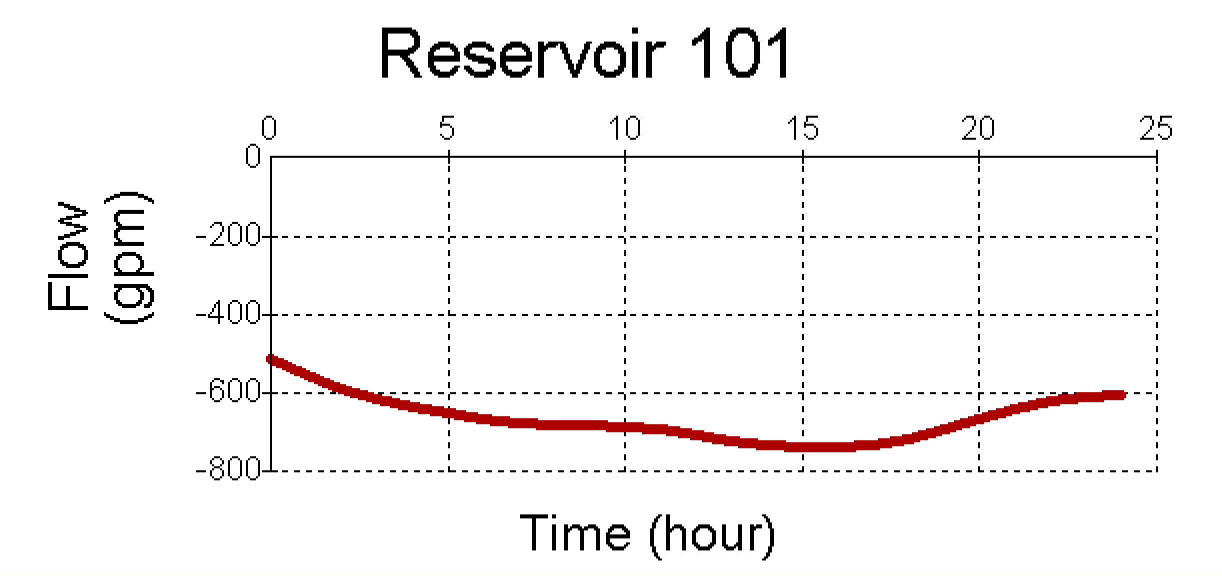


Note - To compare results for various tanks, use the Tank Group Graph.

Other Related Topics - Available Graph Types**,** Average Reaction Rate**,** Energy Curve**,** Frequency**,** HGL Graph**,** Hydrant Curve**,** Junction Graph**,** Junction Group Graph**,** Pipe Graph**,** Pipe Group Graph**,** Pump Curve**,** Pump Graph**,** Pump Group Graph**,** Reservoir Graph**,** Reservoir Group Graph**,** System Curve**,** System Demand**,** Tank group Graph**,** Valve Graph**,** Valve Group Graph**.**

Reservoir Graph

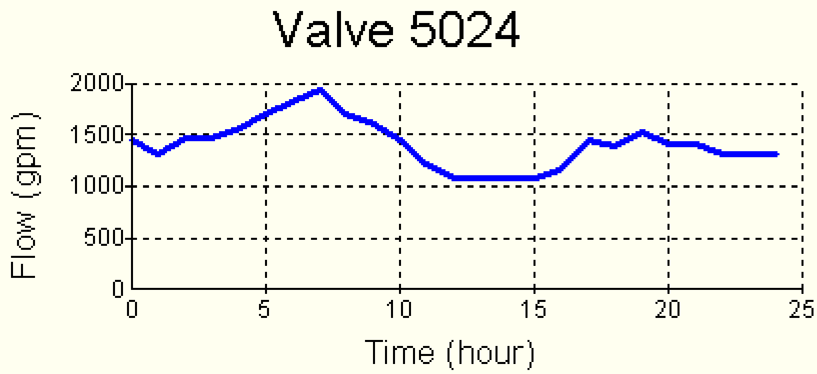
Displays simulation results for one reservoir. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected reservoir. Upon choosing this graph type you are prompted to choose a reservoir. Upon selecting the reservoir, the graph appears in the Output Report Manager window. Reservoir node graphs can display flow, head and/or water quality analysis variable.



Other Related Topics - Available Graph Types, Average Reaction Rate, Energy Curve, Frequency, HGL Graph, Hydrant Curve, Junction Graph, Junction Group Graph, Pipe Graph, Pipe Group Graph, Pump Curve, Pump Graph, Pump Group Graph, Reservoir Group Graph, System Curve, System Demand, Tank Graph, Tank group Graph, Valve Graph, Valve Group Graph.

Valve Graph

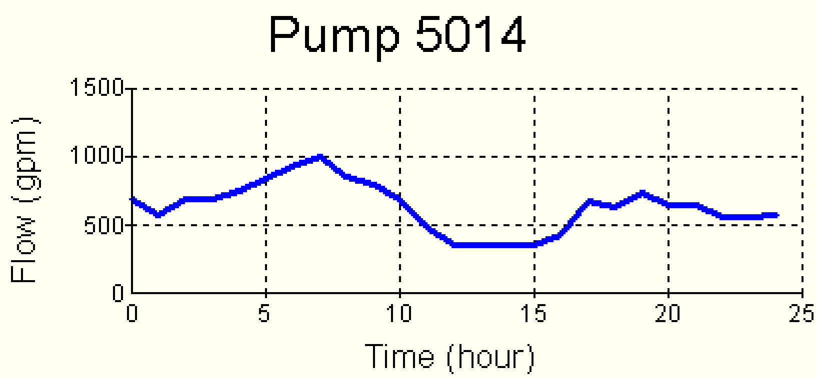
Displays simulation results for one control valve. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected valve. Upon choosing this graph type you are prompted to choose a valve. Upon selecting a valve, the graph appears in the Output Report Manager window. Valve graphs can display valve flow, velocity, headloss, or water quality analysis variable.



Other Related Topics - Available Graph Types, Average Reaction Rate, Energy Curve, Frequency, HGL Graph, Hydrant Curve, Junction Graph, Junction Group Graph, Pipe Graph, Pipe Group Graph, Pump Curve, Pump Graph, Pump Group Graph, Reservoir Graph, Reservoir Group Graph, System Curve, System Demand, Tank Graph, Tank group Graph, Valve Group Graph.

Pump Graph

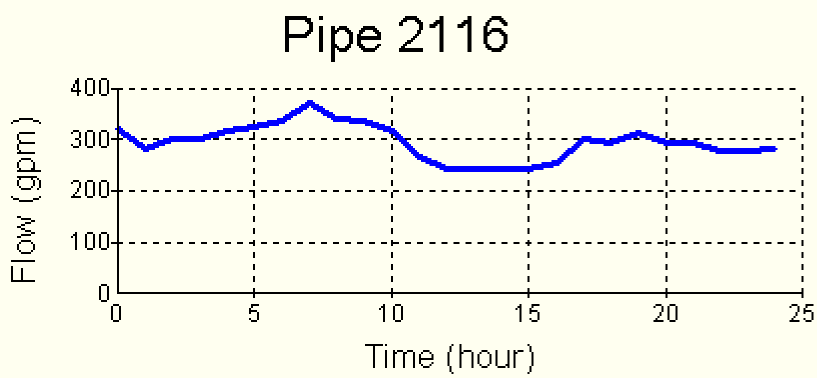
Displays simulation results for one pump. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected pump. Upon choosing this graph type you are prompted to choose a pump. Upon selecting a pump, the graph appears in the Output Report Manager window. Pump graphs can display flow, headloss, and water quality analysis variable. Available net positive suction head (NPSH) and cavitation index are available for pumps that include an NPSH curve.



Other Related Topics - Available Graph Types, Average Reaction Rate, Energy Curve, Frequency, HGL Graph, Hydrant Curve, Junction Graph, Junction Group Graph, Pipe Graph, Pipe Group Graph, Pump Curve, Pump Group Graph, Reservoir Graph, Reservoir Group Graph, System Curve, System Demand, Tank Graph, Tank group Graph, Valve Graph, Valve Group Graph.

Pipe Graph

Displays simulation results for one pipe. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected pipe. Upon choosing this graph type you are prompted to choose a pipe. Upon selecting a pipe, the graph appears in the Output Report Manager window. Pipe graphs can display pipe flow, velocity, headloss, headloss per 1000 feet (m), or water quality analysis variable.



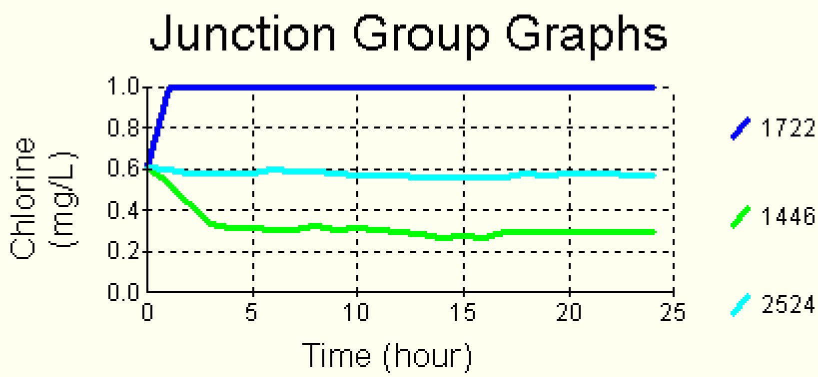
Note - To compare results for various pipes, use the Pipe Group Graph.

Other Related Topics - Available Graph Types**,** Average Reaction Rate**,** Energy Curve**,** Frequency**,** HGL Graph**,** Hydrant Curve**,** Junction Graph**,** Junction Group Graph**,** Pipe Group Graph**,** Pump Curve**,** Pump Graph**,** Pump Group Graph**,** Reservoir Graph**,** Reservoir Group Graph**,** System Curve**,** System Demand**,** Tank Graph**,** Tank group Graph**,** Valve Graph**,** Valve Group Graph**.**

Junction Group Graph

Displays simulation results for two or more junction nodes. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected junction nodes. Upon choosing this graph type you are prompted to choose two or more junction nodes. Upon selecting nodes, the graph appears in the Output Report Manager window. Junction Group graphs can display node demand, grade, pressure, or water quality analysis variable for the selected nodes.

Press the Enter key or the right mouse button to terminate the node selection process.



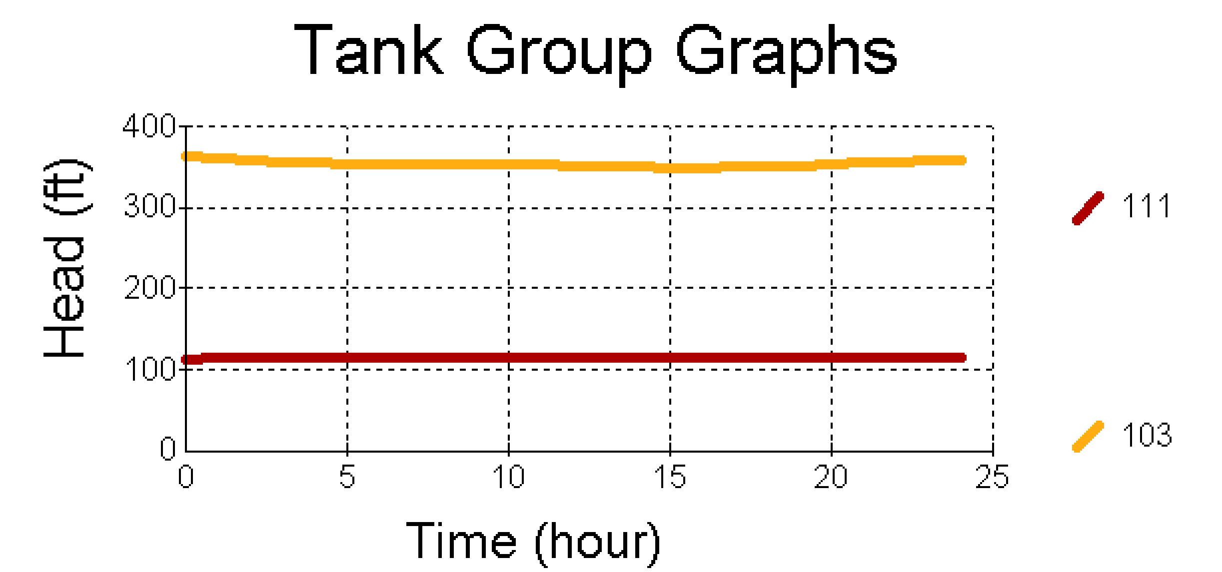
Note - To look at results for one Junction, use the Junction Graph.

Other Related Topics - Available Graph Types, Average Reaction Rate, Energy Curve, Frequency, HGL Graph, Hydrant Curve, Junction Graph, Pipe Graph, Pipe Group Graph, Pump Curve, Pump Graph, Pump Group Graph, Reservoir Graph, Reservoir Group Graph, System Curve, System Demand, Tank Graph, Tank group Graph, Valve Graph, Valve Group Graph.

Tank Group Graph

Displays simulation results for two or more storage nodes. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected storage nodes. Upon choosing this graph type you are prompted to choose two or more storage nodes. Upon selecting nodes, the graph appears in the Output Report Manager window. Tank Group graphs can display node demand, head, pressure, percent full, or water quality analysis variable.

Press the Enter key or the right mouse button to terminate the node selection process.



Note - To view results for one tank, use the Tank Graph.

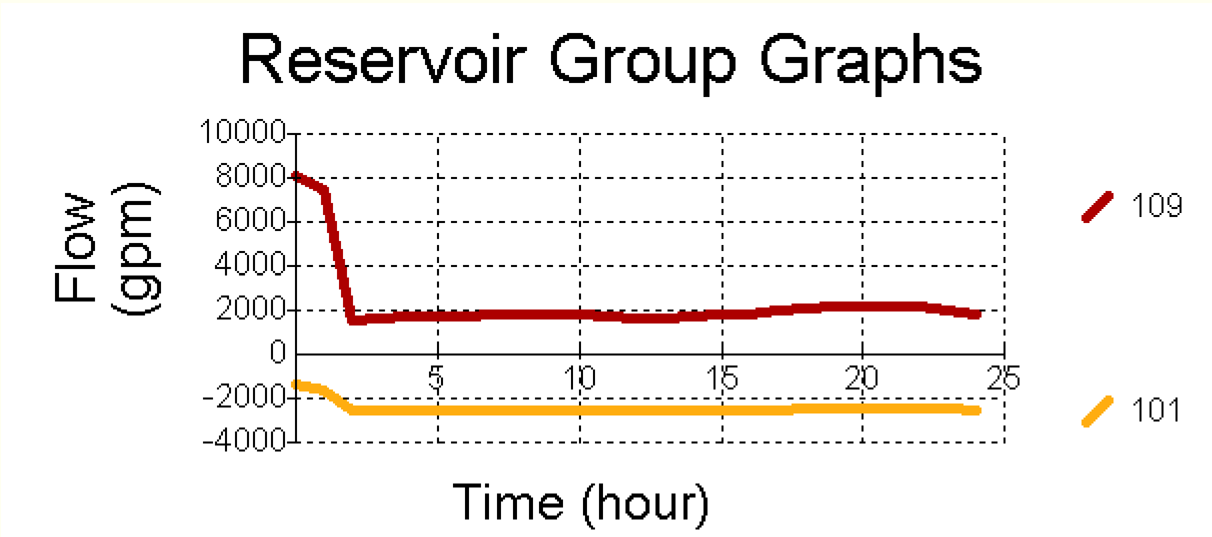
Other Related Topics - Available Graph Types, Average Reaction Rate, Energy Curve, Frequency, HGL Graph, Hydrant Curve, Junction Graph, Junction Group Graph, Pipe Graph, Pipe Group

Graph, Pump Curve, Pump Graph, Pump Group Graph, Reservoir Graph, Reservoir Group Graph, System Curve, System Demand, Tank Graph, Valve Graph, Valve Group Graph.

Reservoir Group Graph

Displays simulation results for two or more storage nodes. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected storage nodes. Upon choosing this graph type you are prompted to choose two or more storage nodes. Upon selecting nodes, the graph appears in the Output Report Manager window. Reservoir group graphs can display flow, head and/or water quality analysis variable.

Press the Enter key or the right mouse button to terminate the node selection process.

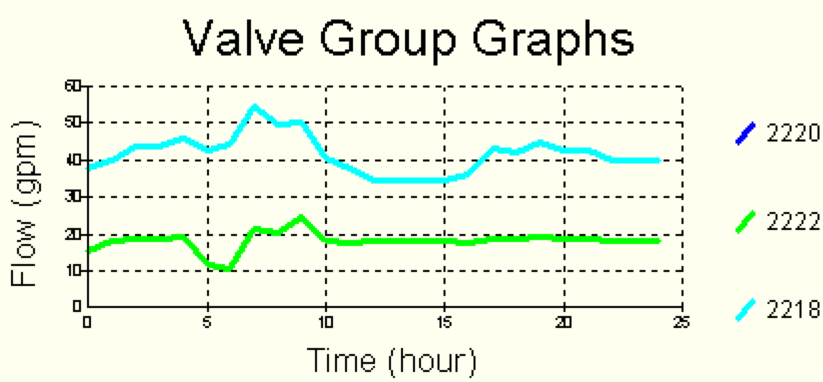


Other Related Topics - Available Graph Types, Average Reaction Rate, Energy Curve, Frequency, HGL Graph, Hydrant Curve, Junction Graph, Junction Group Graph, Pipe Graph, Pipe Group Graph, Pump Curve, Pump Graph, Pump Group Graph, Reservoir Graph, System Curve, System Demand, Tank Graph, Tank group Graph, Valve Graph, Valve Group Graph.

Valve Group Graph

Displays simulation results for two or more control valves. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected valves. Upon choosing this graph type you are prompted to choose two or more valves. Upon selecting a valve, the graph appears in the Output Report Manager window. Valve Group graphs can display valve flow, velocity, headloss, or water quality analysis variable.

Press the Enter key or the right mouse button to terminate the valve selection process.



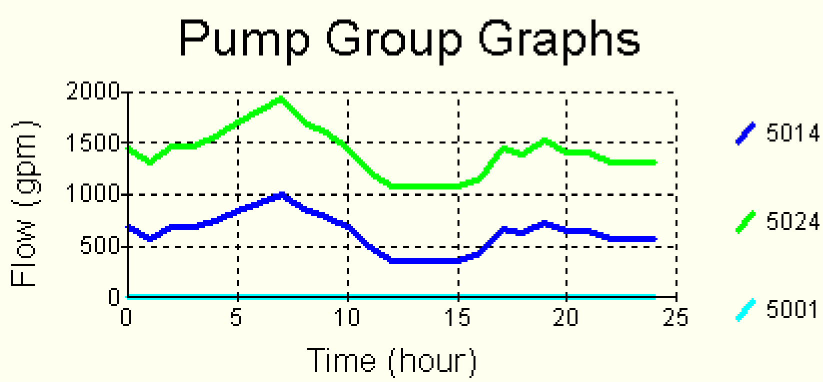
Note: In the example above, valves 2220 and 2222 share the same flow rates.

Other Related Topics - Available Graph Types, Average Reaction Rate, Energy Curve, Frequency, HGL Graph, Hydrant Curve, Junction Graph, Junction Group Graph, Pipe Graph, Pipe Group Graph, Pump Curve, Pump Graph, Pump Group Graph, Reservoir Graph, Reservoir Group Graph, System Curve, System Demand, Tank Graph, Tank group Graph, Valve Graph.

Pump Group Graph

Displays simulation results for two or more pumps. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected pumps. Upon choosing this graph type you are prompted to choose two or more pumps. Upon selecting pumps, the graph appears in the Output Report Manager window. Pump Group graphs can display flow, headloss, and water quality analysis variable. Available net positive suction head (NPSH) and cavitation index are available for pumps that include an NPSH curve.

Press the Enter key or the right mouse button to terminate the pump selection process.



Note - In the example above, pump 5001 is off for the entire duration of the simulation.

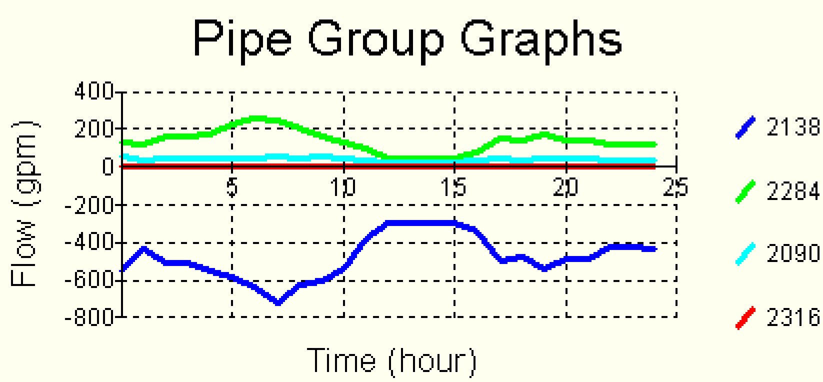
Other Related Topics - Available Graph Types, Average Reaction Rate, Energy Curve, Frequency, HGL Graph, Hydrant Curve, Junction Graph, Junction Group Graph, Pipe Graph, Pipe Group Graph, Pump Curve, Pump Graph, Reservoir Graph, Reservoir

Group Graph, System Curve, System Demand, Tank Graph, Tank group Graph, Valve Graph, Valve Group Graph.

Pipe Group Graph

Displays simulation results for two or more pipes. The graph X-axis displays time in units defined with the Simulation Time command (Report Timestep) and the Y-axis displays simulation results for the selected pipes. Upon choosing this graph type you are prompted to choose two or more pipes. Upon selecting pipes, the graph appears in the Output Report Manager window. Pipe Group graphs can display pipe flow, velocity, headloss, headloss per 1000 feet (m), or water quality analysis variable.

Press the Enter key or the right mouse button to terminate the pipe selection process.



Note 1: Negative pipe flows represent flow moving from the pipe's from-node to its to-node, opposite the pipe orientation.

Note 2 - To view results for one pipe, use the Pipe Graph.

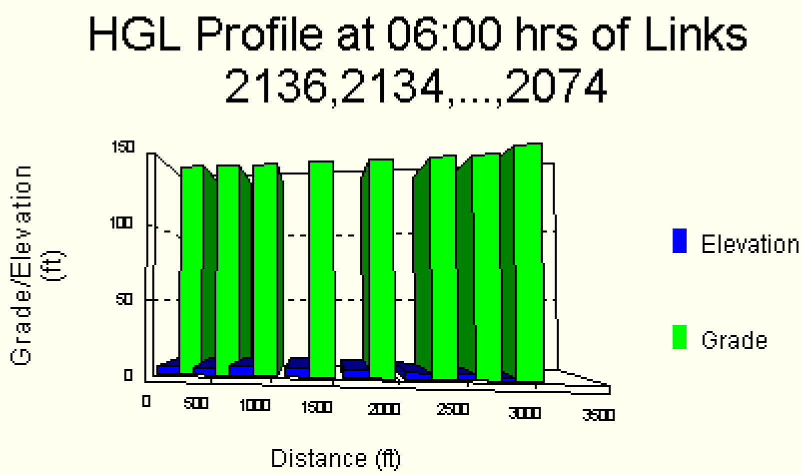
Other Related Topics - Available Graph Types, Average Reaction Rate, Energy Curve, Frequency, HGL Graph, Hydrant Curve, Junction Graph, Junction Group Graph, Pipe Graph, Pump Curve, Pump Graph, Pump Group Graph, Reservoir Graph, Reservoir

Group Graph, System Curve, System Demand, Tank Graph, Tank group Graph, Valve Graph, Valve Group Graph.

HGL Profile

Displays the hydraulic grade line profile along a series of connected pipes. Nodal locations and ground elevations are displayed in conjunction with HGL for reference. The graph X-axis displays length measured from the beginning of the first selected pipe and the Y-axis displays elevation and hydraulic grade.

Press the Enter key or the right mouse button to terminate the pipe selection process. You are notified if you attempt to re-select a previously selected pipe a pipe that is not directly connected to the previously selected pipe via a common node.



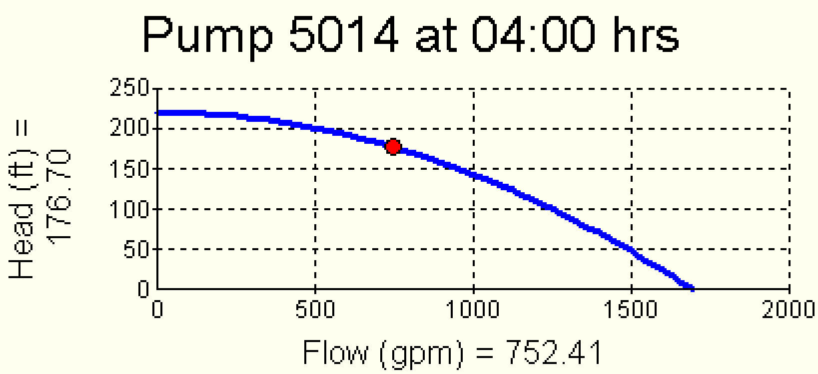
Note: It is important that the pipes selected for the HGL Profile graph be in series.

Other Related Topics - Available Graph Types, Average Reaction Rate, Energy Curve, Frequency, Hydrant Curve, Junction Graph, Junction Group Graph, Pipe Graph, Pipe Group Graph, Pump Curve, Pump Graph, Pump Group Graph, Reservoir Graph, Reservoir

Group Graph, System Curve, System Demand, Tank Graph, Tank group Graph, Valve Graph, Valve Group Graph.

Pump Curve

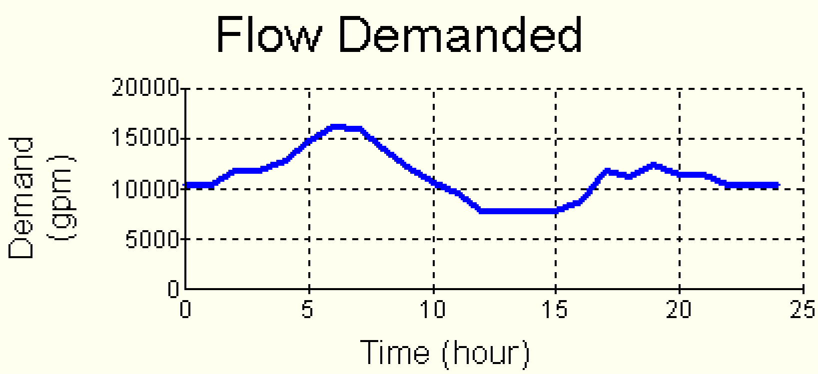
Displays a pump's characteristic curve as defined by the Element Attribute section of the Modeling Browser and operating point as calculated by InfoWater. The graph X-axis displays flow in flow units as defined with the Simulation Times command and the Y-axis displays head in output head unit. Pump curve graphs can display the pump head-flow operating point at any extended period simulation timestep you choose. If the pump is off at the current time period, the words "PUMP OFF" are displayed on the graph. If the pump is on, the pump operating point on the curve is displayed as a red dot annotated with the operating head and flow.



Other Related Topics - Available Graph Types, Average Reaction Rate, Energy Curve, Frequency, HGL Graph, Hydrant Curve, Junction Graph, Junction Group Graph, Pipe Graph, Pipe Group Graph, Pump Graph, Pump Group Graph, Reservoir Graph, Reservoir Group Graph, System Curve, System Demand, Tank Graph, Tank group Graph, Valve Graph, Valve Group Graph.

System Demand

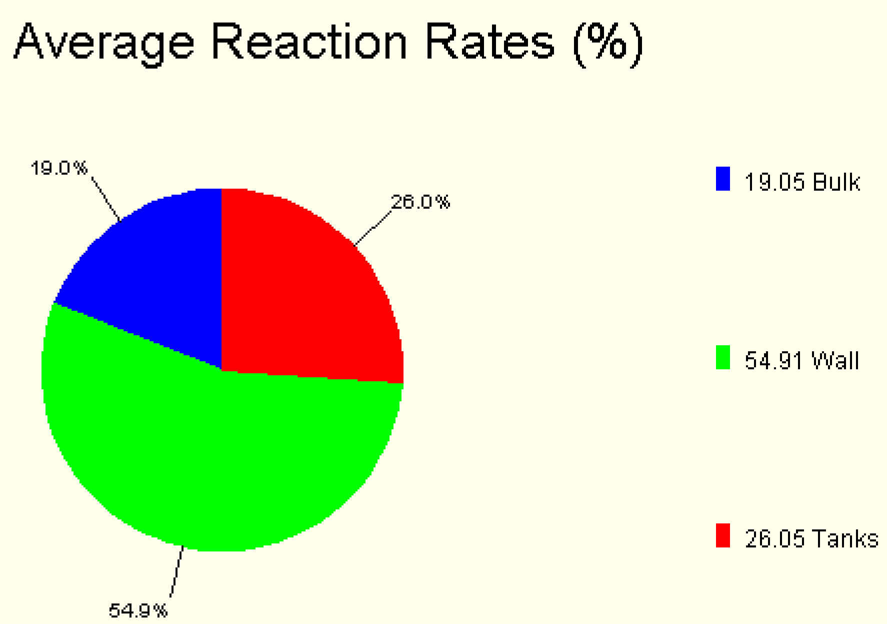
Displays total or net system demand during an extended period simulation. The graph X-axis displays time in units as defined with the Simulation Times command and by default the Y-axis displays total flow supplied from all storage nodes in demand units. The Y-axis can alternately display total system demand and total flow stored in demand units.



Other Related Topics - Available Graph Types, Average Reaction Rate, Energy Curve, Frequency, HGL Graph, Hydrant Curve, Junction Graph, Junction Group Graph, Pipe Graph, Pipe Group Graph, Pump Curve, Pump Graph, Pump Group Graph, Reservoir Graph, Reservoir Group Graph, System Curve, Tank Graph, Tank group Graph, Valve Graph, Valve Group Graph.

Average Reaction Rate

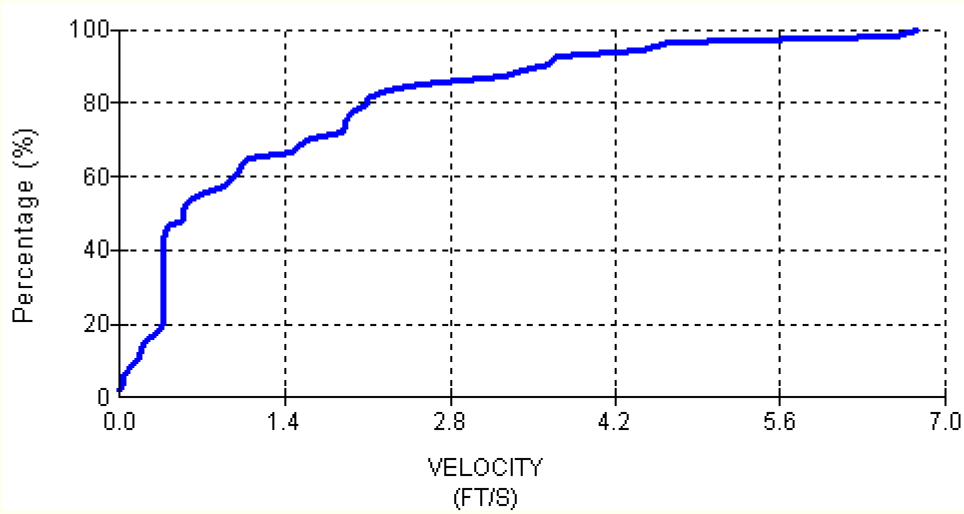
The average reaction rate graph displays the amount of chemical reactions occurring within the hydraulic system. Broken down as a pie chart, the average reactions within each portion of the system are shown as a percentage of all reaction within the system.



Other Related Topics - Available Graph Types, Energy Curve, Frequency, HGL Graph, Hydrant Curve, Junction Graph, Junction Group Graph, Pipe Graph, Pipe Group Graph, Pump Curve, Pump Graph, Pump Group Graph, Reservoir Graph, Reservoir Group Graph, System Curve, System Demand, Tank Graph, Tank group Graph, Valve Graph, Valve Group Graph.

Frequency Graph

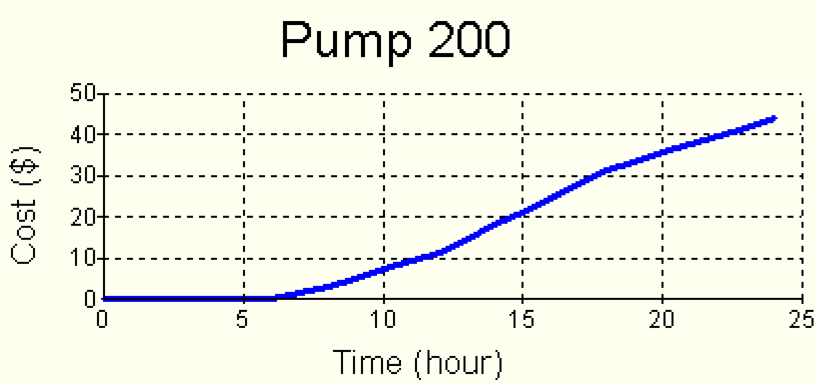
A frequency graph can only be created by viewing an output report. Once an output report is open (pipe report, junction report, etc), highlight the attribute (column) for which a frequency curve is desired and select the frequency icon at the top of the dialog box. A frequency curve will be generated as shown below.



Other Related Topics - Available Graph Types, Average Reaction Rate, Energy Curve, HGL Graph, Hydrant Curve, Junction Graph, Junction Group Graph, Pipe Graph, Pipe Group Graph, Pump Curve, Pump Graph, Pump Group Graph, Reservoir Graph, Reservoir Group Graph, System Curve, System Demand, Tank Graph, Tank group Graph, Valve Graph, Valve Group Graph.

Energy Curve

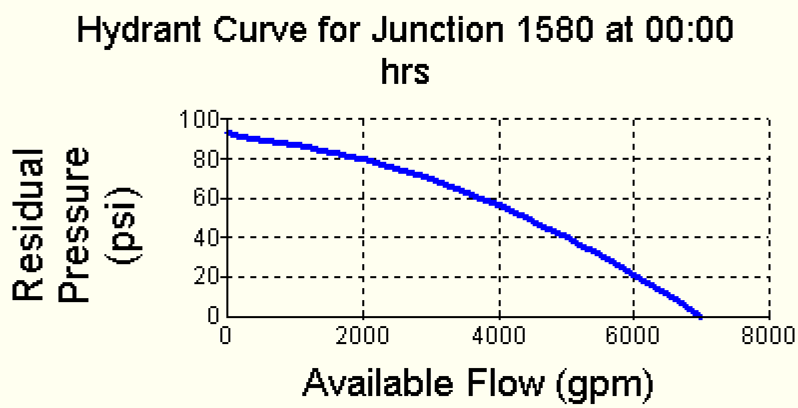
Displaying the results of an Energy Management simulation for one selected pump. The X-axis displays time in units defined with the Simulation Times command and the Y-axis displays energy management simulation results for the selected pump. Upon choosing this graph type you are prompted to choose a pump. By default the graph displays flow rate through the selected pump. Other energy management simulation variables may be selected from the drop-down list at the top right-hand corner of the graph window. You can display pump flow, head, useful power, efficiency, required power, and total cumulative cost on Energy Curve graphs.



Other Related Topics - Available Graph Types, Average Reaction Rate, Frequency, HGL Graph, Hydrant Curve, Junction Graph, Junction Group Graph, Pipe Graph, Pipe Group Graph, Pump Curve, Pump Graph, Pump Group Graph, Reservoir Graph, Reservoir Group Graph, System Curve, System Demand, Tank Graph, Tank group Graph, Valve Graph, Valve Group Graph.

Hydrant Curve

Displays the relationship between flow and pressure for the most recently generated Hydrant Curve simulation via the Run Manager. The current hydrant curve is entitled *active*.Hydrant Curve regardless of the identifier of the node for which the curve was generated (the correct node identifier is displayed in the title of the hydrant curve - see graph below). The graph X-axis displays flow and the Y-axis displays pressure. The X-axis range is from no flow to the maximum flow at zero pressure.

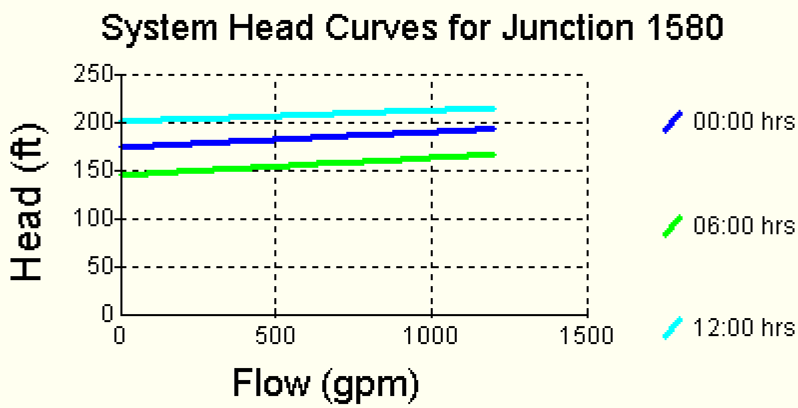


Note: This feature useful for evaluating the capacity of a hydrant to deliver fire flows. Click here for more information on Hydrant Curves and their relationship to Fire-Flows.

Other Related Topics - Available Graph Types, Average Reaction Rate, Energy Curve, Frequency, HGL Graph, Junction Graph, Junction Group Graph, Pipe Graph, Pipe Group Graph, Pump Curve, Pump Graph, Pump Group Graph, Reservoir Graph, Reservoir Group Graph, System Curve, System Demand, Tank Graph, Tank group Graph, Valve Graph, Valve Group Graph.

System Curve

Displaying the relationship between system head and flow for the most recently generated system curve, generated with the Run Manager. The most recently generated system curve is entitled "* ACTIVE *.System Curve" regardless of the identifier of the node for which the curve was generated (the correct node identifier is displayed in the node graph). The graph X-axis displays flow and the Y-axis displays head.



Note: This feature is useful for evaluating the variation in total dynamic head against which pumps will be required to operate under various flow conditions.

Other Related Topics - Available Graph Types, Average Reaction Rate, Energy Curve, Frequency, HGL Graph, Hydrant Curve, Junction Graph, Junction Group Graph, Pipe Graph, Pipe Group Graph, Pump Curve, Pump Graph, Pump Group Graph, Reservoir Graph, Reservoir Group Graph, System Demand, Tank Graph, Tank group Graph, Valve Graph, Valve Group Graph.

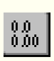
Report Icons

The following functions are available for modifying reports in the Output Report Manager and Custom Report Manager display windows. If you do not see one or more of the following buttons while a report is being displayed, that function is therefore not available for the currently displayed report type.

 **Refresh** - Refreshes the current report with data from the latest simulation run.

 **Print Grid** - Prints the selected records of the current report or all records if none are selected. You may format the report by indicating a report header, footer, margins, orientation, etc. Click here to learn more about the print grid.

 **Format Report** - Choose the columns to be displayed in the report and the order in which those columns appear from left to right. By default all columns are shown when a report is opened. Row number and component ID are always displayed. Additionally, you may specify column width and column/row *header scale*. The value of header scale indicates the number of lines of text that will be used to display column and row header information.

 **Format Column** - Align data display in the selected column as left justified, centered, or right justified. Specify inclusion of a comma for large numbers (1,234,567.00) or turn off comma display (1234567.00), and enter the display precision of the simulation result data.

 **Set Font** - Sets the font used to display information in the Output Report Manager. Choose from among the fonts currently loaded on your workstation.

 **Copy** - Copies the currently highlighted portion of the report (user-specified) to the Windows clipboard. Once the copy icon is

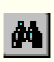
clicked, the user can then open any Windows package to paste the data. Copy is particularly useful for quick viewing, sorting and editing of data in a third party package like Microsoft Excel.

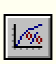
 **Sort Ascending** - Sorts the table rows in ascending order of values in the current column. Click once on the column heading with the mouse to choose a column. Sorts are for visual display only; physical data records are not sorted.

 **Sort Descending** -Sorts the table rows in ascending order of values in the current column. Click once on the column heading with the mouse to choose a column. Sorts are for visual display only; physical data records are not sorted.

 **Filter** - On the Output Report Manager, enter a query statement to be issued against simulation result data. Only those records meeting your criteria will be displayed in the current report. This function can be used to display critical records such as those junctions whose pressure falls below 20 psi, tanks whose grade or percent full falls below a specified value, etc. Records meeting the selected criteria are highlighted with a **red check** on the report panel.

On the Custom Report Manager, displays the Define Data Scope panel (step 3 on the Custom Report Wizard) allowing you to use one of the available options (All, Domain, DB Query Statement) to restrict the records that appear on the report.

 **Find** - The find icon will bring up a dialog box in which the user can type in text to search the output report. InfoWater will then find every occurrence of this matching text.

 **Frequency** - Creates a frequency graph for a selected attribute (column). For example, to create a frequency graph for pipe velocities in a system, open a pipe report, highlight the velocity column and then click the frequency icon. A frequency icon will now be displayed. Click here to learn more.

 **Save as Domain** - Create a domain from the report records selected via a Filter operation.

 **Compare Report** - Displays a dialog box allowing you pick an output source storing simulation results of a type identical to that shown on the current report. The data associated with the selected output source can be displayed in conjunction with the data currently shown on the report. This option is available to those users who have developed custom scenarios and who wish to show common fields from two different scenarios for comparative purposes (ex. flow in pipe 101 for Scenario 1 and Scenario 2). Click here for more information on developing comparison reports.

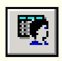
Report Time - The current timestep for which the report displays results. Using this drop down box, the user can switch to another timestep and the results will be automatically updated.

Customized Report Methodology

The customized Report provides you with a means to customize your report and to create a combination of input and output data.

Methodology

The customized report feature of InfoWater is a three step wizard process whereby you can select which input and output data (ex. pipe flow, node pressures, etc.) are desired to be viewed in a customized report.

- To create a customized report, from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Customized Report**. Alternatively you can use the **Customized Report Manager** icon  on your **InfoWater Edit Network** tool bar to launch this command. From here the user is taken through a wizard process to create a customized output report.
- In the Define Data Source dialog box, identify which InfoWater element type, input data and output data are desired and then click on **Next** to launch the **Define Display Fields** dialog box.
- In the Define Display Fields dialog box, define which fields will be included in the report and the order in which they will appear on the report. Field prefixes (ex. JUNCTION:) reflect the component types. Click on **Next** to launch the **Define Display Scope** dialog box.
- Choose the display scope in the Define Display Scope dialog box and click on **Finish** to launch the Customized Report.

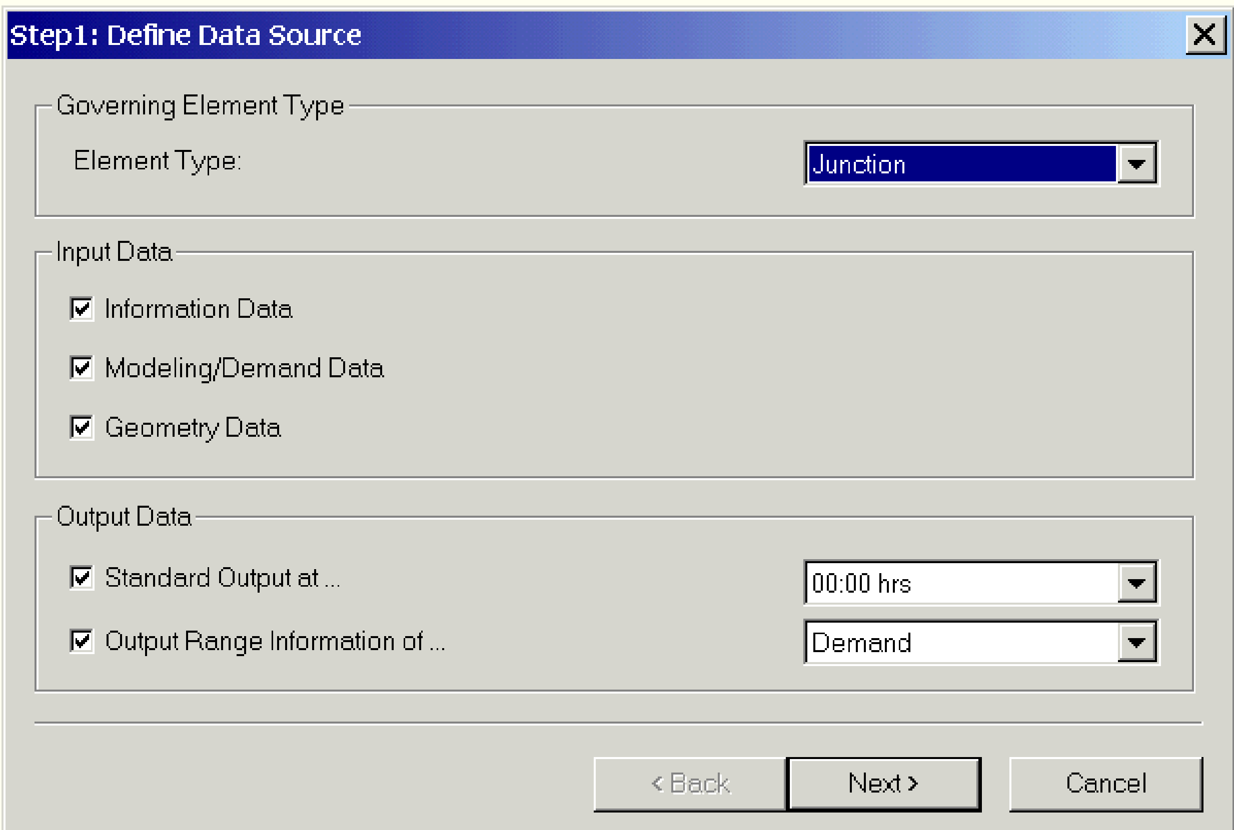
Other Related Topics - Customized Report, Define Data Source, Define Display Fields, Define Display Scope

---

Define Data Source

Use the Define Data Source dialog box to define the element type, the input Data type and the Output data that you want to see in your customized report. Click here for information on the Customized Report creation methodology.

Click on any portion for more information:

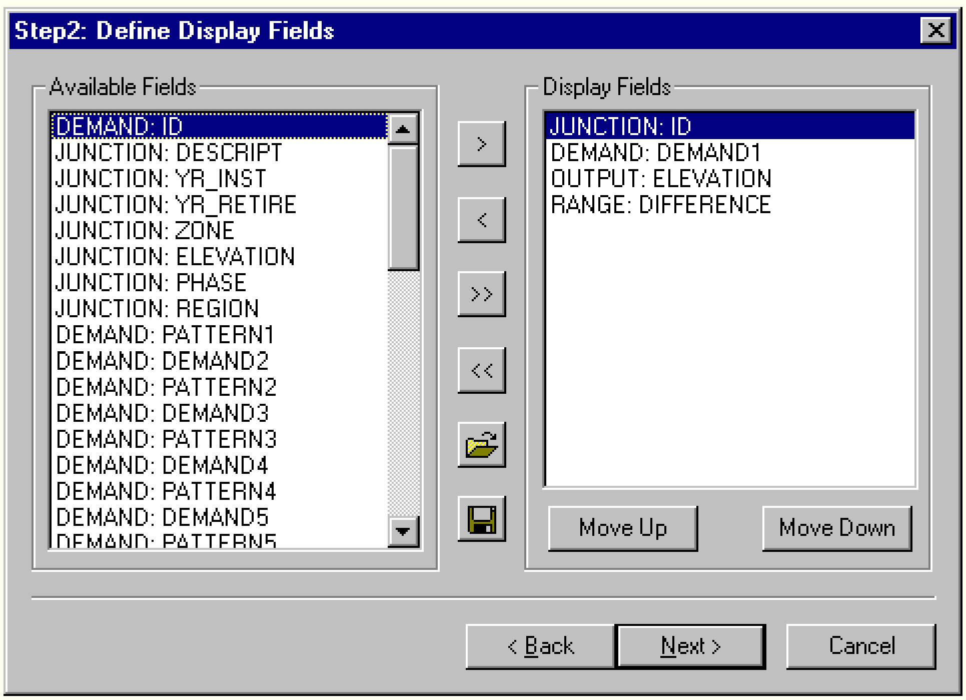


Other Related Topics - Customized Report, Customized Report Methodology, Define Display Fields, Define Display Scope

Define Display Fields

The second step is to define which fields will be included in the report and the order in which they will appear on the report. Field prefixes (ex. JUNCTION:) reflect the component types. The user also has the option of saving a custom configuration for later use. Click here for information on the Customized Report creation methodology.

Click on any portion of the dialog box below to learn more.

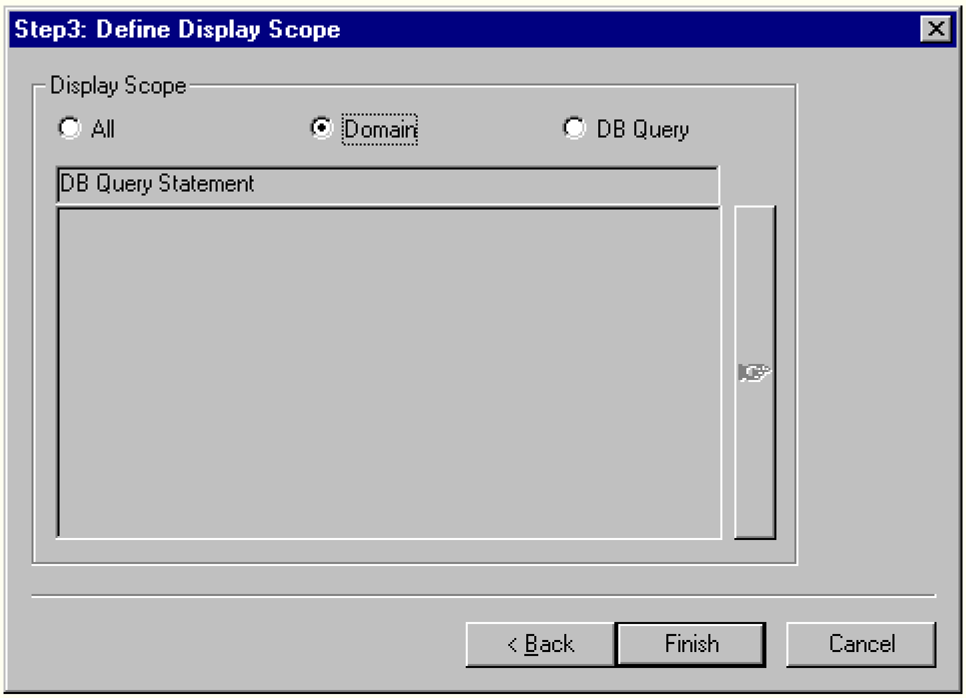


Other Related Topics - Customized Report, Customized Report Methodology, Define Data Source, Define Display Scope

Define Display Scope

The third step is to define which components will be displayed on the custom report. You may show records for all components of the current type, components in the current network domain, or you may define a query statement to restrict the records to be shown on the custom report. Choose the Finish button to display the custom report. Click here for information on the Customized Report creation methodology.

Click on any portion of the dialog box below to learn more.



Other Related Topics - Customized Report, Customized Report Methodology, Define Data Source, Define Display Fields

Output Relate Methodology

The Output Relate option is used to establish relationships between modeling input data and modeling results. Once a relate (link) is established, you can then apply queries against the current project based on modeling inputs and simulation results simultaneously.

Uses of Output Relates in InfoWater

- Output Relates provide a method of temporarily transferring simulation result data to InfoWater's model input database to support database operations (such as queries) using the output data. Click here for information on querying Output Relates.
- Output relates are stored in an external database file that can be linked to in a third party GIS software to view analysis results.

Methodology

To build an Output Relate, do the following:

- Launch the InfoWater **Table of Contents** dialog box from the ArcMap **View** menu.
- From the InfoWater **Table of Contents -> Operation** tab click on the **Output Relate** folder.
- Right mouse click on the Output Relate folder and select the **New** command. Then specify an ID and an optional description (ID no more than 20 characters, no spaces and Description no more than 60 character, may contain spaces) and click on the **OK** button.
- Choose the target scenario from the drop down box (choose either the Base scenario or any other custom scenario).

- Choose the target simulation from the drop down box (choose from <Standard, Fireflow, SCADA>).
- Choose the desired output report from the drop down box (All reports pertaining to the simulation type will be available for update).
- Choose the desired time step for the update.
- Click on **OK** to create the Relate.
- Also if you choose to update your Relate automatically, check the Automatic Update Check box.

Now you may use your output relate to import the outputs into your GIS or create InfoWater queries to query based on the output results.

Other Related Topics - Output Relate Dialog Box**,** Query Output Relate**,** Query Output relate Methodology**.**

Query Report Methodology

Like a customized report, a query report is used to selectively choose which input and output data will appear in a report. However, unlike a customized report, a query report is required to relate data from an Output Relate with output data in a report format. In other words, the only way to see output data in a report format is to create a query report that contains a desired output relate.

Methodology

Do the following to create a Query Report -

- Run a Hydraulic Simulation - Using the Run Manager, run either a steady-state or EPS simulation.
- Generate Query Statement using an Output Relate - Open the **DB Query** Manager under the **Table of Contents -> Operation** tab and create a new query, querying your Output Relate. For instance an Output Relate for Junctions and a DB Query identifying junctions with pressure < 35 psi may form a good combination. Click here for more information on Querying Output Relates.
- From the **InfoWater Control Center -> InfoWater** button -> **Tools** menu, select the **Query Report Manager** command. This launches the Query Report Dialog box.
- From the **Source Query ID**, select the appropriate database query.
- The **Available Fields** box will show all of the fields available from the output relate while the **Display Fields** box shows which fields will be included in the query report.
- Use the Display buttons to add and remove fields from inclusion in the output report.
- When all desired fields have been added to the **Display Fields** box, click **OK** and the Query Report will be generated.
- Review and print report as desired.

Other Related Topics - Query Report, Query Report Dialog Box

Query Report

Once the selection process is completed, based on the options chosen the Query Report is displayed as below. Click here for more information on the Query Report Creation process.

	JUNCTION: ID (Char)	DEMAND: DEMAND1 (Real)
1	9	14.00
2	89	21.00
3	87	0.00
4	85	14.00
5	83	14.00
6	81	14.00

Other Related Topics - Query Report Dialog Box, Query Report Methodology

Query Summation Report Methodology

The Query Summation Report may be used in conjunction with Data base queries to determine and evaluate the different attributes associated with the InfoWater data elements (Junctions, Tanks, Reservoirs, Pumps, Valves and Pipes) that satisfy the **DB query**.

Methodology

Do the following to launch a Query Summation Report -

- Create a DB Query using the DB Query dialog box. Click here to learn more about the DB Query methodology.
- Launch the Query Summation Report dialog box from your **InfoWater Control Center -> InfoWater** button **-> Tools** menu **-> Query Summation Report**.
- Specify the **DB Query** in the **ID** field of the **DB Query** section of the **Query Summation** dialog box. A report or graph depending on which tab (Report, Bar or Pie) you choose will be displayed in the display section of the **Query Summation** dialog box for all the elements that satisfy your DB Query.
- Choose the **Summary Field** and the **Summary Class Field** in your Query Summation dialog box.
- Click on the **Refresh** button to view your Query Summation Report. You may change your Query ID, Summary Field and Summary Class Field to view other attributes.
- Click on the Bar tab to view a Bar chart and choose the Pie chart option to view a Pie chart. You may use the Graph Modification icons to customize your graphs.

Other Related Topics - Query Summation Report**,** Query Summation Report Bar Chart**,** Query Summation report Pie Chart

---

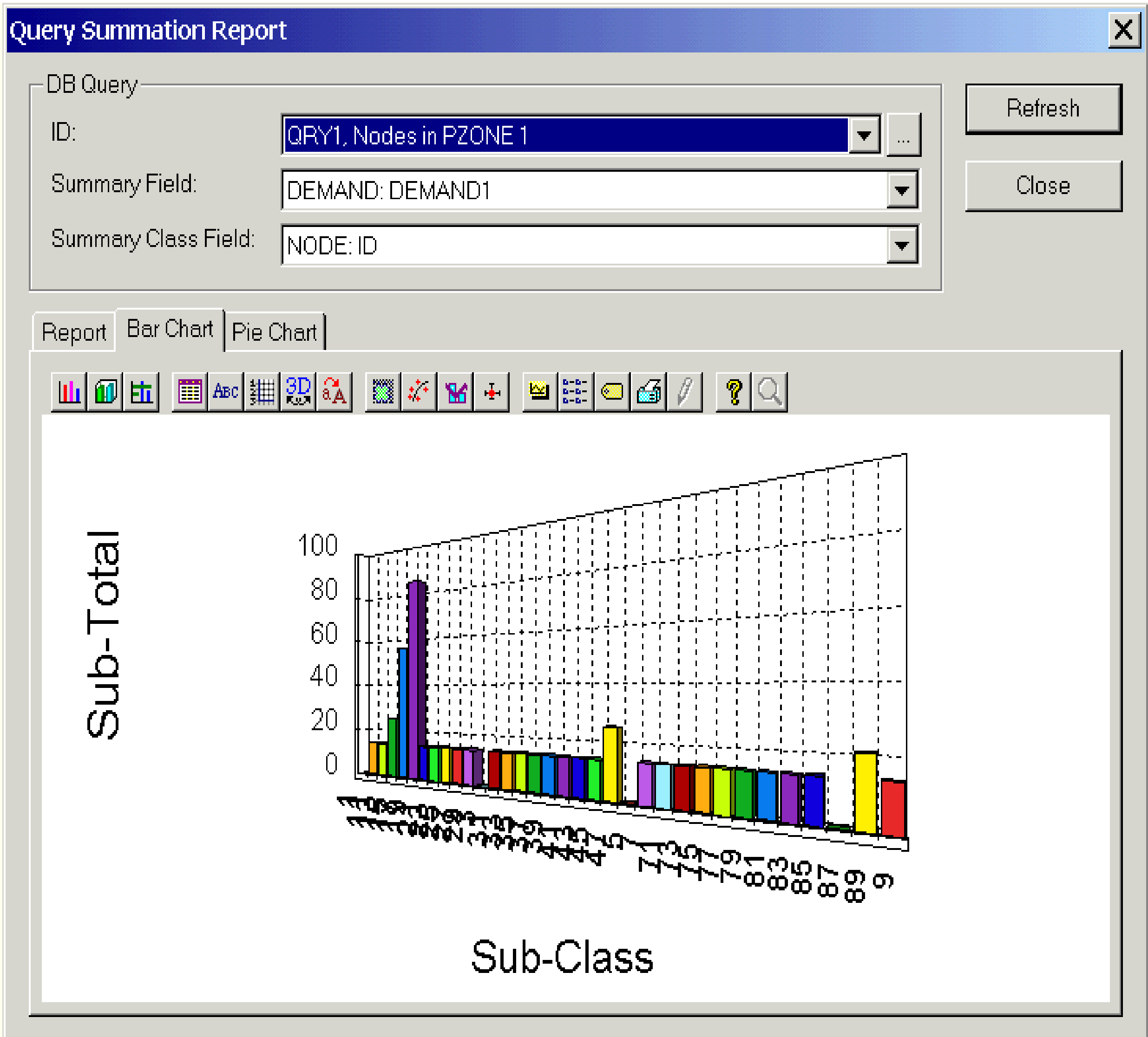
Bar Chart

The Query Summation Report may be used in conjunction with DB Queries to determine and evaluate the different attributes associated with the InfoWater data elements (Junctions, Tanks, Reservoirs, Pumps, Valves and Pipes) that satisfy the DB query.

For instance in the following Bar Chart you may evaluate all the attributes of junctions in pressure zone 1 in the form of a bar chart here.

You may choose to customize this graph by using any of the different customization icons available at the top of the bar chart. To learn more about the Query Summation Report methodology click here.

Click on any portion to learn more.



Other Related Topics - Query Summation Report Bar Chart, Query Summation Report Methodology, Query Summation report Pie Chart

---

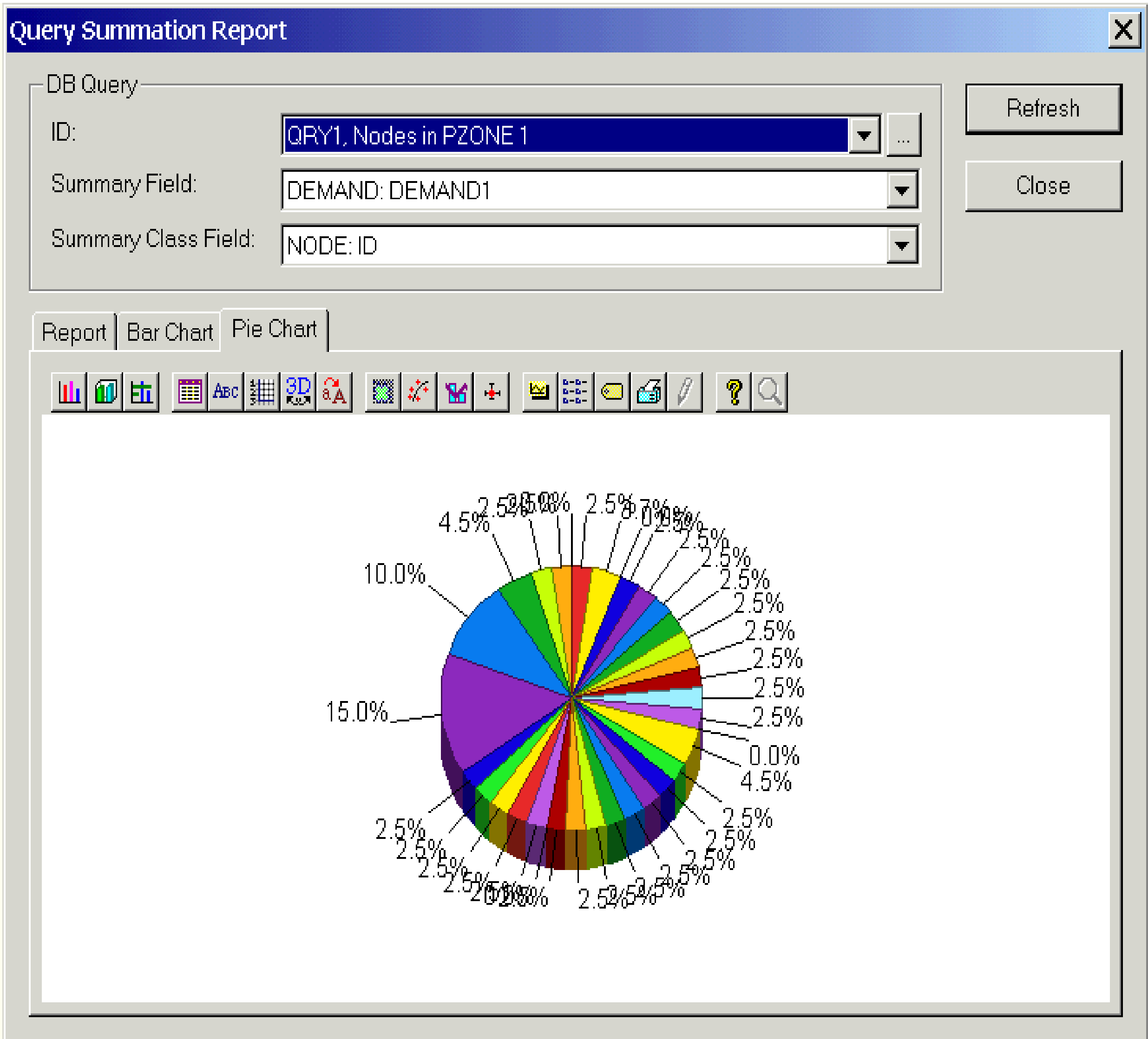
Pie Chart

The Query Summation Report may be used in conjunction with DB Queries to determine and evaluate the different attributes associated with the InfoWater data elements (Junctions, Tanks, Reservoirs, Pumps, Valves and Pipes) that satisfy the DB query.

For instance in the following Pie Chart you may evaluate all the attributes of junctions in pressure zone 1 in the form of a pie chart here.

You may choose to customize this graph by using any of the different customization icons available at the top of the pie chart. To learn more about the Query Summation Report methodology click here.

Click on any portion to learn more.



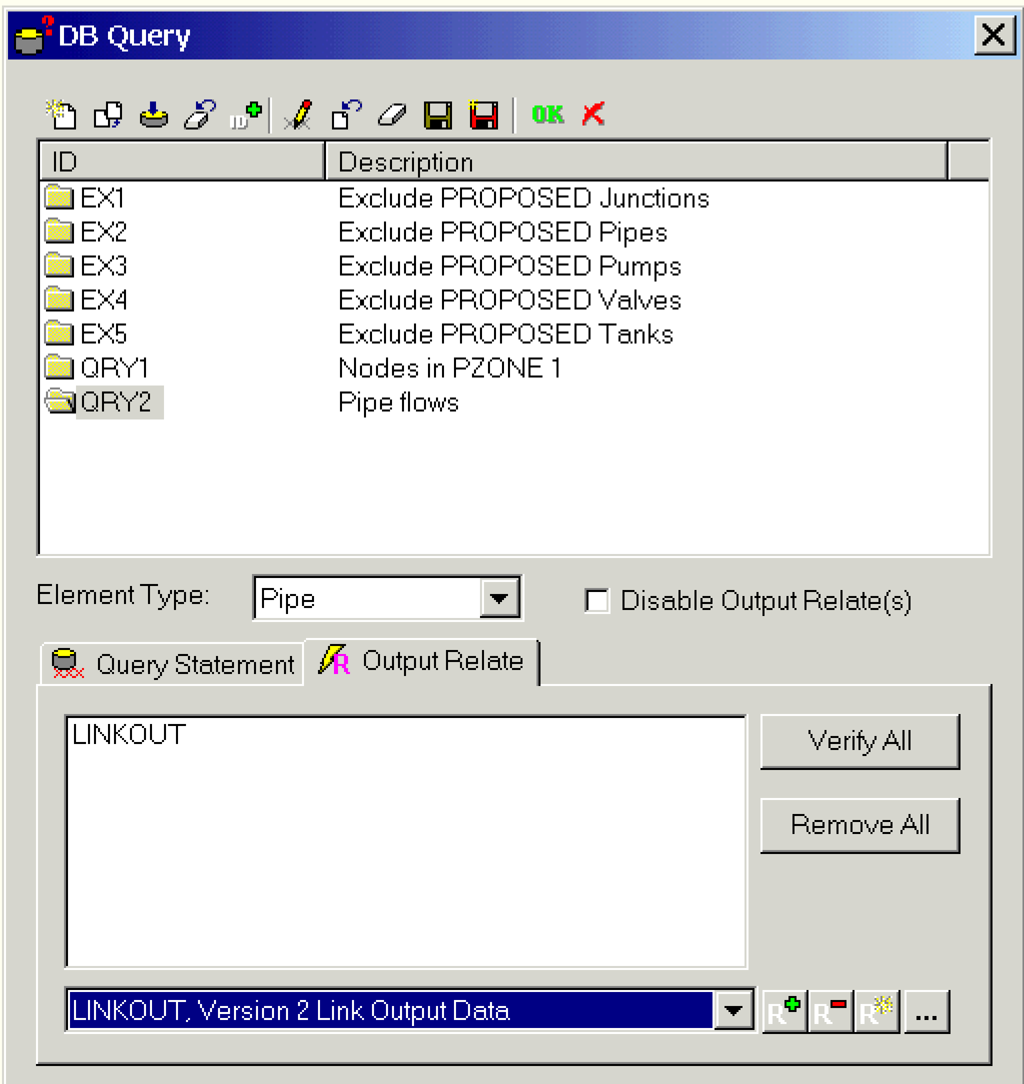
Other Related Topics - Query Summation Report, Query Summation Report Bar Chart, Query Summation Report Methodology

---

Output Relate Db Query Tab

The Output Relate option is used to establish relationships between modeling input data and modeling results. Once a relate (link) is established, you can then apply queries against the current project based on modeling inputs and simulation results simultaneously. For more information on creating Output Relates click here. For more information on using Output Relates in conjunction with DB Queries click here.

Click on any section for more information.



Other Related Topics - Output Relate Dialog Box, Output Relate Methodology, Query Output relate Methodology.

Query Output Relate Methodology

The Output Relate option is used to establish relationships between modeling input data and modeling results. Once a relate (link) is established, you can then apply queries against the current project based on modeling inputs and simulation results simultaneously.

Output Relates provide a method of temporarily transferring simulation result data to InfoWater's model input database to support database operations (such as queries) using the output data. When you execute an Output Relate, simulation results stored in a standard (hydraulic and water quality) energy, fireflow, or SCADA output source you select are written to the desired output report table (e.g., Junction Report, Pipe Report, etc.). The table contains simulation results for all network components at a single extended period simulation timestep.

Each output relate is identified by a name and narrative description and may be accessed by that name when developing query statements using several InfoWater commands and tools.

The data transferred as a result of the relate is a "snapshot" of simulation results and will therefore be static regardless of subsequent simulation runs. To update the contents of an output relate with results from a more recent simulation run, use either the Update Current Relate Now or Update All buttons. The Update operation replaces the original related output data with the new output data. You must specify the Target Simulation (output source such as standard, energy, fireflow, SCADA) and the Desired Output Report (Junction Report, Pipe Report, etc.) which the output data will be mapped to. The related output will remain intact for later use (i.e., after the project is closed). Both the Query Set command and the Query Builder dialog box can access output relate data for use in developing query sets and query statements.

Results in an output relate can be made static, as a permanent snapshot of a single model run, or can be automatically updated every time you rerun a simulation for a given scenario. To make updates automatic, from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Project Preferences**. Under the **Operation Settings** tab, select **Auto Output Relate Update**.

Output relates are stored in an external database file that can be linked to in a third party GIS software to view analysis results.

Methodology

To build a query statement using an Output Relate, do the following:

- From the InfoWater **Table of Contents -> Operation** tab click on the **DB Query** folder.
- Right mouse click and select the **New** command and specify an ID and an optional description (ID no more than 20 characters, no spaces and Description no more than 60 character, may contain spaces) and click on **OK**.
- Choose the **Element Type** that you want to create a query for (Choose from Junctions, Pumps, Tanks, Reservoirs, Valves and Pipes).
- Click on the **Output Relate** tab in the **DB Query** dialog box. Choose the Output relate from the Query list select box and click on the **Add Relate** icon . For more information on creating Output Relates, click here.
- Click on the **Query Statement** tab of the Query Builder dialog box.

- Click on the **Query Builder** icon  to launch the Query Builder dialog box.
- The **Query Builder** dialog box will now contain a fourth tab which contains the output data relating to the Output Relate.
- Click on the column for the data type you want to query on.
- Choose the operator from the functions and operator section.
- Enter the value that you want as your second operand in the Value Box.
- Click on the **Add** button to add the query in the Statement section of the Query Builder.
- Click on **Validate** to validate your query.
- Click on **OK** to save your query and exit from the query builder dialog box.

Other Related Topics - Output Relate Dialog Box**,** Output Relate Methodology**,** Query Output Relate**.**

Group Editing Methodology

The InfoWater Group Editing feature allows the user to assign values and settings to a group of elements "on the fly". This is extremely useful when conducting Water Quality analysis, Fire-Flow Simulations and other operational studies.

Using the Group Editing feature in InfoWater

Group Editing provides a time-saving tool that can be used for:

- Assign emitter coefficient to a group of junctions (see Emitter).
- Specify pressure dependent demand values to a group of junctions (see Pressure Dependent Demand).
- Assigning Initial Quality Concentrations to a group of Nodes (see Initial Quality).
- Assigning Water Quality Source Concentrations to a group of Nodes (see Quality Source).
- Specifying pipe status for a group of Pipes (see Pipe Status).
- Assigning Pipe Reaction values to a group of Pipes (see Pipe Quality
- Specifying Fire-Flow values for a group of Hydrants (see Fire-Flows).
- Assigning Tank Reaction values to a group of tanks (see Tank Reaction).
- Assigning Tank Mixing characteristics to a group of tanks (see Tank Mixing).

Methodology

To use the Group Editing feature, do the following:

Group Editing on Domain

- Choose **Domain Manager** from the **InfoWater Control Center -> InfoWater** button **-> Tools** Menu.
- Use the Domain Manager to select the elements that you want to include in your Domain for the group Editing process. Click here for more information on the domain creation process.
- Close out of the Domain Manager dialog box and click on **Group Editing on Domain** under the **InfoWater Control Center -> InfoWater** button **-> Edit** menu to launch the Group Editing on Domain dialog box.
- Choose the appropriate tab depending on the type of group assignment you would like to perform and click on **Apply**.
- Once any of the elements selected for a group edit are populated with data, the information input by the user will also appear in various database tables under the DB Editor.

Group Editing on Selection

- Launch the Group Editing on Selection command from the **InfoWater Control Center -> InfoWater** button **-> Edit** menu.
- Select the elements that you want to include in your group editing process and then right click and choose **Enter**.
- Choose the appropriate tab depending on the type of group assignment you would like to perform and click on **Apply**.
- Once any of the elements selected for a group edit are populated with data, the information input by the user will also appear in various database tables under the DB Editor.

Other Related Topics - Edit Domain Attributes, Edit Selection Attributes, Emitter, Fire-Flow, Group Editing, Initial Quality, Pipe Quality, Pipe Status, Pressure Dependent Demand, Quality Source, Tank Mixing, Tank Reaction

Database Queries

A query of a database allows the user to extract a record (or records) of a data using logical statements on a field (or fields) stored within a specific database. With InfoWater, the user is able to select network components and related data by creating logical query statements through the user-friendly DB Query feature.

Using Queries in InfoWater

DB Queries are a time-saving tool that can be used for:

- Activating a portion of the network for simulation, typically as a facility set associated with a custom scenario (see Facility Manager).
- Creating a domain to highlight a subset of network components (see Domain Manager.)
- Restricting the database records that are available for editing using the Database Editor.
- Customizing the map display by color-coding network components (see Query Sets).
- Creating Selection Sets (see Selection Sets)
- Output Relates (stored output result data) can also be associated with DB Queries which are important when creating Query Reports.
- Restricting model results to identify network components whose modeled performance (pressures, flows, etc.) meet or do not meet user specified criteria (e.g., nodes where pressure drops below 20 psi) using the Output Report Manager.

You may develop your own query statements or use one of the pre-defined InfoWater special queries to select portions of your distribution system model.

Methodology

To build a query statement, do the following:

- From the InfoWater **Table of Contents -> Operation** tab click on **DB Query**.
- Right click and choose the **New** command, and specify an ID and description (ID no more than 20 characters, no spaces and Description no more than 60 character, may contain spaces) and click on **OK**.
- Choose the **Element Type** that you want to create a query for (Choose from Junctions, Pumps, Tanks, Reservoirs, Valves and Pipes) then Click on OK.
- Click on the Query Builder icon  to launch the Query Builder dialog box.
- Choose the appropriate data tab from among the three tabs at the top of the Query Builder dialog box.
- Click on the column for the data type you want to query on.
- Choose the operator from the functions and operator section.
- Enter the value that you want as your second operand in the **Value Box**.
- Click on **Add** to add the query in the Statement section of the Query Builder.
- Click on **Validate** to validate your query.
- Click on **OK** to save your query and exit from the query builder dialog box.

Other Related Topics - DB Query Dialog Box**,** Query Builder Dialog Box**,** Query Set Dialog Box**,** Query Sets Methodology**.**

Query Sets

A Query Set is a collection of individual database queries (made from the DB Query command) stored as a unique set of data. Query sets are used to group logical expressions for InfoWater network elements (pipes, junctions, etc) for network management and graphical presentation purposes.

Using Query Sets in InfoWater

Query Sets are a time-saving tool that can be used for:

- Activating a portion of the network for simulation, typically as a facility set associated with a custom scenario (see Facility Manager).
- Creating a domain to highlight a subset of network components (see Domain Manager.)
- Customizing the map display by color-coding network components.
- Creating Selection Sets (see Selection Sets)
- Output Relates (stored output result data) can also be associated with DB Queries which are important when creating Query Reports.

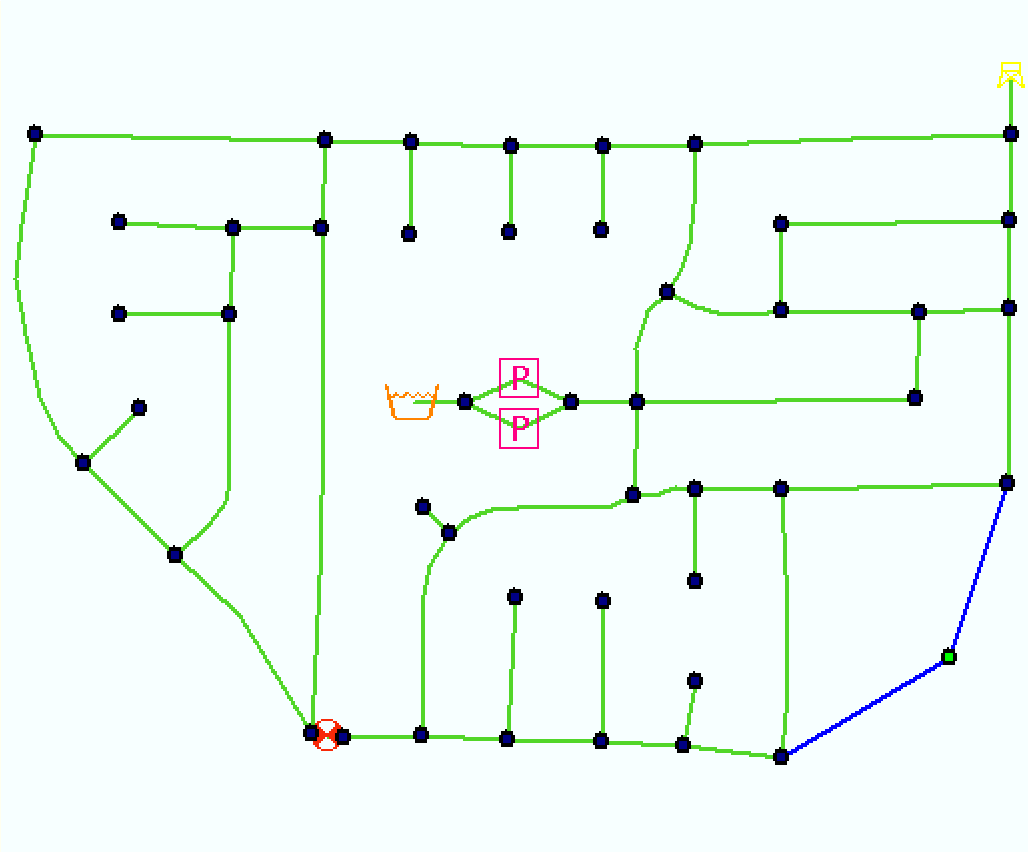
You may develop your own query statements or use one of the pre-defined InfoWater special queries to select portions of your distribution system model.

Methodology

To build a query set, do the following:

- From the InfoWater **Table of Contents -> Operation** tab click on Query Set.
- Right click and select the **New** command, and specify an ID and description (ID no more than 20 characters, no spaces and Description no more than 60 character, may contain spaces) and click on **OK**.
- From the **DB Query** select box under the **Association Details** section of the **Query Set** dialog box, select a DB Query. IF no DB query is available click on the Browse button  to launch the DB Query dialog box. Click here for more information on using the DB query dialog box.
- Repeat the earlier step to include more DB Queries.
- Assign color to each of the DB Queries by highlighting them in the **Association details & color** section and clicking on the **Color** button. This launches the **Color palette**. Select the color you want and click on **OK**.
- Finally click on the Green OK button  to save and close out of the Query Set dialog box.
- To apply your newly created query, left click on the Query ID (under the Query Set folder) and right mouse click and choose **Apply Query Set**. This should color your network facilities

The following graphic shows the how query sets may be used to color code your InfoWater network map.



Other Related Topics - DB Query Dialog Box, DB Query Methodology, Query Builder Dialog Box, Query Set Dialog Box.

Selection Sets

Selection sets are collections of data elements that can be saved and later recalled for activation.

Using Selection Sets in InfoWater

Selection sets may be used for the following purposes:

- Define domains – A domain can be created from one or more selection sets.
- Define facility sets – A facility set used for defining a scenario network subset can be defined from one or more selection sets.

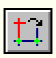
Selection sets are flexible in that the network components in a given-selection set may be used to add facilities to a domain or facility set or alternately to remove facilities from a domain or a facility. These options are specified on the Domain Manager and Facility Manager dialog boxes.

The network components included in a selection set can be defined from the following fashions:

- Graphical Selection
- Domain Selection

Methodology

To create a selection set, and . The user is prompted to enter an ID and a description. Once created, the Selection Set dialog box appears.

- From the **Operation** tab of the **Table of Contents**, highlight the Selection Set folder.
- Right mouse click and select the **New** command and specify an ID and an optional description (ID no more than 20 characters, no spaces and Description no more than 60 characters, may contain spaces) and click on OK.
- From the **Selection Set** Dialog box, now either populate a selection set from a previously created domain (**Load Domain** icon ) or you can select the elements you desire to populate this newly created selection set by using the **Load User Selection** icon .
- Click on **Yes** to confirm the selection set over write and in the case of the **Load User Selection** option specify the facilities you want to include in your selection set by left clicking on them and then right click to bring up the Selection set dialog box.

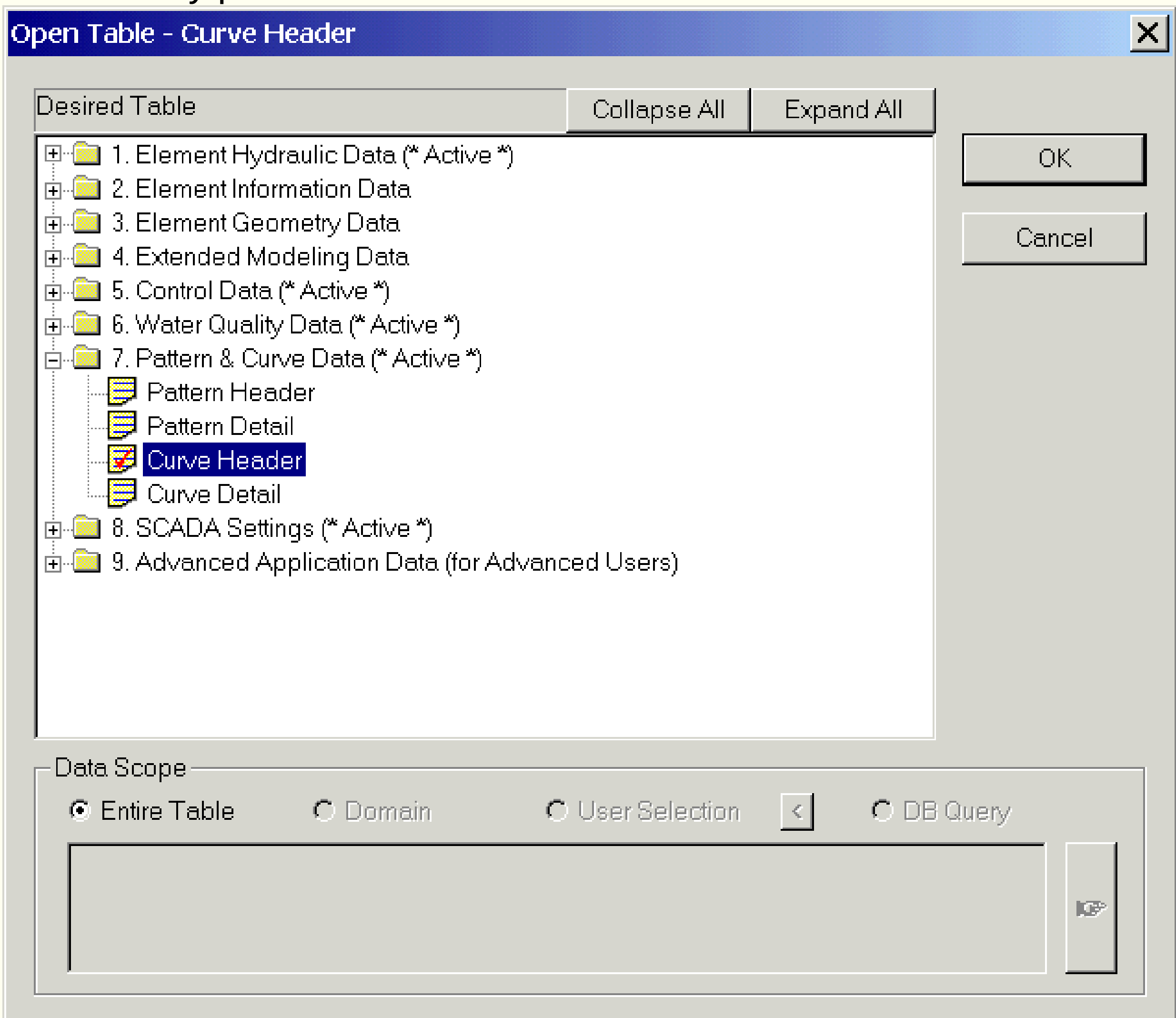
Other Related Topics - Selection Set Dialog Box

DB Editor

All the InfoWater project data including data related to the different InfoWater data elements such as pipes, junctions, tanks, reservoirs, pumps and valves are stored in the DB Tables. The Database Editor allows the user to open any InfoWater database and edit/modify or delete user input fields (except ID). To access the DB Tables, from the **InfoWater Control Center -> InfoWater** button **-> Edit** menu, select **DB Editor** to see the Open Table dialog box below.

From the **Open Table** dialog box, you can see that InfoWater stores database tables in nine separate folders. After selecting the desired table from under the relevant folder, you have the option of choosing a display scope found at the bottom of the dialog box to open the desired records of the database. Once this is done, click the OK button to open the database table.

Click on any portion for more details:



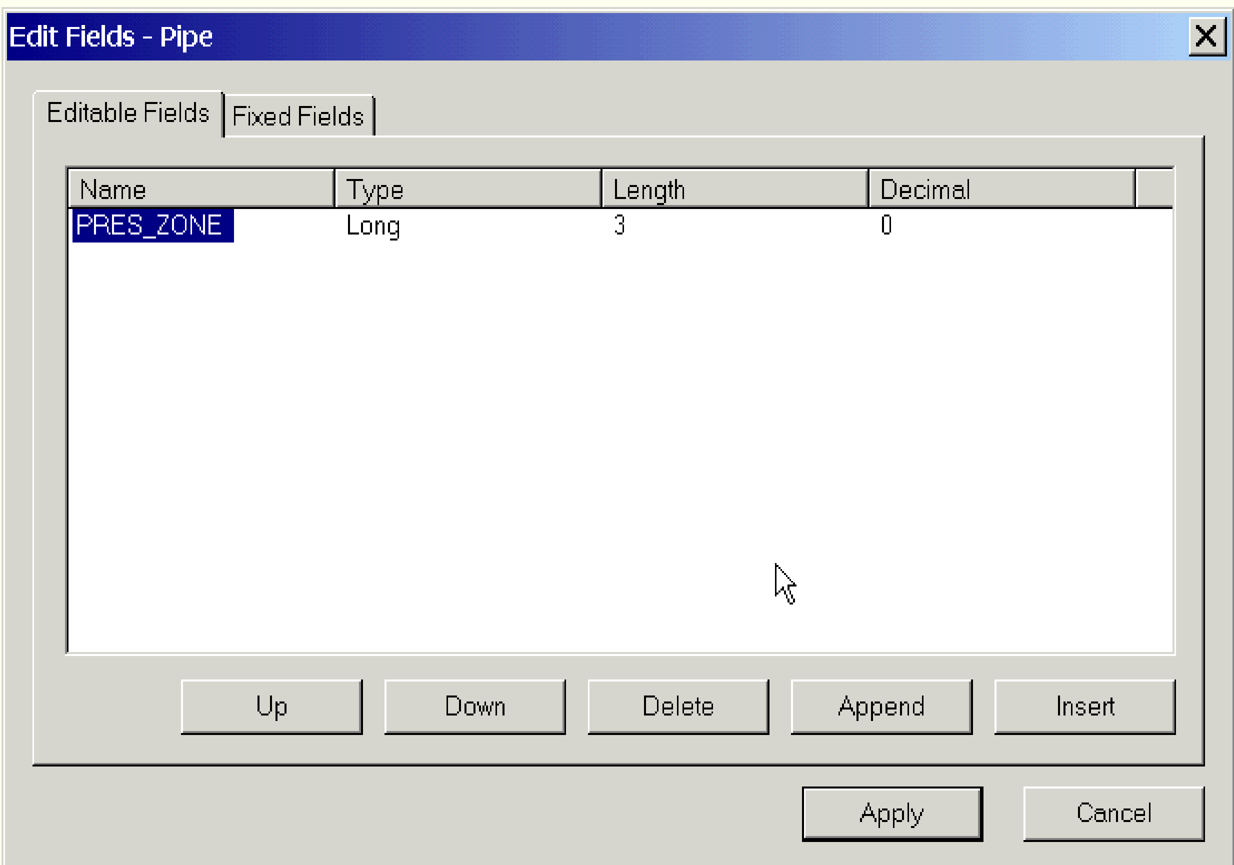
Other Related Topics - Advanced Application Data, Control Data Tables, Element Geometric Data Tables, Element Hydraulic Data Tables, Element Information Data Tables, Extended Modeling Data Tables, Pattern & Curve Data Tables, SCADA Settings Tables, Water Quality Data Tables, DB Editor.

Edit Fields

You can customize InfoWater by adding additional fields to your Information tables. Custom fields will be available for inclusion on output reports and for inclusion in queries, query sets and import/export process.

To add, reorder, or remove custom fields, perform the following:

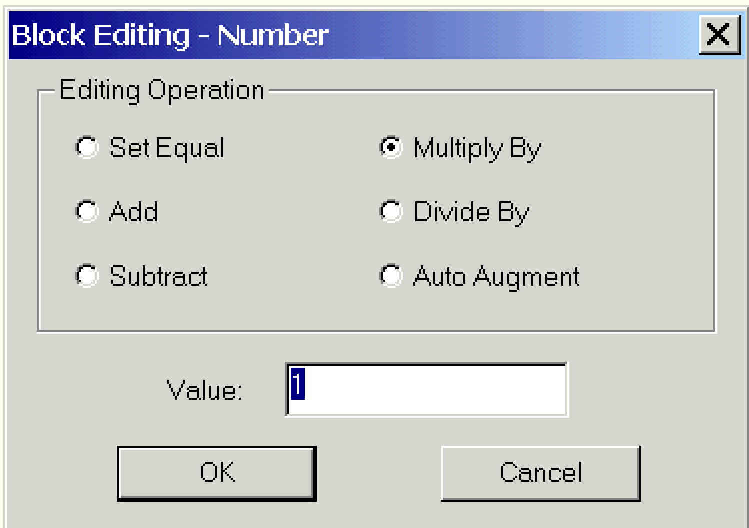
- Open the Database Editor and click the New button.
- From the dialog box, select the desired table from folder 2. Element Information Data.
- Click on the edit fields icon  to launch the Edit Fields dialog box as shown below:



- Choose from [Insert, Append], reorder [Up, Down], or remove [Delete] custom fields.
- The user must know if the desired field to be added will be character or numeric based. Once a field has been added to the selected database, it will be available for editing via the Element Browser (under the Information tab). You can also choose to Block Edit the new field with a pre-defined value.

Block Editing

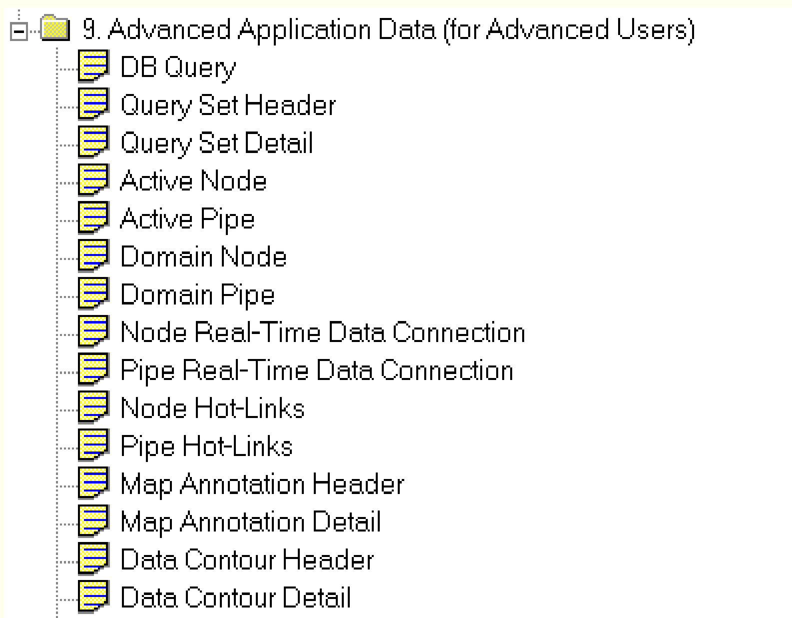
Use the block editing option to globally edit your data. Highlight the cells by left clicking and dragging your mouse. Once the cells that you want to edit are selected, click on the block editing option and choose from the 6 different available methods to modify your data.



Advanced Application Data

This table contains advanced application data and it is recommended not to alter or modify data through these tables.

Data that can be accessed through these tables include data related to DB Queries, Query Sets, Annotation & contour details, domain details etc.



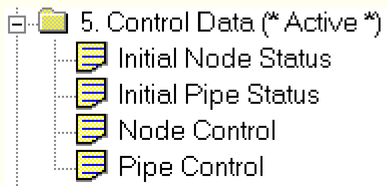
Note - Please do not use these Tables to alter or modify data.

Other Related Topics - Control Data Tables, Element Geometric Data Tables, Element Hydraulic Data Tables, Element Information Data Tables, Extended Modeling Data Tables, Pattern & Curve Data Tables, SCADA Settings Tables, Water Quality Data Tables, DB Editor, DB Tables.

Control Data

All the control information is included in these tables. Controls will alter the status of pipes, valves and pumps based on system criteria specified by you. For instance a Pump may be specified to turn on if the level of water in a Tank is below a prescribed level. Refer to the section on Initial Status and simple controls to learn mode about the different controls in InfoWater.

Click on any section for more information.

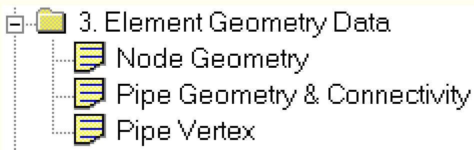


Other Related Topics - Advanced Application Data, Element Geometric Data Tables, Element Hydraulic Data Tables, Element Information Data Tables, Extended Modeling Data Tables, Pattern & Curve Data Tables, SCADA Settings Tables, Water Quality Data Tables, DB Editor, DB Tables.

Element Geometric Data

The element geometry tables contain all the geometry information for all the InfoWater data elements such as pipes, pumps, control valves, junctions, reservoirs and tanks.

Click on any section for more information.



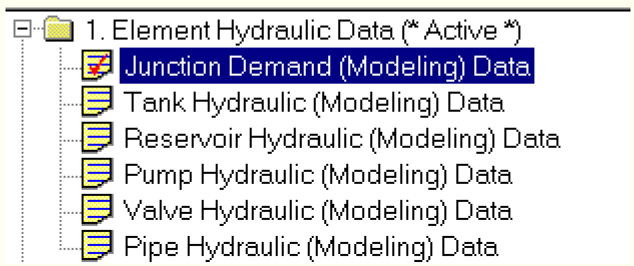
Note - Please do not use these Tables to alter or modify data.

Other Related Topics - Advanced Application Data**,** Control Data Tables**,** Element Hydraulic Data Tables**,** Element Information Data Tables**,** Extended Modeling Data Tables**,** Pattern & Curve Data Tables**,** SCADA Settings Tables**,** Water Quality Data Tables**,** DB Editor**,** DB Tables**.**

Element Hydraulic Data

These tables contain all the Modeling data for all the different data element types such as Junctions, Tanks, Reservoirs, Pumps, Control Valves and Pipes. These data are required by InfoWater to conduct the various simulations. However, not all fields need to be entered.

Click on any section below to learn more:

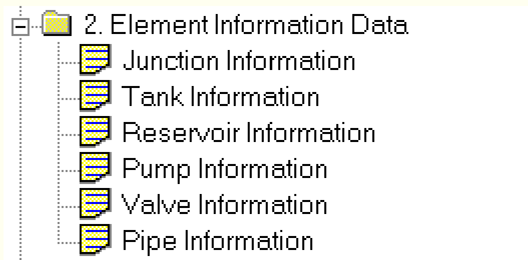


Other Related Topics - Advanced Application Data, Control Data Tables, Element Geometric Data Tables, Element Information Data Tables, Extended Modeling Data Tables, Pattern & Curve Data Tables, SCADA Settings Tables, Water Quality Data Tables, DB Editor, DB Tables.

Element Information Data

These tables contain all the Information data for all the different data element types such as Junctions, Tanks, Reservoirs, Pumps, valves and Pipes. These data may be queried against and also special custom columns can be added to these tables. This data will be available in the Output Reports and also may be exported from the InfoWater project. Data may also be imported into these columns from an external source.

Click on any section to learn more.

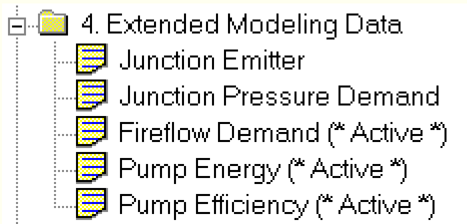


Note - Custom Columns may be added to these tables. Click here for more information on the Custom Column creation process.

Other Related Links - Advanced Application Data, Control Data Tables, Element Geometric Data Tables, Element Hydraulic Data Tables, Extended Modeling Data Tables, Pattern & Curve Data Tables, SCADA Settings Tables, Water Quality Data Tables, DB Editor, DB Tables.

Extended Modeling Data

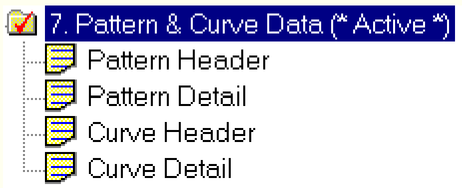
These tables contain Junction Emitter and pressure dependent demand data along with fire-flow and energy data. Click on any section for more information.



Other Related Topics - Advanced Application Data, Control Data Tables, Element Geometric Data Tables, Element Hydraulic Data Tables, Element Information Data Tables, Pattern & Curve Data Tables, SCADA Settings Tables, Water Quality Data Tables, DB Editor, DB Tables.

Pattern & Curve Data

This section of the DB Tables contains all the pattern and curve data. Click on any section for more information.

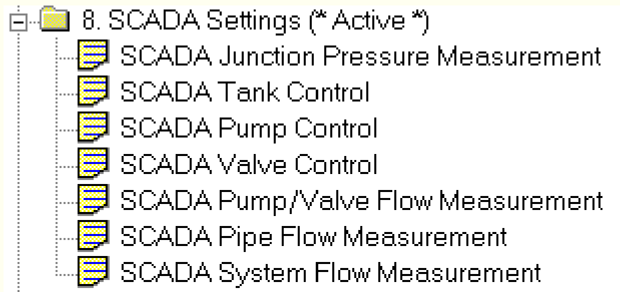


Note - Please do not use these Tables to alter or modify data.

Other Related Topics - Advanced Application Data, Control Data Tables, Element Geometric Data Tables, Element Hydraulic Data Tables, Element Information Data Tables, Extended Modeling Data Tables, SCADA Settings Tables, Water Quality Data Tables, DB Editor, DB Tables.

SCADA Settings

This table contains all the SCADA data. It is recommended not to alter or modify this data through these tables. Refer to the section on SCADA for more information on SCADA Analysis.



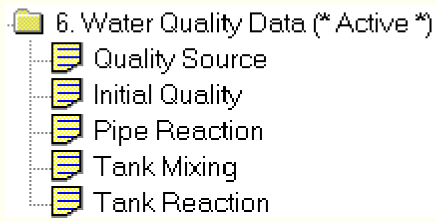
Note - Please do not use these Tables to alter or modify data.

Other Related Topics - Advanced Application Data, Control Data Tables, Element Geometric Data Tables, Element Hydraulic Data Tables, Element Information Data Tables, Extended Modeling Data Tables, Pattern & Curve Data Tables, Water Quality Data Tables, DB Editor, DB Tables.

Water Quality Data

These tables contain all the water quality data such as initial concentrations, source concentrations, concentration type etc.

Click on any section for more information.



NOTE : You may add rows to this table using the **Insert** or the **Append** icon. You may also delete rows from this table by using the **Delete** icon.

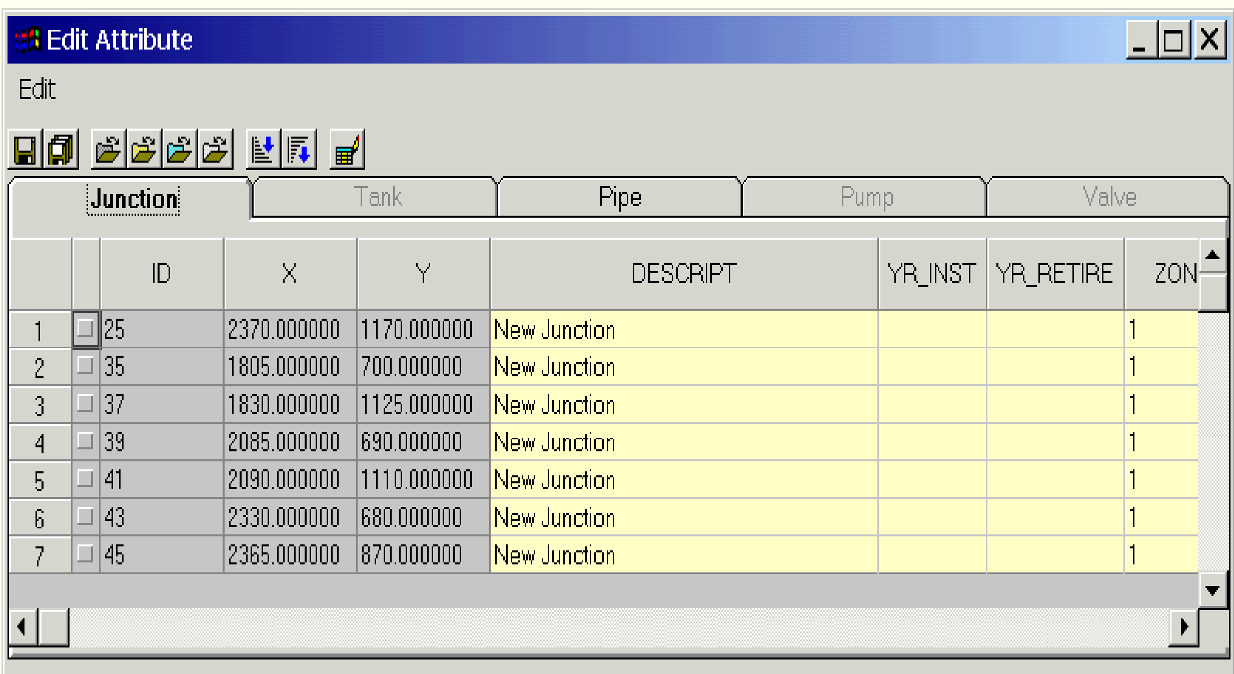
Other Related Topics- Advanced Application Data, Control Data Tables, Element Geometric Data Tables, Element Hydraulic Data Tables, Element Information Data Tables, Extended Modeling Data Tables, Pattern & Curve Data Tables, SCADA Settings Tables, DB Editor, DB Tables.

Edit Domain Attributes

The Edit Domain Attribute dialog box allows the user to edit any/all data fields associated with all the elements included in the currently active domain. This feature allows quick edits of database data and also provides access to all the data field corresponding to the included element (s).

To run the **Edit Domain Attribute** command, from the **InfoWater Control Center -> InfoWater** button **-> Edit** menu, select the **Edit Domain Attributes** command. If you have created a domain, the dialog box below will immediately appear.

Click on the any section below to learn more.



		ID	X	Y	DESCRIPT	YR_INST	YR_RETIRE	ZON
1		25	2370.000000	1170.000000	New Junction			1
2		35	1805.000000	700.000000	New Junction			1
3		37	1830.000000	1125.000000	New Junction			1
4		39	2085.000000	690.000000	New Junction			1
5		41	2090.000000	1110.000000	New Junction			1
6		43	2330.000000	680.000000	New Junction			1
7		45	2365.000000	870.000000	New Junction			1

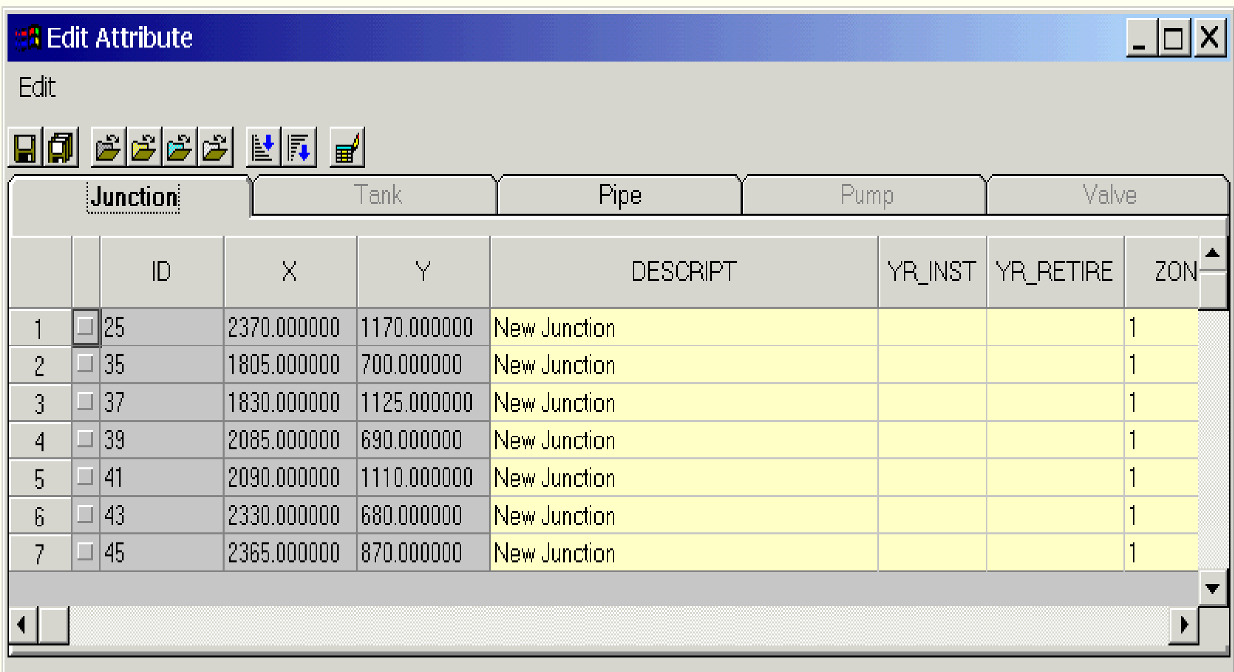
Other Related Topics - Edit Selection Attributes, Emitter, Fire-Flow, Group Editing, Group Editing Methodology, Initial Quality, Pipe Quality, Pipe Status, Pressure Dependent Demand, Quality Source, Tank Mixing, Tank Reaction

Edit Selection Attributes

The Edit Selection Attribute dialog box allows the user to edit any field from a graphically selected set of InfoWater data elements.

To run the Edit Selection Attribute command, from the **InfoWater Control Center -> InfoWater** button **-> Edit** menu, select the **Edit Selection Attributes** command. Drag a selection window across the screen to select the data elements for editing in the Edit Attribute table.

Click on the icons below to learn more.



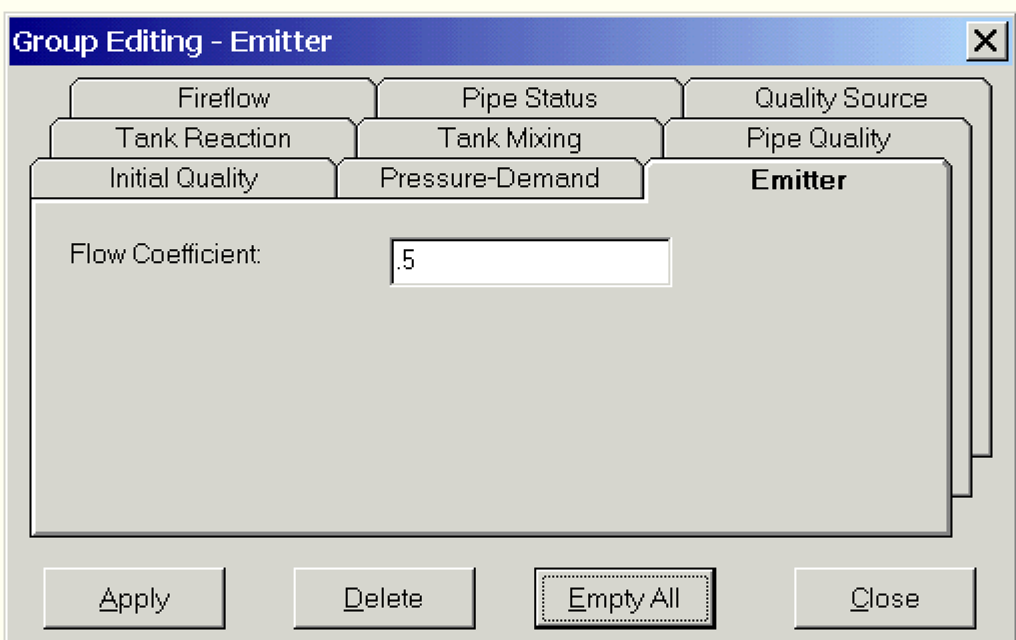
Other Related Topics - Edit Domain Attributes, Emitter, Fire-Flow, Group Editing, Group Editing Methodology, Initial Quality, Pipe Quality, Pipe Status, Pressure Dependent Demand, Quality Source, Tank Mixing, Tank Reaction

Emitter

Emitters are devices associated with junctions that model the flow through a nozzle or orifice that discharges to the atmosphere. The flow rate through the emitter varies as a function of the pressure available at the node. Emitters can be used to model flow through sprinkler systems and irrigation networks. They can also be used to simulate leakage in a pipe connected to the junction (if a discharge coefficient and pressure exponent for the leaking crack or joint can be estimated). A very important characteristic of emitters is their ability to also effectively simulate the effects of pressure-dependent demands.

For nozzles and sprinkler heads the manufacturer usually provides the value of the discharge coefficient in units of gpm/psi0.5 (stated as the flow through the device at a 1 psi pressure drop). Specify the discharge coefficient in the Flow Coefficient value box here. Click here to learn more about the group editing methodology.

Click on any section below to learn more:

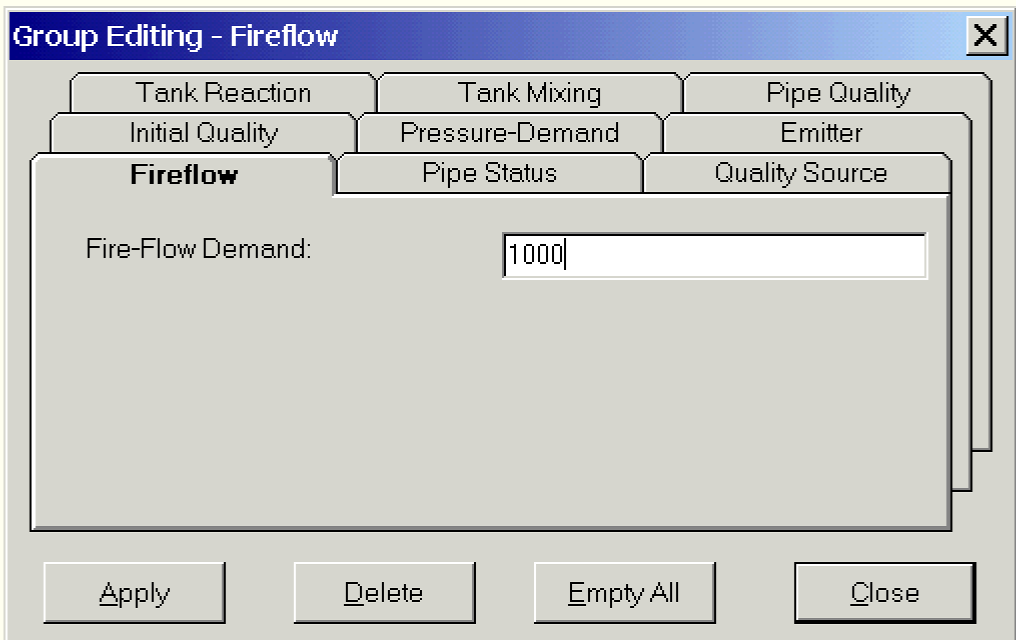


Other Related Topics - Edit Domain Attributes, Edit Selection Attributes, Fire-Flow, Group Editing, Group Editing Methodology, Initial Quality, Pipe Quality, Pipe Status, Pressure Dependent Demand, Quality Source, Tank Mixing, Tank Reaction

Fire-Flow

The fireflow tab is used to control the amount of fireflow assigned to the selected junction nodes. Click here to learn how to run a fireflow simulation. For more information on the Group-Editing Methodology, click here.

Click on any section below to learn more.

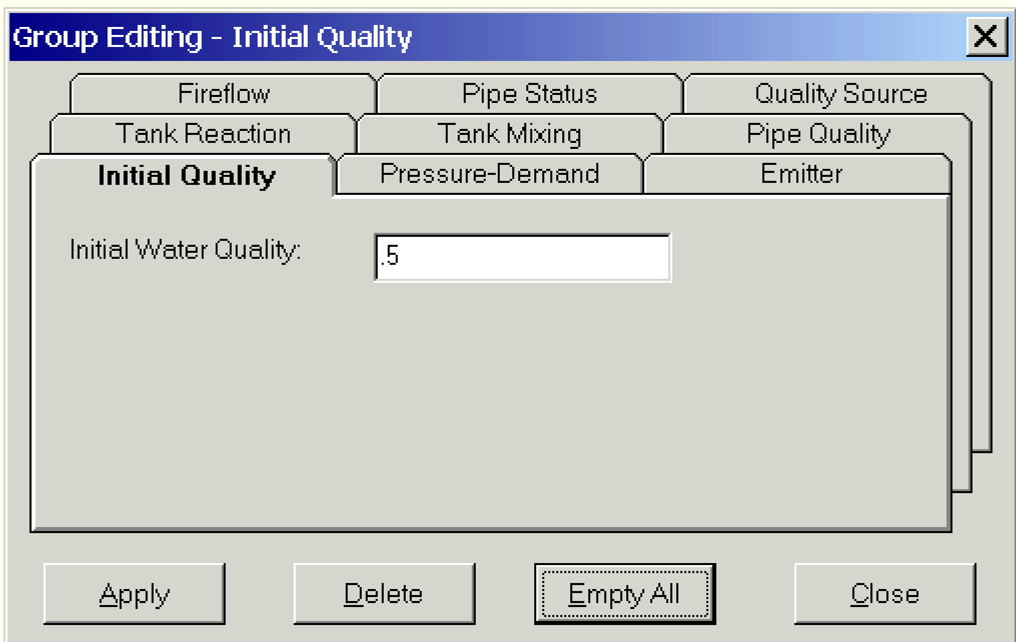


Other Related Topics - Edit Domain Attributes, Edit Selection Attributes, Emitter, Group Editing, Group Editing Methodology, Initial Quality, Pipe Quality, Pipe Status, Pressure Dependent Demand, Quality Source, Tank Mixing, Tank Reaction

Initial Quality

The initial quality tab is used to assign an initial water quality to the selected nodes. This is essential prior to calibrating a water quality model. For more information on the Group-Editing Methodology, click here.

Click on any section below to learn more.

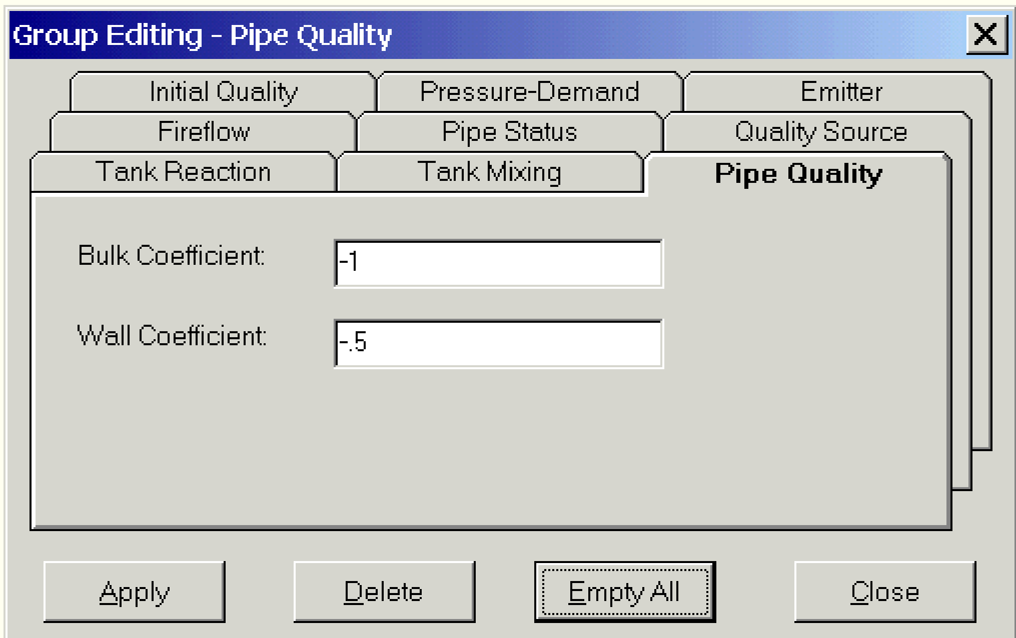


Other Related Topics - Edit Domain Attributes, Edit Selection Attributes, Emitter, Fire-Flow, Group Editing, Group Editing Methodology, Pipe Quality, Pipe Status, Pressure Dependent Demand, Quality Source, Tank Mixing, Tank Reaction

Pipe Quality

The pipe reaction tab is used to assign a different bulk and/or wall reaction coefficient that is different than the global bulk and wall reaction coefficients. This option is valuable for calibrating water quality models with older, tuberculated pipes in the system. For more information on the Group-Editing Methodology, click here.

Click on any section below to learn more.

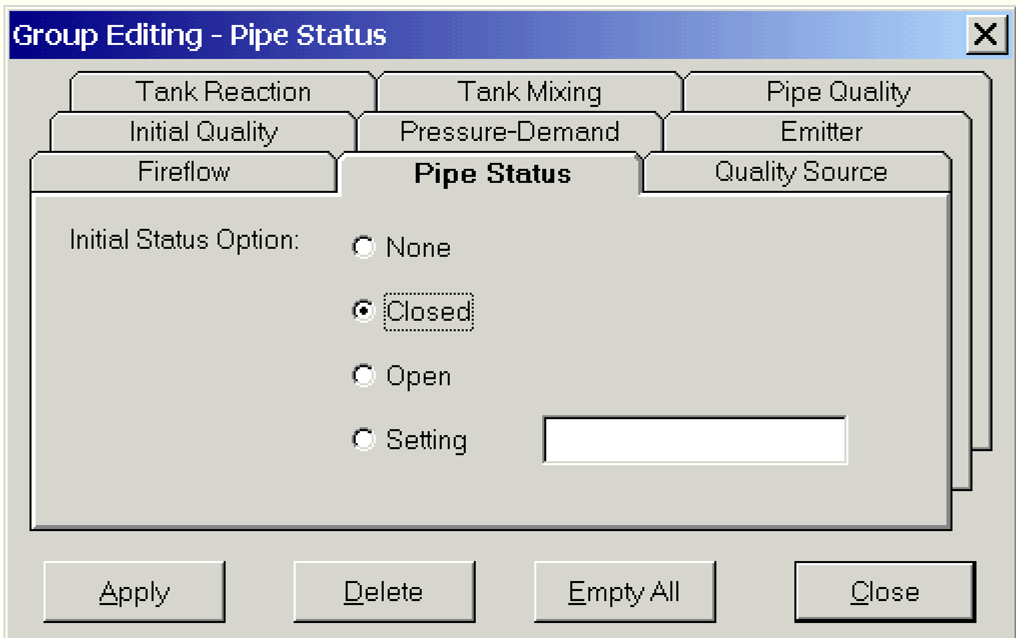


Other Related Topics - Edit Domain Attributes, Edit Selection Attributes, Emitter, Fire-Flow, Group Editing, Group Editing Methodology, Initial Quality, Pipe Status, Pressure Dependent Demand, Quality Source, Tank Mixing, Tank Reaction

Pipe Status

The pipe status tab is used to specify the initial operating status for the selected pipes. For more information on the Group-Editing Methodology, click here.

Click on any section below to learn more.



Other Related Topics - Edit Domain Attributes, Edit Selection Attributes, Emitter, Fire-Flow, Group Editing, Group Editing Methodology, Initial Quality, Pipe Quality, Pressure Dependent Demand, Quality Source, Tank Mixing, Tank Reaction

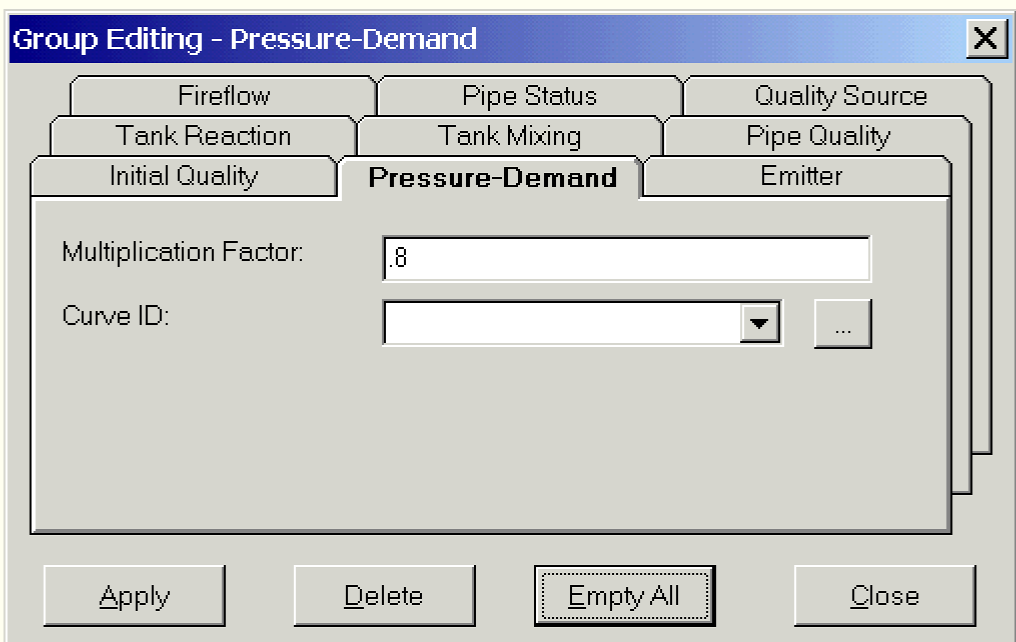
Pressure Dependent Demand

Pressure dependent demand simulations allow the user to create a hydraulic model that more accurately reflects "real-world" conditions.

In some

water systems, as pressure drops, the ability to deliver the modeled demand also decreases with respect to a decrease in system pressure. Knowing this, the user can assign a demand vs. pressure curve for selected nodes in the system. Click here to learn more about the group editing methodology.

Click on any section below to learn more:

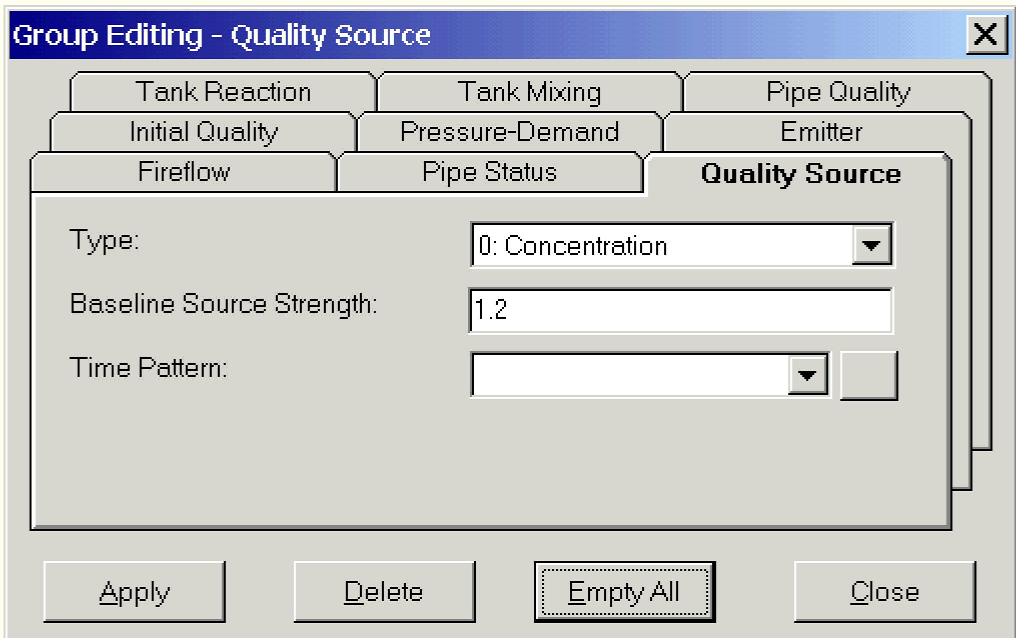


Other Related Topics - Edit Domain Attributes, Edit Selection Attributes, Emitter, Fire-Flow, Group Editing, Group Editing Methodology, Initial Quality, Pipe Quality, Pipe Status, Quality Source, Tank Mixing, Tank Reaction

Quality Source

Use this to assign source concentration entering the network at a specific node. This source might represent the main treatment works, a well head or satellite treatment facility, or an unwanted contaminant intrusion. For more information on the Group-Editing Methodology, click here.

Click on any section for details.

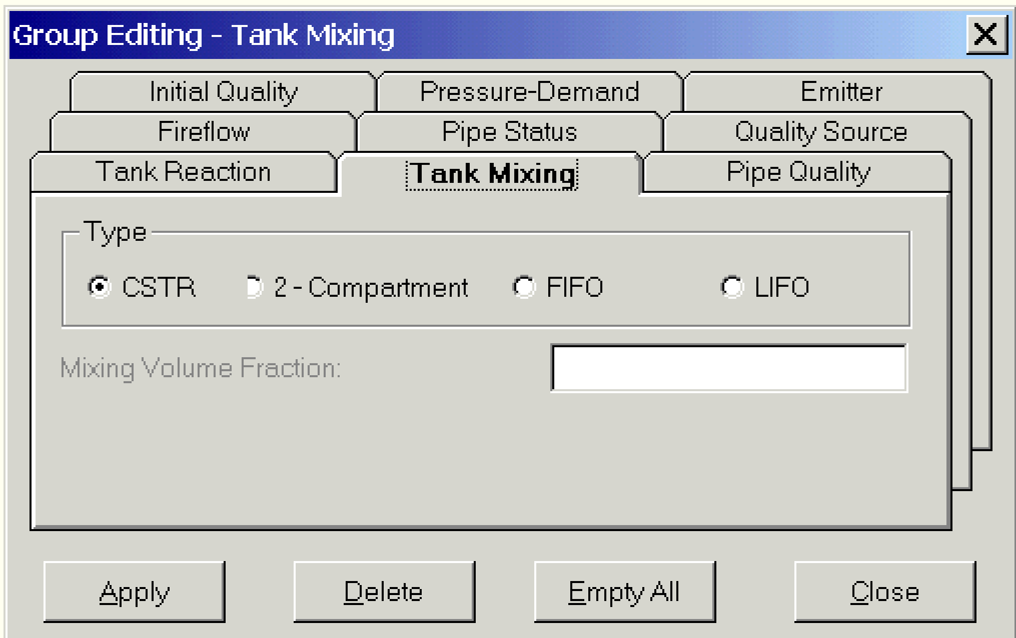


Other Related Topics - Edit Domain Attributes, Edit Selection Attributes, Emitter, Fire-Flow, Group Editing, Group Editing Methodology, Initial Quality, Pipe Quality, Pipe Status, Pressure Dependent Demand, Tank Mixing, Tank Reaction

Tank Mixing

The tank mixing tab is used to evaluate the type of mixing occurring at a tank. Using this tab you can assign globally to all the tanks selected in your domain or through the use of the Group Edit on Selection feature. Click here to learn more about the group editing methodology.

Click on any section below to learn more:

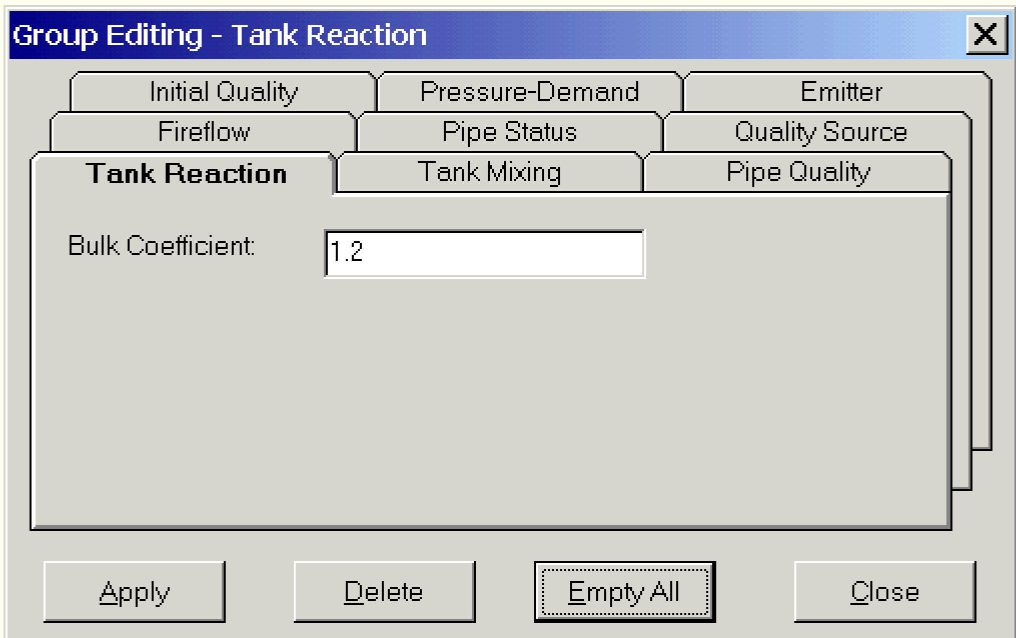


Other Related Topics - Edit Domain Attributes, Edit Selection Attributes, Emitter, Fire-Flow, Group Editing, Group Editing Methodology, Initial Quality, Pipe Quality, Pipe Status, Pressure Dependent Demand, Quality Source, Tank Reaction

Tank Reaction

The tank reaction tab is used to prescribe the measured bulk reaction rate within a storage tank. This value is generally different that the global bulk due to the fact that most tanks have minimal cycling and variable temperatures than the rest of the water system. For more information on the Group-Editing Methodology, click here.

Click on any section below to learn more.

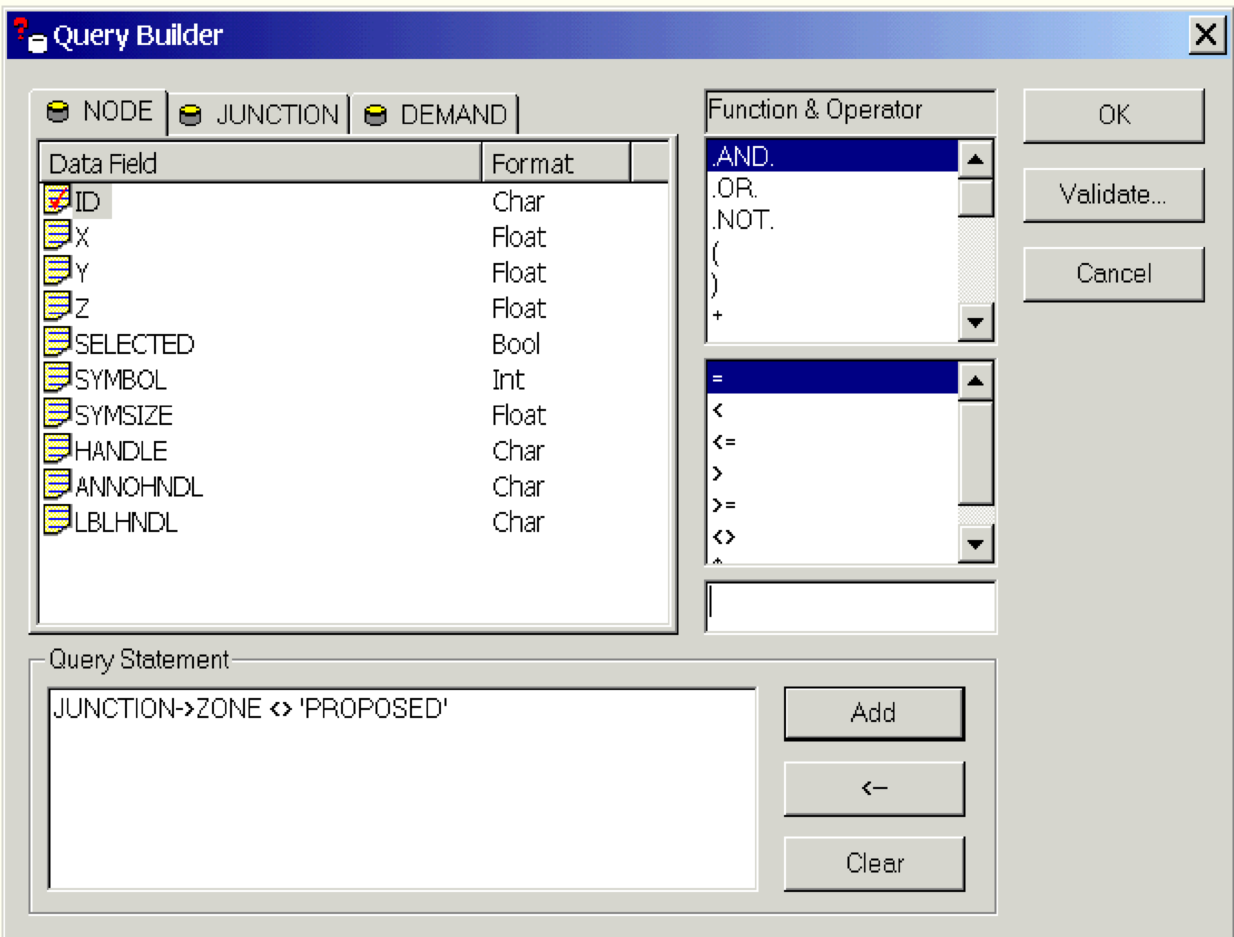


Other Related Topics - Edit Domain Attributes, Edit Selection Attributes, Emitter, Fire-Flow, Group Editing, Group Editing Methodology, Initial Quality, Pipe Quality, Pipe Status, Pressure Dependent Demand, Quality Source, Tank Mixing

Query Builder Dialog Box

Use the Query Builder dialog box to build your DB query. Click here for more information on the DB Query methodology.

Click on any section for more information.



Other Related Topics - DB Query Dialog Box, DB Query Methodology, Query Set Dialog Box, Query Sets Methodology.

Convert Polyline Methodology

The convert polyline command is used to automate the InfoWater pipe and node network creation. If the user has a pipe network in another third party application like AutoCAD or Microstation, the network can be bought into InfoWater as a background layer (.dxf or .dwg) and then converted to an InfoWater pipe and node network.

Methodology

To Convert a Polyline drawing into an InfoWater project use the following procedure:

- Import the external CAD drawing or shapefile into InfoWater. Use the GIS Gateway command under the **InfoWater Control Center -> InfoWater** button **-> Exchange** pull down menu to launch the **GIS Gateway** dialog box.

- Create a new GIS Exchange Cluster by clicking on the **Add** button. Click on the Browse button next to the **GIS Data Source** command and select the AutoCAD drawing that you want to import. For most AutoCAD drawings the **Polyline** choice is the best choice. Select **Pipe Tables** as the **InfoWater Data Type**. Specify **Tabular Join** as your **Relate Type** and **0: Bi-Direction** as your **Update Direction**.

- Under the **Tabular Join** tab of the **GIS Exchange Cluster** choose to **Create New Records**, **Create Unique IDs** and to **Update Geometry Data** and click on the **OK** button at the bottom of the dialog box. Click here to learn more about GIS Exchange Clusters.

- Now click on the **Load** button on your **GIS Gateway** dialog box to launch the **Load Data from GIS Layer or Table** dialog box. Select the Exchange cluster that you want to load and click on the **Load** button.

- Once the drawing has been inserted into your InfoWater project choose the **Convert Polyline** command from your **InfoWater Control Center -> InfoWater** button **-> Exchange** pull down menu to launch the Polyline Conversion dialog box.

- Specify the Node Searching radius. It is recommended that half the size of the node symbol size would be a good beginning point. You may find out the size of the node symbol from your

InfoWater Control Center -> InfoWater button **-> Project Preferences -> Default Symbol Sizes** tab. Also specify your default **Node Type** here. Normally it is good practice to select **Junction** as the default node type.

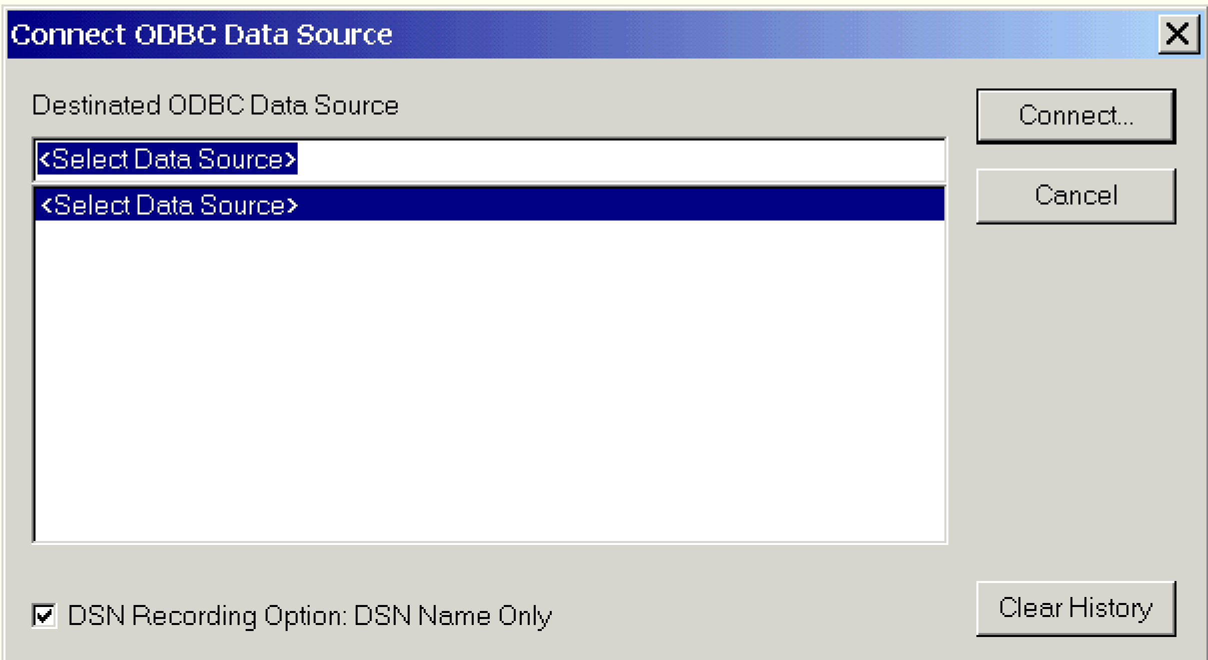
- Choose the **Automatic Mode** option if you want InfoWater to automatically convert your polylines for you.

- Once you have entered and selected all your preference choices, click on **Convert** to launch the conversion process.

Other Related Topics - Convert Polyline Dialog Box, Data Source, EPANET Export, EPANET Import, EPANET v2.x Methodology, Export, Export Manager, Generate File Export, Generate File Import, Generate File Methodology, Import, Import and Export Overview, Import Manager, ODBC dialog box, ODBC Methodology, Select data Fields Export Process, Select data Fields Import Process, **OLE DB Connection**

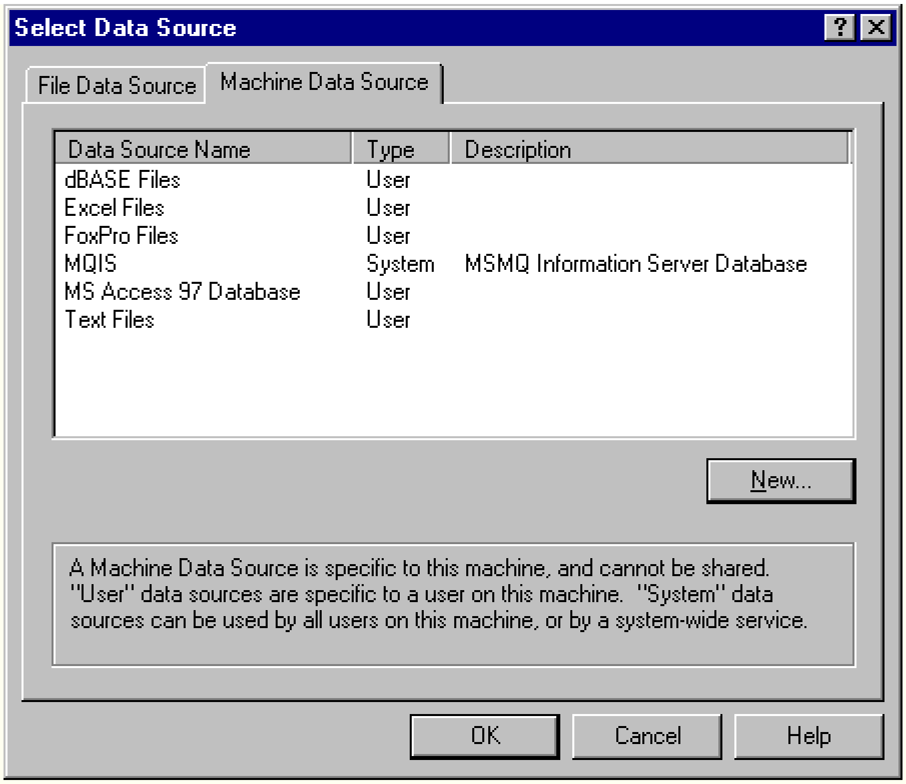
Data Source

The ODBC data source is required in order for InfoWater to use the correct driver when linking to the desired external data file. The data source also supplies InfoWater with the directory path to the external files so that a link can be established. When clicking on the arrow box just to the right of the ODBC Data Source box, the following dialog box appears:



The Destinated ODBC Data Source informs InfoWater of the DSN (Data Source Name) and driver that will be used to map to the external database (in this example, dBase). To use or create a different DSN, select the <Select Data Source> option and click on the Connect button.

Once this is done, you will now see the following dialog box:



The Machine Data Source refers to the DSN's and drivers currently mapped on your operating system. (Note: If you are using Microsoft Windows/NT, these DSN's are the same as those shown through the Windows Control Panel -> ODBC.) It is here that the user selects one of the DSN's available to link to the external file or creates a new DSN tied to a specific driver.

Example - Creating a New DSN and Mapping a Network Path

To create a new DSN, select the New button. The next step is to supply the type of data source. Read the information related to both data sources and make a decision that is relevant to you. Once a selection is made, choose Next. At this point, you need to specify the software driver that is to be used to link to the external file (ex. Excel files need Excel drivers). Once you have selected the driver, choose Next then select Finish.

You will now be asked to supply a unique DSN name, description, and the network path where the files in question are stored. The network path is the directory in which the file you wish to link to is stored (ex. the directory where an Excel file is stored). Once you have done this, select OK. You have now created a new DSN that is ready to link from any InfoWater database table to the external file.

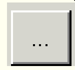
You can create as many DSN's as you like. Each DSN needs to have three things:

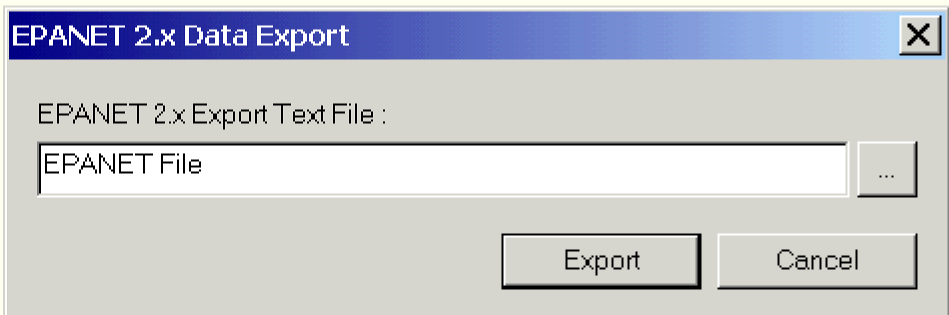
1. A unique name independent of other DSN's
2. A database driver specified in which you are linking. For instance, if your external file is a dBase file, then select the dBase driver.
3. The directory path in which the external data file is stored. Because each DSN is unique, it is recommended that the user create a DSN for each directory where external files are stored. Doing so merely simplifies the linking process.

Other Related Topics - Convert Polyline Dialog Box, Convert Polyline Methodology, EPANET Export, EPANET Import, EPANET v2.x Methodology, Export, Export Manager, Generate File Export,

Generate File Import, Generate File Methodology, Import, Import and Export Overview, Import Manager, ODBC dialog box, ODBC Methodology, Select data Fields Export Process, Select data Fields Import Process, **OLE DB Connection**

EPANET Export

Use the EPANET 2.x Data Export option to export to an EPANET 2 file. Click on the Browse button  to select the location on your hard drive or network where you want to save your file. Also enter the name of the EPANET 2.x file name and click on export to conclude the export process. To learn more about the Export process click here.



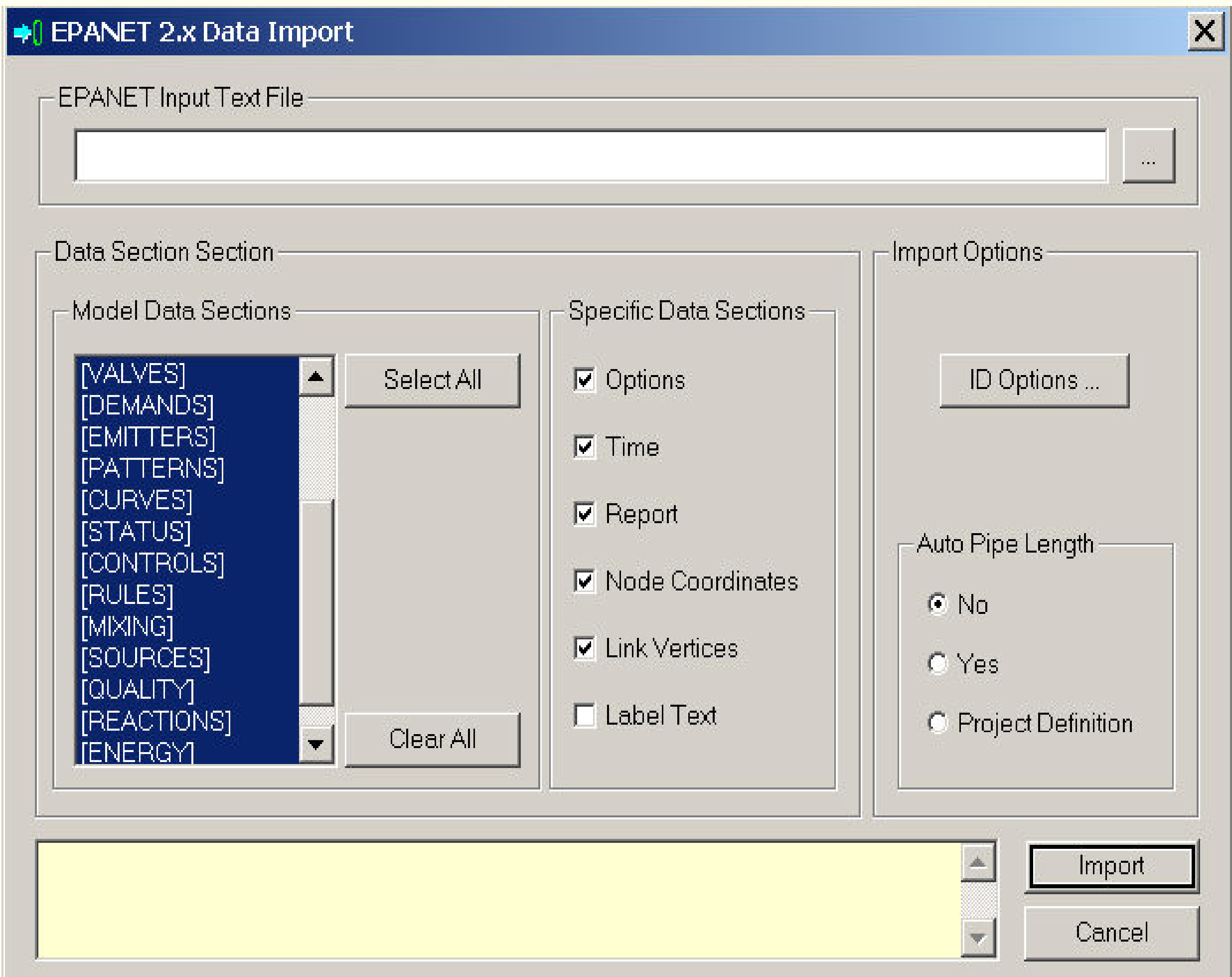
After the export process is completed, you should be able to locate the new file at your selected location.

Other Related Topics - Convert Polyline Dialog Box, Convert Polyline Methodology, Data Source, EPANET Import, EPANET v2.x Methodology, Export, Export Manager, Generate File Export, Generate File Import, Generate File Methodology, Import, Import and Export Overview, Import Manager, ODBC dialog box, ODBC Methodology, Select data Fields Export Process, Select data Fields Import Process, **OLE DB Connection**

EPANET v2.x Import

Use this to import EPANET v2.x files into InfoWater or merge EPANET v2.x files to an existing InfoWater project. To learn more about the Import or merge process click here.

Click on any section below to learn more.



Other Related Topics - Convert Polyline Dialog Box, Convert Polyline Methodology, Data Source, EPANET Export, EPANET v2.x Methodology, Export, Export Manager, Generate File Export, Generate File Import, Generate File Methodology, Import, Import and

Export Overview, Import Manager, ODBC dialog box, ODBC Methodology, Select data Fields Export Process, Select data Fields Import Process, **OLE DB Connection**

EPANET v2.x Import/Export Methodology

Use this to Export to EPANET v2.x or import from EPANET v2.x.

Choose the appropriate direction from below: 

Export Methodology

For Exporting to EPANET v2.x do the following:

- From the **Exchange** pull down menu under the **InfoWater Control Center -> InfoWater** button choose **EPANET v2.x -> Export** option. This launches the EPANET Export dialog box.
- Click on the **Browse** button ... to specify the location where you want to save your EPANET v2.x file and specify a name for the new file.
- Click on Export to conclude the Export process. InfoWater will automatically export an .INP EPANET file.
- Navigate to the location on your hard drive or your Network where your EPANET v2.x file resides and verify the contents by importing into your EPANET program

.

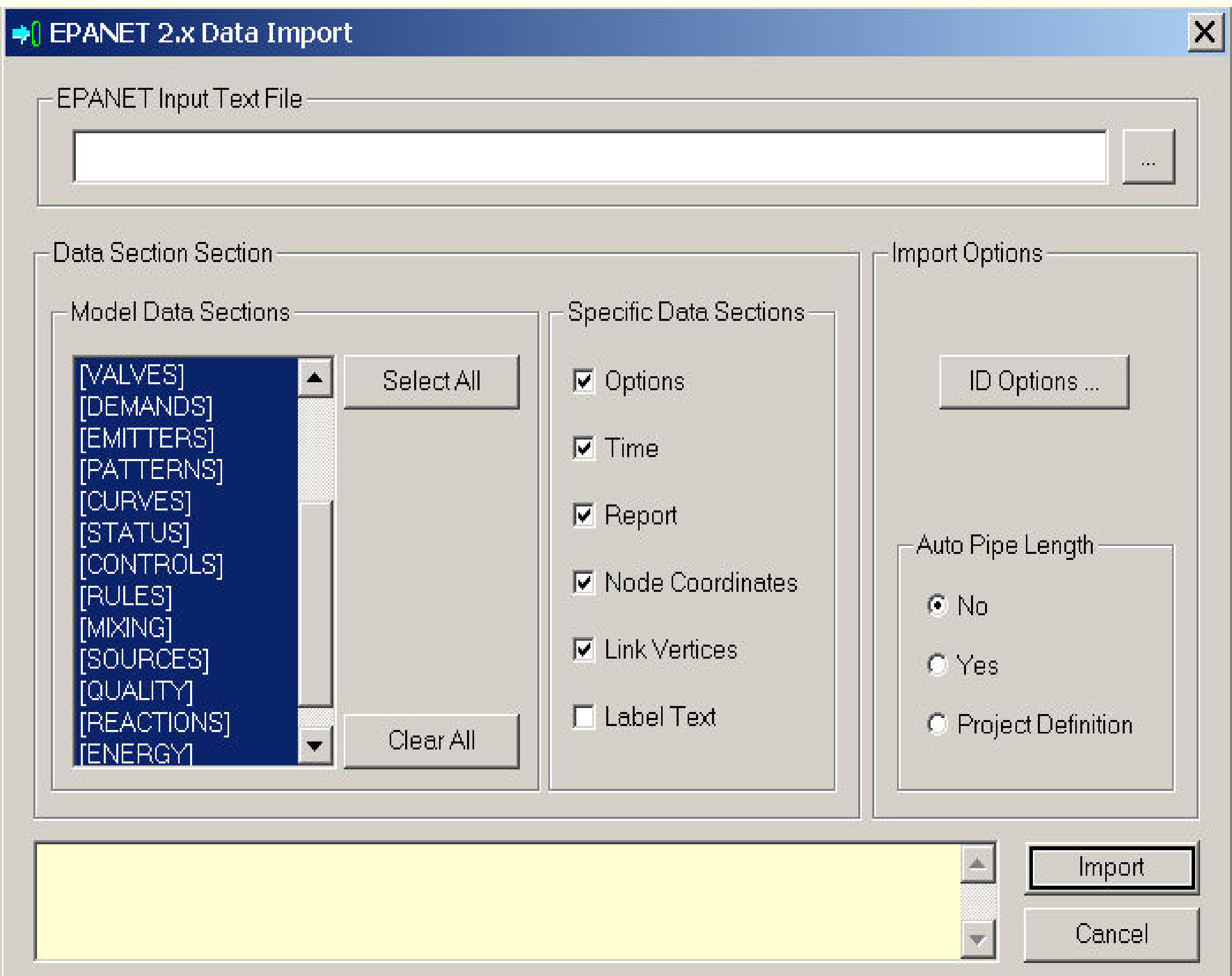
Note - During the EPANET v2.x export process it is normal to see a warning since certain InfoWater elements such as Vacuum Breaker Valves and Float Valves are not supported by EPANET.

Import Methodology

InfoWater can import any EPANET 2.x .INP file. If you have a .NET EPANET file, you will first need to convert it to an .INP file. To do this, launch the EPANET 2.x program and Open your .NET project. Then go to **FILE -> EXPORT** and choose the **NETWORK** option and **Save** it to the desired .INP file name. This will create the .INP file for use by InfoWater.

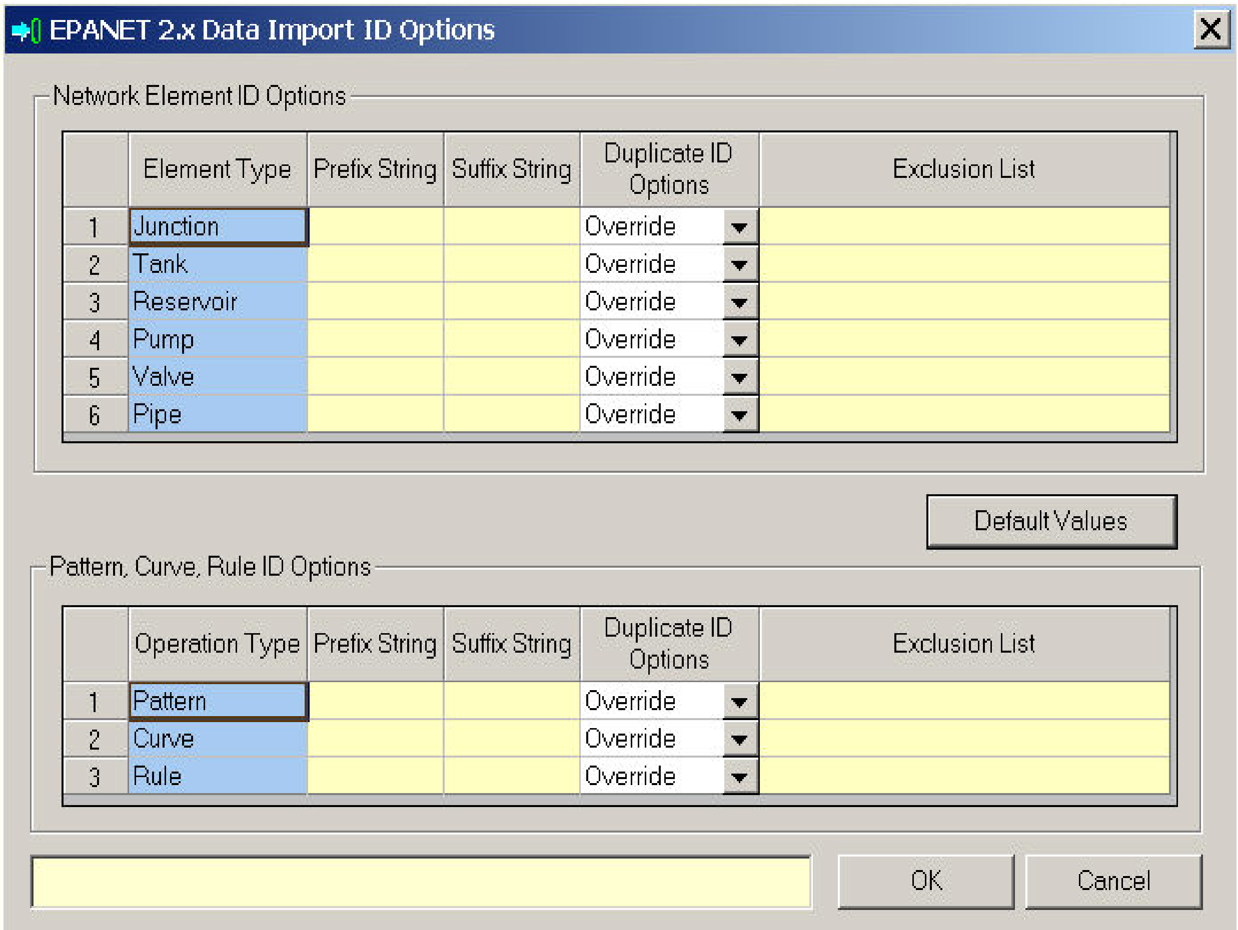
For importing from EPANET v2.x do the following:

- From the **Exchange** pull down menu under the **InfoWater Control Center -> InfoWater** button choose **EPANET v2.x -> Import** option. This launches the EPANET Import dialog box. Click on any section below to learn more.



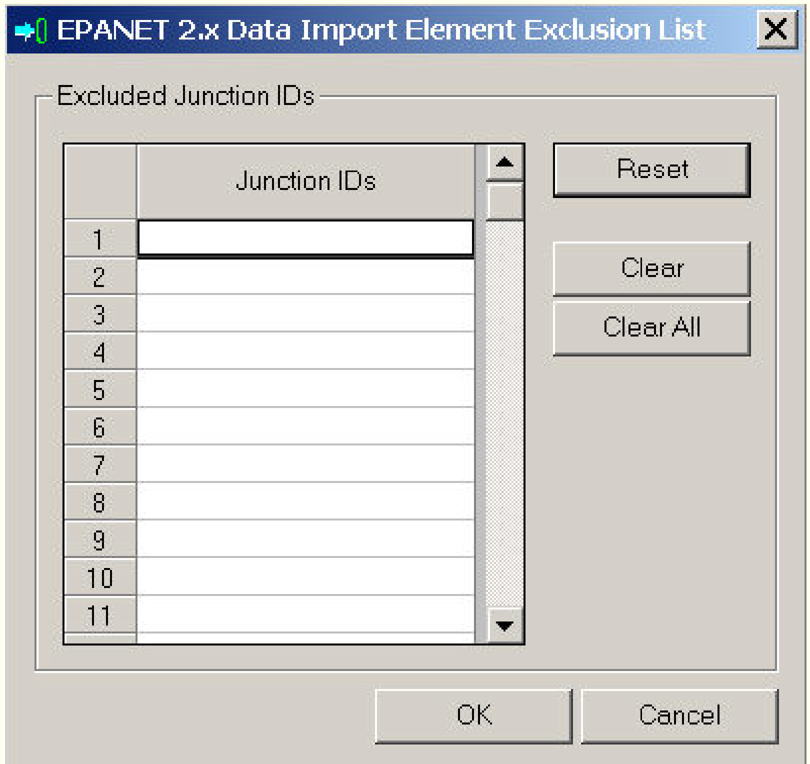
- Specify the EPANET input file. Click on the More button to choose the location of your EPANET file that you want to import into your InfoWater project.
- Select the data that you want to import into InfoWater, by default all the data fields are selected.
- From **Specific Data Sections**, choose the Graphics or the Map related data that you want to import into InfoWater. By default Options, Time, Report, Node Coordinates, and Link Vertices are selected for new project import, while for importing or merging EPANET v2.x files to an existing InfoWater project Node Coordinates and Link Vertices are selected. It is a standard practice to import the Node Coordinates and the Link vertices since InfoWater will need this data to accurately show your model on the map. If Options, Time, Report, and Label Text are checked then the Options, Time, Report, and Label Text definitions used in the EPANET project will override existing definitions. Especially if you are merging EPANET file to an existing project these options (i.e., Options, Time, Report, and Label Text) need to be unchecked otherwise the EPANET project definitions will override the existing project definitions.
- **Auto Pipe Length** is used to specify to InfoWater the process of calculating the InfoWater pipe lengths. If Project definition is chosen then InfoWater will check to see if the EPANET project has defined pipe lengths. If not it will calculate the lengths based on the project coordinates. If the Yes option is chosen then InfoWater will over-write the Pipe lengths with the calculated pipe lengths based on the InfoWater coordinates. Choosing No will not allow InfoWater to overwrite the lengths with the calculated lengths, and by default No is selected.
- **ID Options** 

is a powerful tool to import EPANET v2.x files into InfoWater or merge EPANET v2.x files to an existing InfoWater project though its ID Control features.

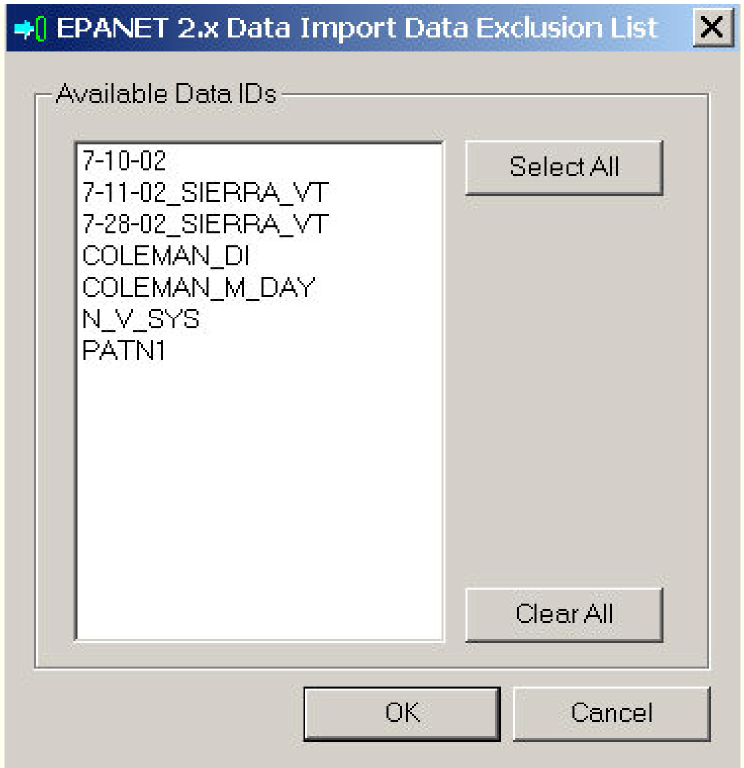


- In the **Data Import ID Options** dialog box an optional prefix and/or suffix can be added to element IDS (Junction, Tanks, Reservoirs, Pumps, Valves, and Pipes) and operational data IDS (Patterns, Curves, and Rules). The prefix and suffix are up to 2 characters long and they can be used in conjunction with each other (i.e., both prefix and suffix can be used at the same time).
- In addition, **ID Options** provides a powerful tool to override, ignore or modify element and operational data IDS upon ID duplication (i.e., same IDS for EPANET data file and existing data base). When you selected Override EPANET data will override existing data filed values, when Ignore is selected EPANET data will be ignored or bypassed, and when Modified ID is selected EPA data will assume a slightly modified ID.

- All elements/operational data IDS listed in the **Exclusion List** will not be imported. To add element ID to the **Exclusion List** double click the filed corresponding to the type of element you want to add to the list. For example, the following dialog box will appear for junctions and you can type up to 25 junction IDS to be included in the exclusion list. Once you have typed the IDS, click **OK**.



- For operational data IDS (Patterns, Curves, and Rules) you can selected the IDS that need to be added to the exclusion list from available data IDS. Once you have selected IDS, click **OK.**



- When you have specified all the options, click on **OK**, and this will take you back to the **EPANET 2.x Data Import** dialog box.
- Finally when you have specified all the parameters for the Import process, click on **Import** to begin the Import process.

Other Related Topics - Convert Polyline Dialog Box, Convert Polyline Methodology, Data Source, EPANET Export, EPANET Import, Export, Export Manager, Generate File Export, Generate File Import, Generate File Methodology, Import, Import and Export Overview, Import Manager, ODBC dialog box, ODBC Methodology, Select data Fields Export Process, Select data Fields Import Process, **OLE DB Connection**

Export

There are many ways data can be exported from InfoWater. Depending on the type of data you want to export from your InfoWater data project choose from among the following.

InfoWater Export Manager - Use the InfoWater Export manager to export shapefiles (**GIS** data), text files and Map Info files (MIF/MID). To learn more about the InfoWater Export Manager click here.

ODBC Exchange - Use the ODBC exchange if you want to link to an external data base such as dBASE, FoxPro, Access etc. Click here to learn more about the ODBC exchange process.

EPANET v2.x - Use this to export EPANET v2.x files. To learn more about the EPANET export process click here.

Generate File - Use this to export ArcInfo Generate files. To learn more about the GENERATE files export process click here.

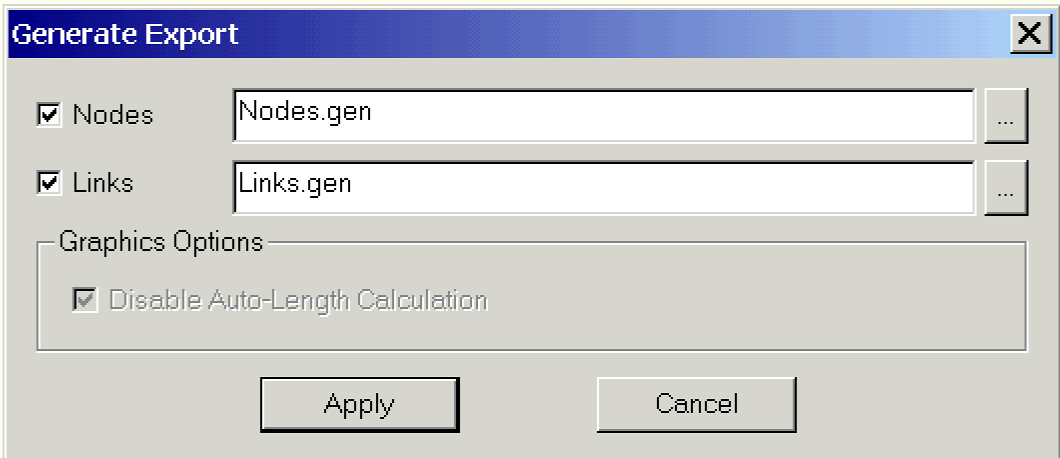
Windows Based Export - Because InfoWater is Windows based, any simulation report or database table that resides in an InfoWater project is capable of being copied at any time. Just open the desired output report or database table and highlight the data you wish to export. Then, right-mouse click over the highlighted area and select Copy. Open any third party package like Microsoft Excel and select Paste from the Edit menu.

Other Related Topics - Convert Polyline Dialog Box**,** Convert Polyline Methodology**,** Data Source**,** EPANET Export**,** EPANET Import**,** EPANET v2.x Methodology**,** Export Manager**,** Generate File Export**,** Generate File Import**,** Generate File Methodology**,** Import**,** Import and Export Overview**,** Import Manager**,** ODBC dialog box**,** ODBC Methodology**,** Select data Fields Export Process**,** Select data Fields Import Process**,** **OLE DB Connection**

Generate File Export

Use this to Export to Generate file. Click here to learn more about the Export methodology.

Click on any portion for more details.

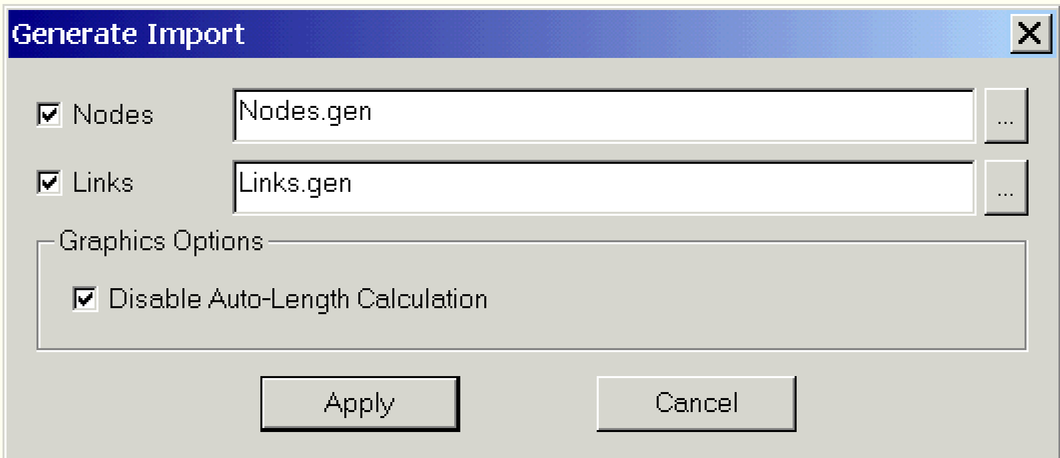


Other Related Topics - Convert Polyline Dialog Box, Convert Polyline Methodology, Data Source, EPANET Export, EPANET Import, EPANET v2.x Methodology, Export, Export Manager, Generate File Import, Generate File Methodology, Import, Import and Export Overview, Import Manager, ODBC dialog box, ODBC Methodology, Select data Fields Export Process, Select data Fields Import Process, **OLE DB Connection**

Generate File Import

Use this to Import into InfoWater from Generate files. Click here to learn more about the Import methodology.

Click on any portion for more details.



Other Related Topics - Convert Polyline Dialog Box, Convert Polyline Methodology, Data Source, EPANET Export, EPANET Import, EPANET v2.x Methodology, Export, Export Manager, Generate File Export, Generate File Methodology, Import, Import and Export Overview, Import Manager, ODBC dialog box, ODBC Methodology, Select data Fields Export Process, Select data Fields Import Process, **OLE DB Connection**

ArcInfo Generate File

Use this to Export to Generate file or import from Generate file into InfoWater. Choose the appropriate direction from below: 

__Export Methodology__

For Exporting to Generate File do the following:

- From the **InfoWater Control Center -> InfoWater** button **-> Exchange** pull down menu choose **Generate File -> Export** option. This launches the Export dialog box as shown below :
- Choose the appropriate generate files you want to create and specify the location on your network or your hard drive you want to save your generate files by clicking on the **Browse** button .
- Click on Apply to create the generate files

Import Methodology

For Importing Generate Files do the following:

- From the **InfoWater Control Center -> InfoWater** button **-> Exchange** pull down menu choose **Generate File -> Import** option. This launches the Import dialog box as shown below :
- Choose the appropriate generate files you want to import and specify the location on your network or your hard drive you want to import your generate files from by clicking on the **Browse** button .
- Choose to Enable or Disable the InfoWater Auto-length calculation. Enabling the InfoWater auto length calculation will over write the lengths with InfoWater calculated pipe lengths based on the AutoCAD coordinates.
- Click on Apply to create the generate files.

Note: An ArcInfo Generate file typically contains only element coordinates. It is essential to also import a CSV file that contains all the database information.

The following illustrates a sample node and link ESRI generate file. Refer to the ESRI Arc/Info documentation for more information on the Generate file format:

Node Generate File In the following illustration, several nodes are written to the Generate file. For each node, the ID, X-coordinate, Y-coordinate, and a placeholder for a Z value are provided: 1 1678.750000 1703.500000 0.000000

3 1993.750000 1703.500000 0.000000

5 2193.250000 1703.500000 0.000000

7 3030.000000 1715.000000 0.000000

END

Other Related Topics - Convert Polyline Dialog Box, Convert Polyline Methodology, Data Source, EPANET Export, EPANET Import, EPANET v2.x Methodology, Export, Export Manager, Generate File Export, Generate File Import, Import, Import and Export Overview, Import Manager, ODBC dialog box, ODBC Methodology, Select data Fields Export Process, Select data Fields Import Process, **OLE DB Connection**

Import

There are many ways data can be imported into InfoWater. Depending on the type of data you want to import into your InfoWater data project choose from among the following.

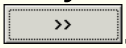
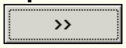
InfoWater Import Manager - Use the InfoWater Import manager to bring in data such as shapefiles (GIS data), text files and Map Info files (MIF/MID). To learn more about the InfoWater Import Manager click here.

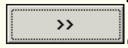
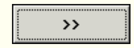
ODBC Exchange - Use the ODBC exchange if you want to link to an external data base such as dBASE, FoxPro, Access etc. Click here to learn more about the ODBC exchange process.

OLE DB Connection - Use the OLE DB exchange if you want to link to an external data base such as Access.

EPANET v2.x - Use this to bring in EPANET v2.x files. To learn more about the EPANET import process click here.

Generate File - Use this to import ArcInfo Generate files. To learn more about the GENERATE files import process click here.

H2OMAP Project - (can be accessed through the **Project** sub menu) Use this to import an existing H2OMAP Project into InfoWater. Be sure to set your drawing limits by choosing the **Create New** option and double arrow button . Now chose the import button  and browse to your H2OMAP Project. Choose the default settings for the remaining options and then Ok to close the last dialogue box. InfoWater will create an InfoWater database and a drawing. Navigate down to the appropriate H2OMAP map file (.H2O extension) and click on it to select it. InfoWater will first create the InfoWater project database and subsequently create an InfoWater drawing from the database. After a successful import, save the newly created InfoWater project into a location on your hard-drive by using the **Save As** command under the **File** menu.

H2ONET Project -(can be accessed through the **Project** sub menu) Use this to import an existing H2ONET Project into InfoWater. Be sure to set your drawing limits by choosing the **Create New** option and double arrow button . Now chose the import button  and browse to your H2ONET Project. Choose the default settings for the remaining options and then Ok to close the last dialogue box. InfoWater will create an InfoWater database and a drawing. Navigate down to the appropriate H2ONET file (.DWG extension) and click on it to select it. InfoWater will first create the InfoWater project database and subsequently create an InfoWater drawing from the database. After a successful import, save the newly created InfoWater project into a location on your hard-drive by using the **Save As** command under the **File** menu.

Issues to Consider When Importing GIS data into InfoWater

If you import data from a GIS you should ensure the following before importing into InfoWater:

- All pipes (arcs, lines) in the GIS are connected to two unique nodes.
- All pipes in the GIS are snapped together where those pipes represent connected pipes in your water distribution system. In other words, be sure that where two or more pipes should be connected, those pipes should share the same node in the GIS or other external database. It is common to find that in the GIS those two or more pipes are actually dead-end pipes, each pipe with a unique, different node, and those nodes simply share the same location, appearing to a map viewer as being the same node.
- There are no disconnected nodes in the GIS database. Each node should be connected to at least one pipe in the GIS.
- The graphical representation of the pipes in the GIS corresponds to the values in your database that represent the from- and to-node identifiers. In other words, if a pipe in your GIS database has a from-node value of “1” and a to-node value of “2”, then the graphical representation of that pipe on the map display shows that pipe starting at node “1” and ending at node “2”.

Note: If you do not meet the above-listed criteria, you may experience one or more errors when running a simulation in InfoWater. The two most common situations that may occur if you do not meet the above-listed criteria before importing are the following:

- Isolated Node – One or more nodes are disconnected from the network.
- Coincident Nodes – Two or more nodes share the same location, appearing to a viewer as a single node.

InfoWater has tools that will help you correct these problems in the event that your GIS is too burdensome. From the **InfoWater Control Center -> InfoWater** button **-> Utilities** menu, point to **Connectivity** and select **Orphan Nodes** or **Orphan Pipes**. By doing this, InfoWater will identify which nodes are not connected to pipes and which pipes do not have both an upstream and downstream node.

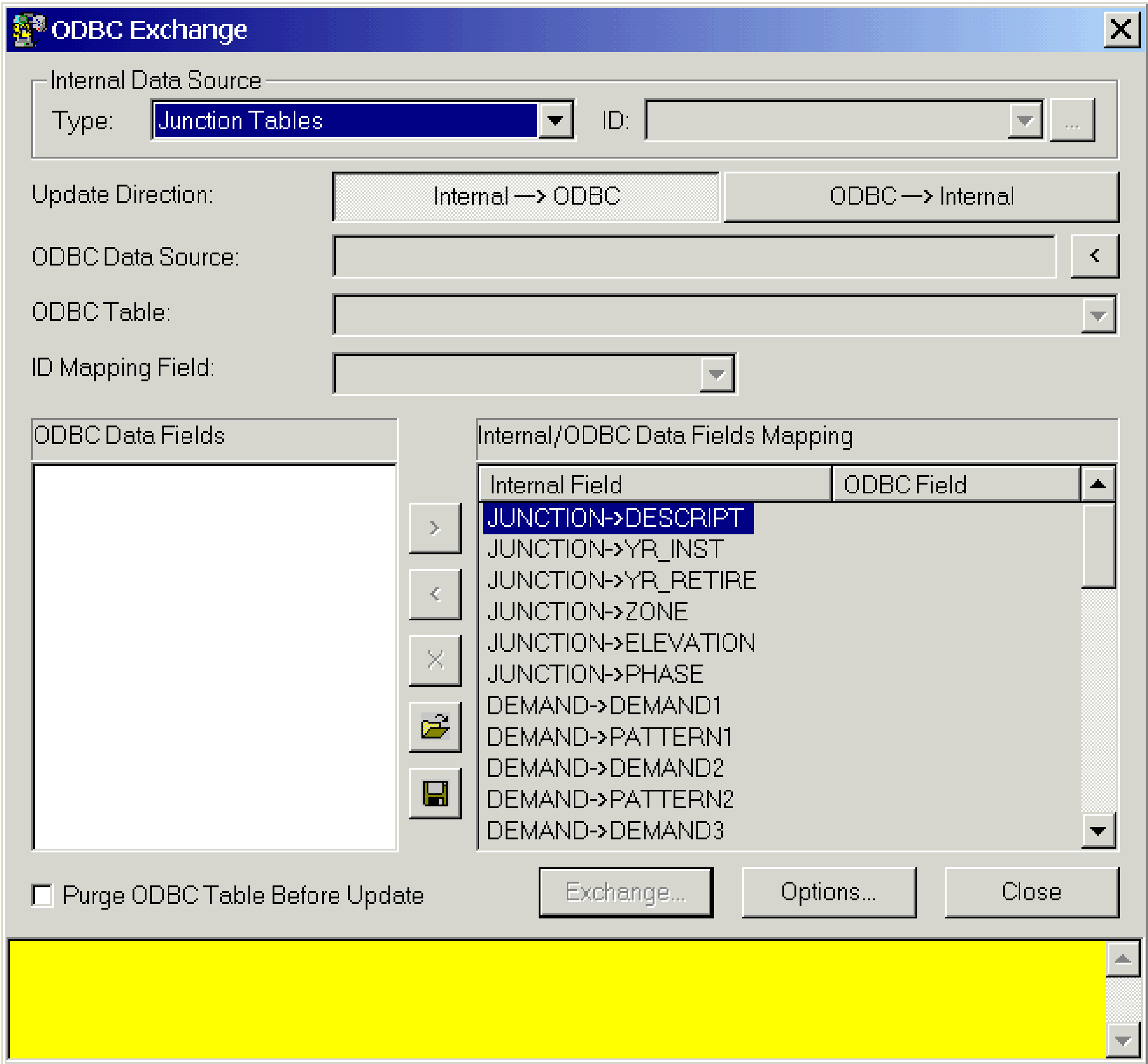
Other Related Topics - Convert Polyline Dialog Box**,** Convert Polyline Methodology**,** Data Source**,** EPANET Export**,** EPANET Import**,** EPANET v2.x Methodology**,** Export**,** Export Manager**,** Generate File Export**,** Generate File Import**,** Generate File Methodology**,** Import and Export Overview**,** Import Manager**,** ODBC dialog box**,** ODBC Methodology**,** Select data Fields Export Process**,** Select data Fields Import Process**,** **OLE DB Connection**

ODBC dialog box

The Open Database Connectivity (ODBC) module of InfoWater allows the user to actively link to any external database and import that data into the current InfoWater session. Likewise, the user can also update an external database with InfoWater data.

To initiate the ODBC Exchange, from the **InfoWater Control Center -> InfoWater** button **-> Exchange** menu, select **ODBC Exchange** and the following dialog box appears. To learn more about the ODBC Exchange procedure click here.

Click on any portion to learn more.



Other Related Topics - Convert Polyline Dialog Box, Convert Polyline Methodology, Data Source, EPANET Export, EPANET Import, EPANET v2.x Methodology, Export, Export Manager, Generate File Export, Generate File Import, Generate File Methodology, Import, Import and Export Overview, Import Manager, ODBC Methodology, Select data Fields Export Process, Select data Fields Import Process, **OLE DB Connection**

ODBC Methodology

The Open Database Connectivity (ODBC) module of InfoWater allows the user to actively link to any external database (Such as dBASE, Excel, ForPro, MQUIS (SQL server), MS Acess etc) and import that data into the current InfoWater session. Likewise, the user can also update an external database with the InfoWater data.

Methodology

Use the following procedure to use the InfoWater ODBC data exchange platform.

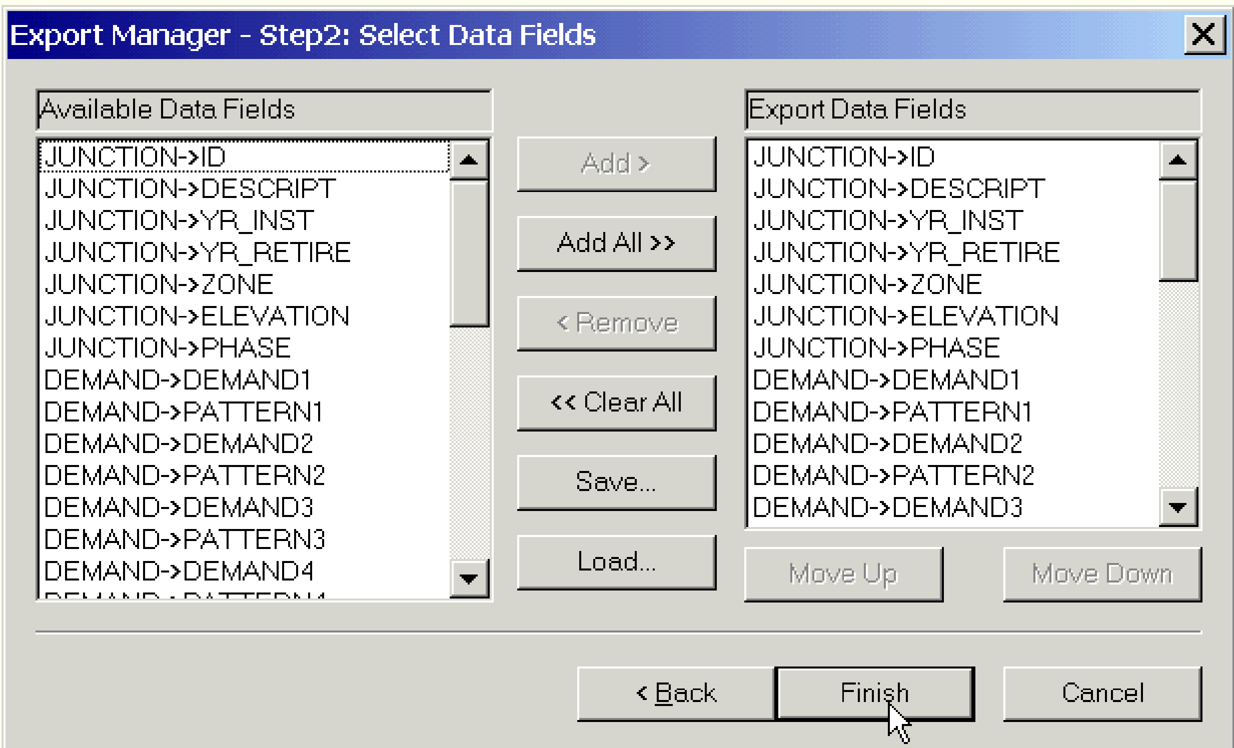
- Launch the ODBC Exchange dialog box from the **InfoWater Control Center -> InfoWater** button **-> Exchange** menu.
- Choose the InfoWater Data Source. These data sources correspond with the different InfoWater tables.
- Choose the appropriate Update direction. In order to import data choose **ODBC->InfoWater** and to export data out of the InfoWater project choose **InfoWater ->ODBC.**
- Click on the **ODBC Data Source** to specify the external data base that you want to link to. The ODBC data source is required in order for InfoWater to use the correct driver when linking to the desired external data file. The data source also supplies InfoWater with the directory path to the external files so that a link can be established. Click here to learn more about ODBC Data Sources.
- Once the Data Source has been established, choose the ODBC table and the ID mapping field (when importing into InfoWater)
- All the available fields (during the import process) will be listed under the ODBC Data fields section of the ODBC dialog box. Click on the appropriate field under the ODBC fields section and then select the corresponding InfoWater field by clicking on it and then click on the mapping arrow **>** to map the two fields. Continue this process till all the desired ODBC fields have been successfully associated with the InfoWater fields.
- Finally when all the fields have been mapped, choose the **Options** button to specify your Import or Export options and then click on **Exchange** to start the exchange process.

Other Related Topics - Convert Polyline Dialog Box, Convert Polyline Methodology, Data Source, EPANET Export, EPANET Import, EPANET v2.x Methodology, Export, Export Manager, Generate File Export, Generate File Import, Generate File Methodology, Import, Import and Export Overview, Import Manager, ODBC dialog box, Select data Fields Export Process, Select data Fields Import Process, **OLE DB Connection**

Select Data Fields to Export

Use this to select the export data fields from among the available ones to export. The available InfoWater fields are listed under the Available Data fields section of the dialog box. Click on any of the fields that you want to export and click on the Add button. Choose Add All to export all the fields.

Click on any portion to learn more.

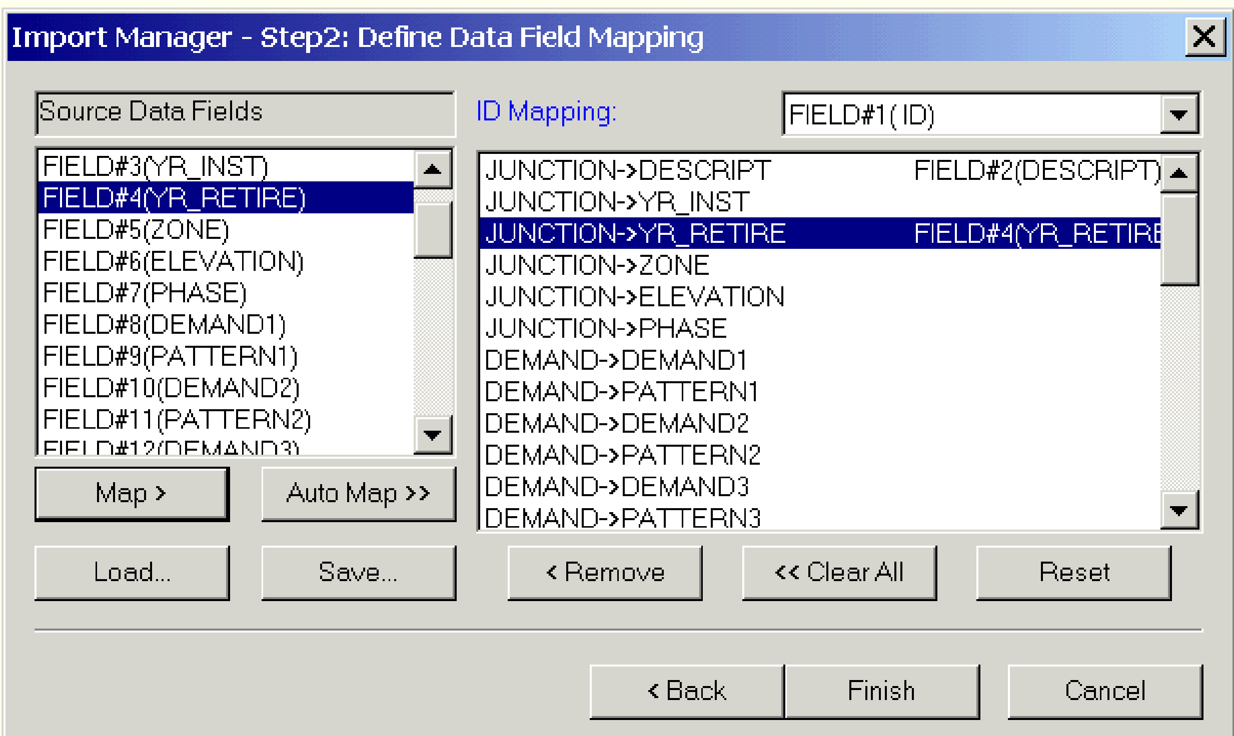


Other Related Topics - Convert Polyline Dialog Box, Convert Polyline Methodology, Data Source, EPANET Export, EPANET Import, EPANET v2.x Methodology, Export, Export Manager, Generate File Export, Generate File Import, Generate File Methodology, Import, Import and Export Overview, Import Manager, ODBC dialog box, ODBC Methodology, Select data Fields Import Process, **OLE DB Connection**

Select Data Fields Import Process

Use this to associate the Imported fields with the different InfoWater Data fields. The list of available field in the newly to be imported file are included under the Source Data Fields section. The available InfoWater data fields are included on the right window. Left click on any field under the source Data fields and left click on the appropriate InfoWater data field and then click on Map to associate the two fields. the most important step in this process is the InfoWater ID mapping where the field from the Source data field that contains a unique identifier is declared as the InfoWater ID.

Click on any portion to learn more.

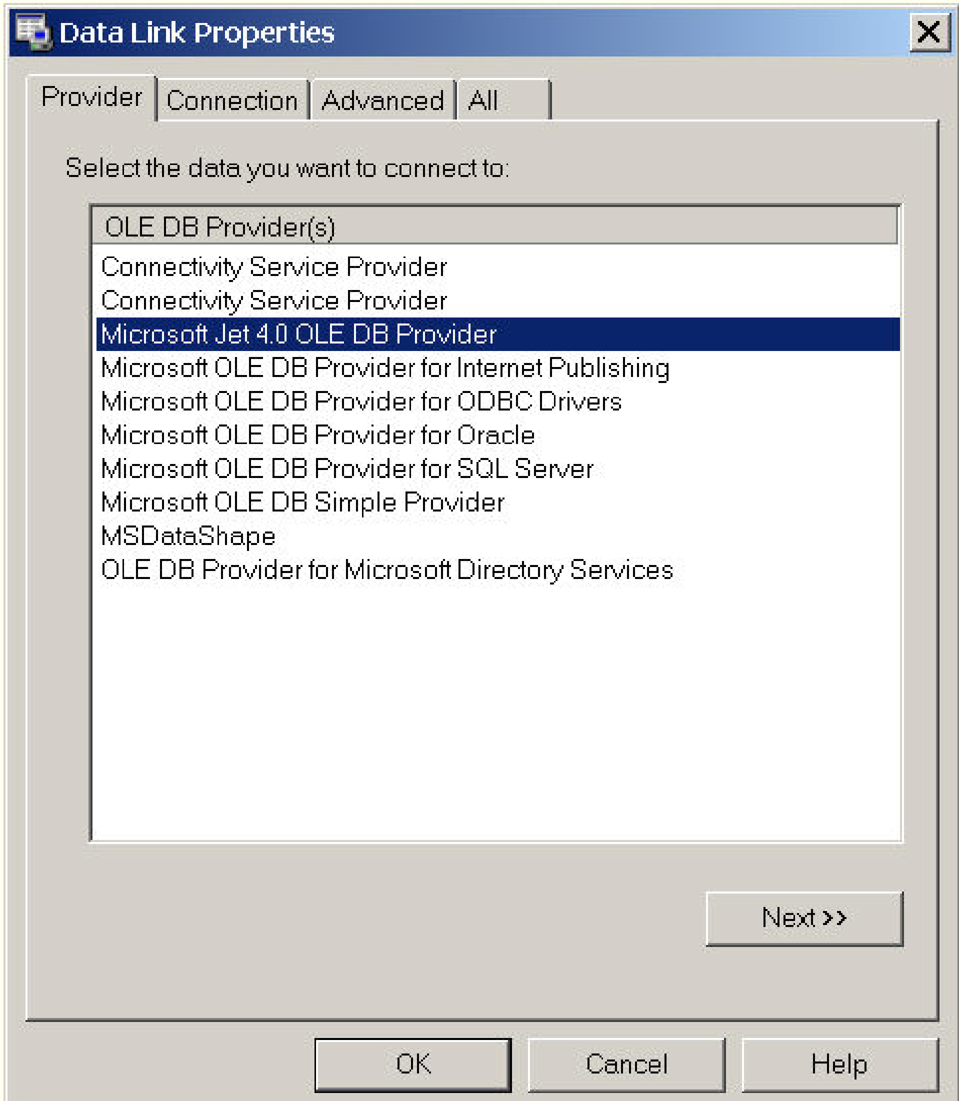


Other Related Topics - Convert Polyline Dialog Box, Convert Polyline Methodology, Data Source, EPANET Export, EPANET Import, EPANET v2.x Methodology, Export, Export Manager, Generate File Export, Generate File Import, Generate File Methodology, Import, Import and Export Overview, Import Manager, ODBC dialog box, ODBC Methodology, Select data Fields Export Process, OLE DB Connection

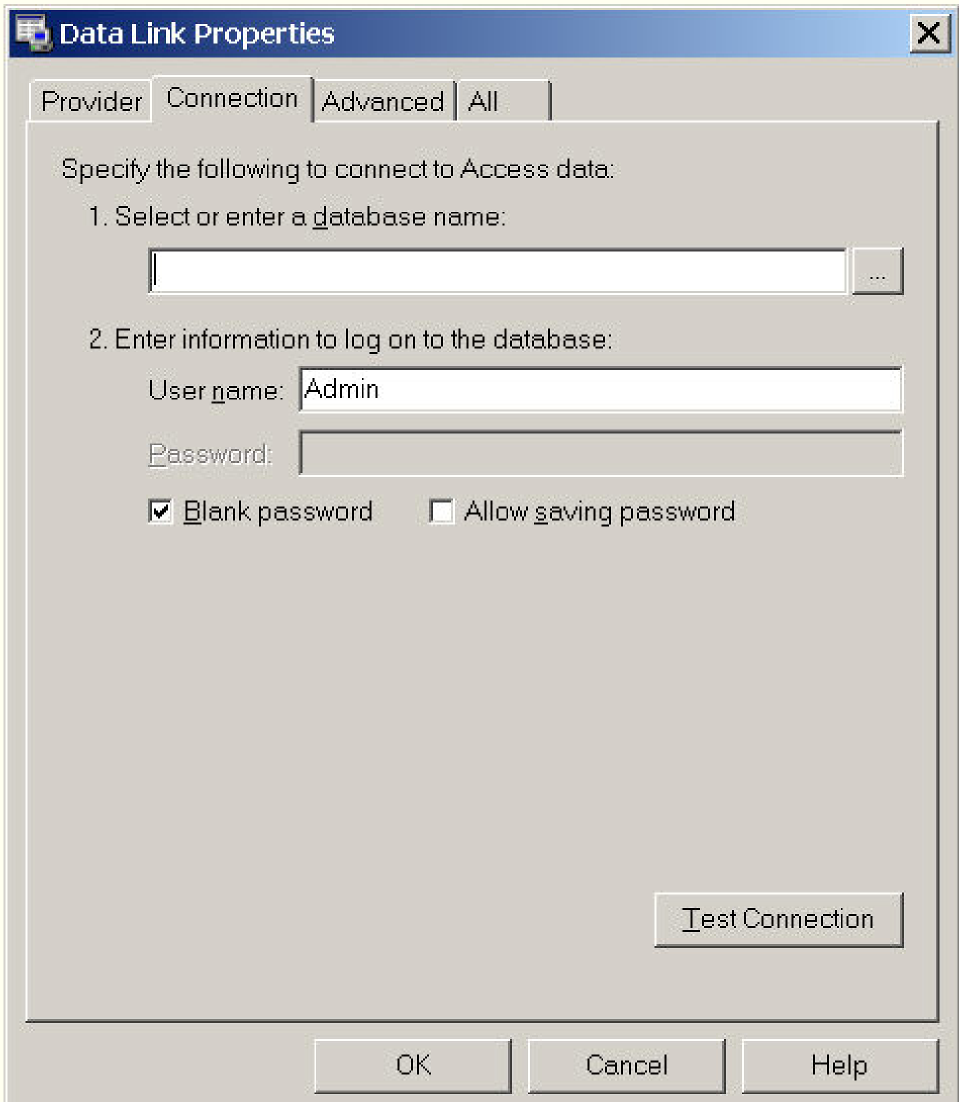
OLE DB

The OLE DB module of InfoWater allows the user to actively link to any external database and import that data into the current InfoWater session.

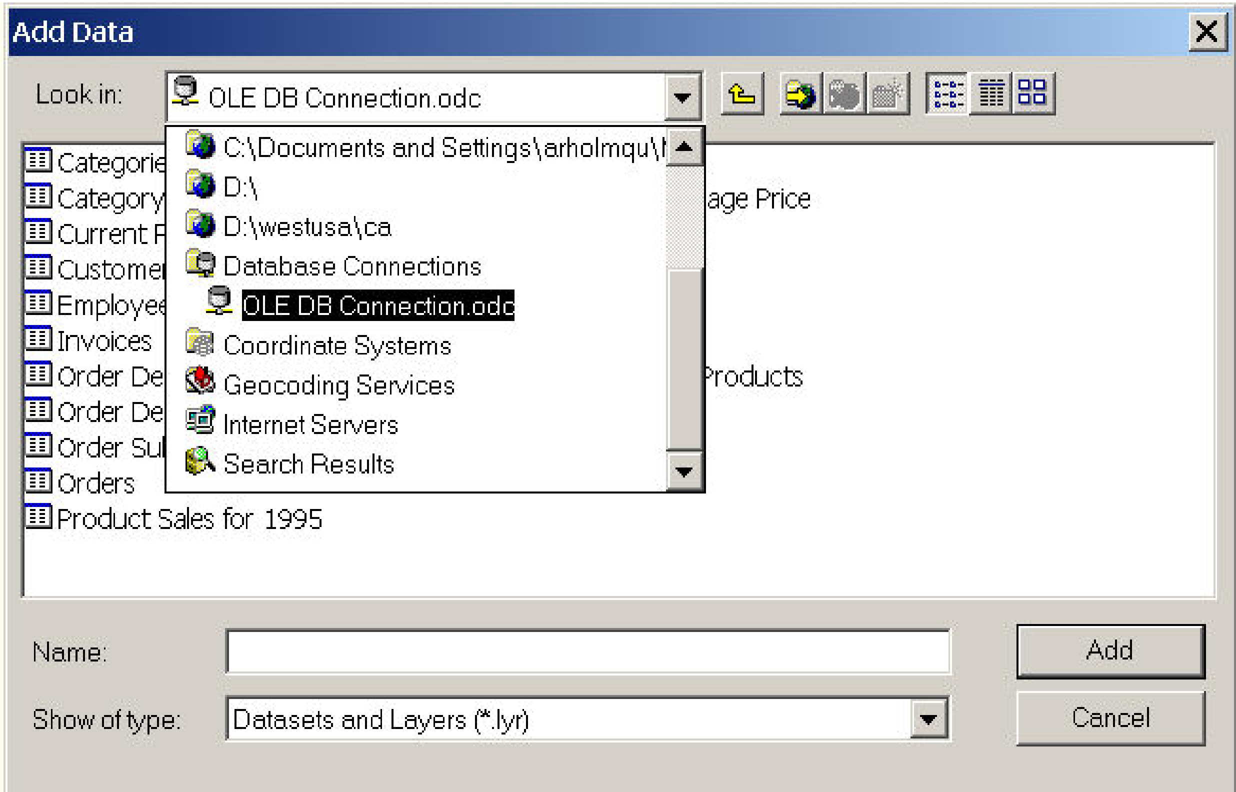
To initiate the OLE DB Exchange, from **ArcCatalog** expand the **Database Connections**. Then Double Click on the **Add OLE DB Connection** option. Now choose the appropriate **Microsoft OLE DB Provider**.



Choose the **Next** button and **Browse** to your desired database. It is recommended you click on **Test Connection** before you close this window. Click on the **OK** button to create the connection.



To add data to your project, Click on the **Add Layer** button and browse to your **Database Connections**. You will now be able to choose your **OLE DB Connection** and add appropriate files to add to your project.



Other Related Topics - Convert Polyline Dialog Box, Convert Polyline Methodology, Data Source, EPANET Export, EPANET Import, EPANET v2.x Methodology, Export, Export Manager, Generate File Export, Generate File Import, Generate File Methodology, Import, Import and Export Overview, Import Manager, ODBC Methodology, Select data Fields Export Process, Select data Fields Import Process, **OLE DB Connection**

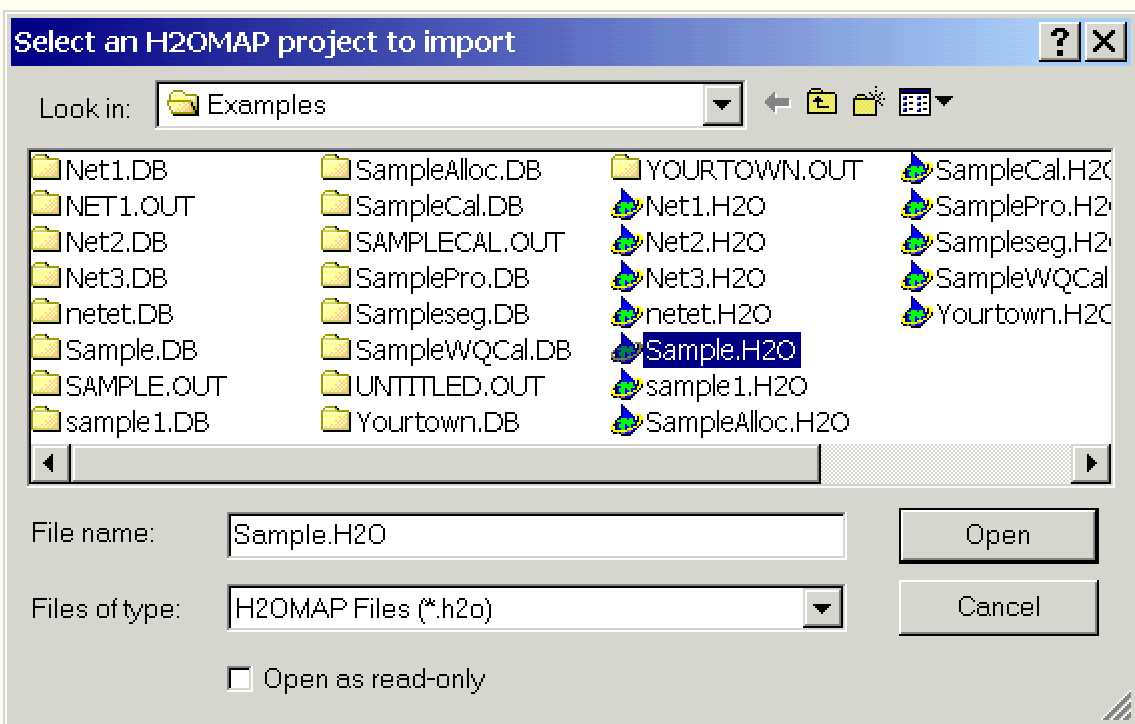
Import H2OMAP

Use this to import an existing H2OMAP Project into InfoWater. InfoWater will create an InfoWater database and a drawing from the H2OMAP project.

Be sure to set your drawing limits before you import your H2OMAP project.

Click on the InfoWater Button 

, browse to the Project menu and select the Import H2OMAP option. Navigate down to the appropriate H2OMAP file (.H2O extension) and click on it to select it. InfoWater will first create the InfoWater project database and subsequently create an InfoWater drawing from the database. After a successful import, save the newly created InfoWater project into a location on your hard-drive by using the **Save As** command under the **File** menu.



Note - All versions including and subsequent to **H2OMAP Water Version 3.0** can be imported into InfoWater using the process discussed above.

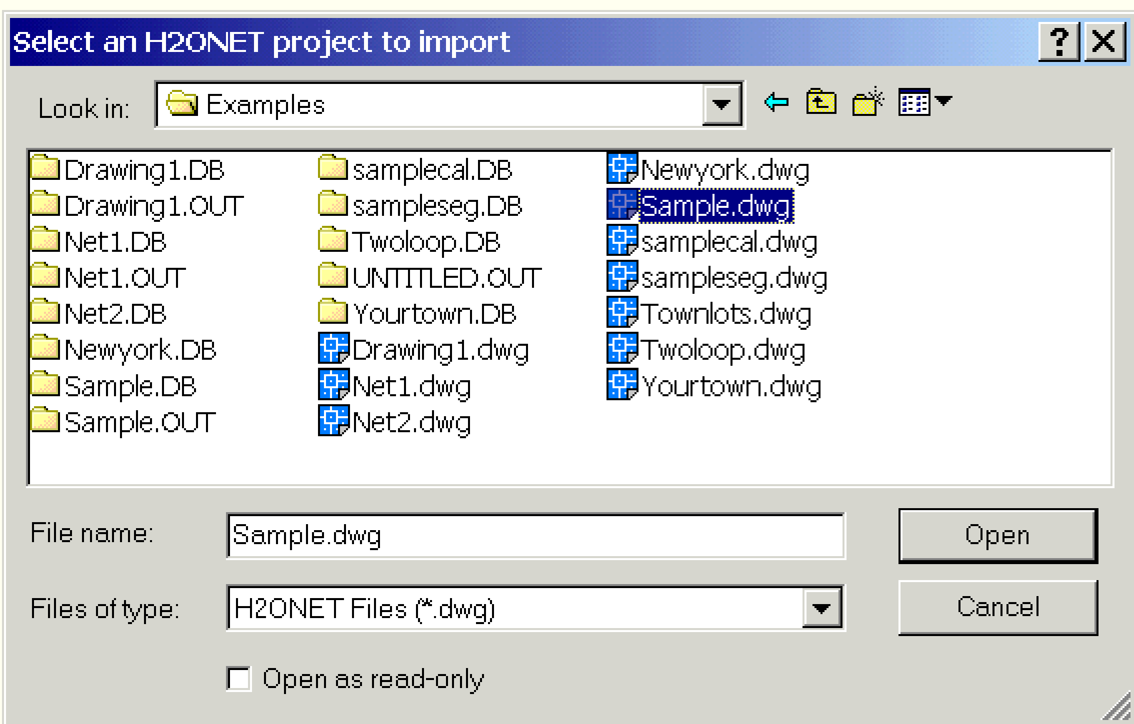
Import H2ONET

Use this to import an existing H2ONET Project into InfoWater. InfoWater will create an InfoWater database and a drawing from the H2ONET project.

Be sure to set your drawing limits before you import your H2ONET project.

Click on the InfoWater Button 

, browse to the Project menu and select the Import H2ONET option. Navigate down to the appropriate H2ONET file (.DWG extension) and click on it to select it. InfoWater will first create the InfoWater project database and subsequently create an InfoWater drawing from the database. After a successful import, save the newly created InfoWater project into a location on your hard-drive by using the **Save As** command under the **File** menu.



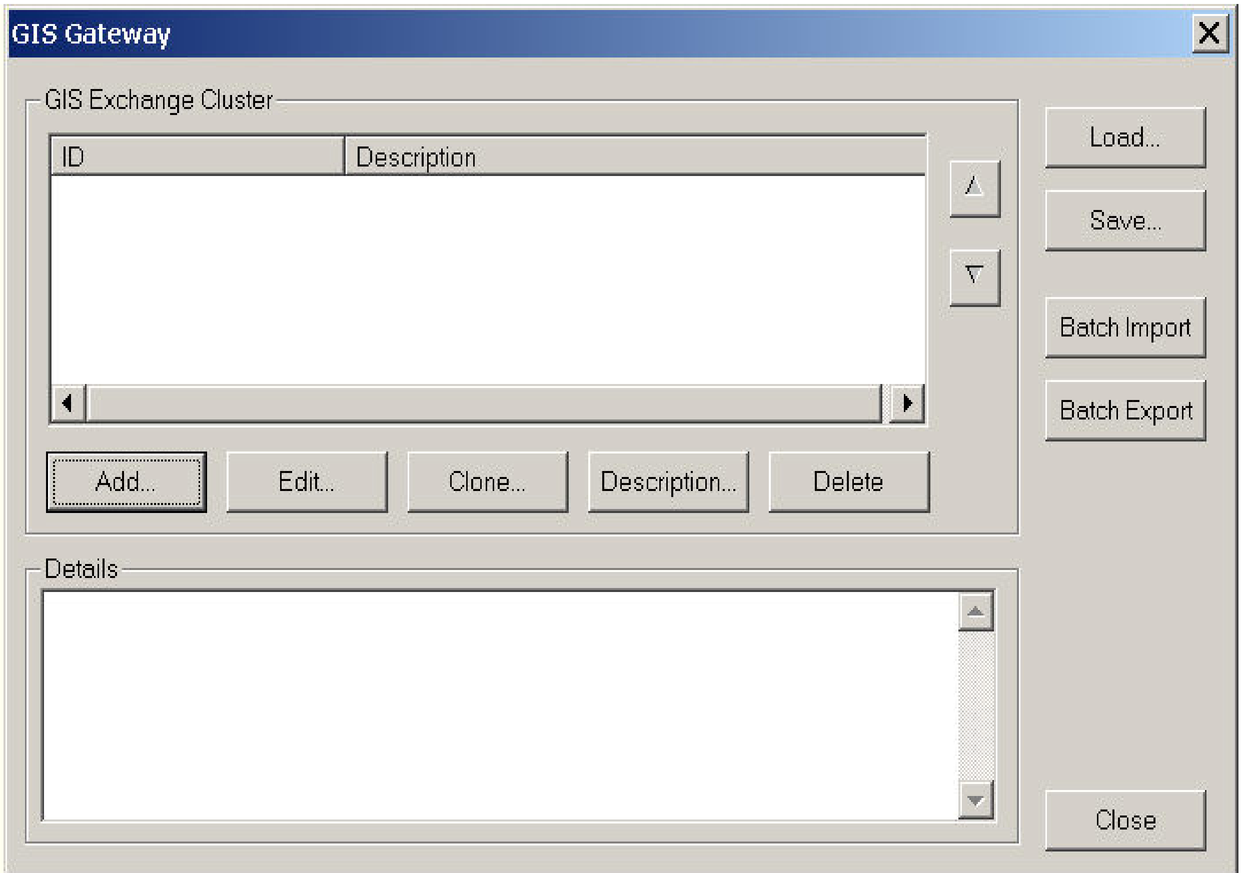
Note - All versions including and subsequent to **H2ONET Analyzer Version 3.1** can be imported into InfoWater using the process discussed above.

GIS Gateway

GIS Gateway provides a means to create data exchange parameters that allow you to quickly load and save data between InfoWater and other GIS formats (including Geodatabases and personal Geodatabases).

To run GIS Gateway, from the **InfoWater Control Center** toolbar **-> InfoWater** button **-> Exchange** menu, select the **GIS Gateway** command. Alternatively, you may launch the GIS Gateway dialog box by clicking on the GIS Gateway icon  on your **InfoWater Control Center**. You will now see the GIS Gateway dialog box and be ready to manage your GIS Exchange Clusters and perform desired data exchange with various external GIS data sources. A GIS Exchange Cluster defines a set of data exchange parameters which allows you to quickly load and save data between InfoWater and other GIS formats. Click here to learn more about **Working with ESRI Geodatabases**.

Click on any section below to learn more:



Other Related Topics - Working with ESRI Geodatabases, Geodatabase Theory, GIS Cluster - Spatial Join, GIS Exchange Cluster, GIS Exchange Cluster - Tabular Join, GIS Field Mapping

---

Simulation Set Methodology

Simulation Sets provide the following different options to customize each of your scenarios:

- **Report Option Set** – Standard reporting options associated with a scenario.
- **Simulation Option Set** – Simulation options associated with a scenario, including hydraulic and water quality modeling options, whether or not logical controls will be used during the next simulation run, etc.
- **Time Option Set** – Simulation duration and time steps associated with a scenario.

Each scenario may be customized to have different simulation options such as the Type of simulation, the duration of simulation and other options pertaining to running of a simulation including the simulation report.

Using Simulation Sets in InfoWater

Simulation Sets in InfoWater may be used for the following:

- To create scenarios that have customized report options. For instance one scenario many have a Hydraulic Reporting status as Full while another one might have it set as None.
- Similarly the customization of Simulation Options allow for defining differing analysis types for each of the scenarios. For instance one scenario may have a Chemical Water Quality analysis defined while another scenario may be customized for Water Age analysis. All the different Simulation options may be customized to suit the simulation needs of each scenario.

- Scenarios may also be customized to run Steady state analysis in one case and Extended Period Simulations in the other. Additionally the other Simulation Time options may also be customized.

Methodology

Do the following to customize the Simulation Sets:

- Launch the Scenario Manager from the **InfoWater Control Center -> InfoWater** button **-> Scenario -> Scenario Manager** command or use the **InfoWater Control Center** Scenario Manager icon .

- Choose the Scenario that you want to customize by clicking on it. Make sure that this scenario or its children are not presently active. Click here for more information about the Parent-Child relationship for scenarios.

- Click on the **General** tab on the right window of the Scenario Manager.

- Check the option that you want to customize from among the **Report Options**, **Simulation Options** and **Time Setting**. You may select all of them if you like.

- Click on the **Browse** icon next to the Option that you want to customize. This launches the appropriate options dialog box.

- In the now launched **Report Options**, **Simulation Options** or the **Simulation Time** dialog box, click on the **New** icon to create a new Options set. Specify an ID and Description for the Option set and customize the different parameters.

- Click on the button at the top of the Options dialog box to return to the **Scenario Manager** dialog box. You will notice that the newly customized Options Set has been associated with the Scenario.

- Once the Simulation Sets association process has been completed, click on the 

 button at the top of your Scenario Manager dialog box to save and exit out of your Scenario manager dialog box.

- Your Scenario can now be activated through the **Activate Scenario** select box of your **InfoWater Control Center.**

Note : Active Scenarios or their Parents cannot be customized. To customize the Facility sets for active scenarios you would first need to de-activate the scenario i.e., have another Scenario active.

Other Related Topics - Active Scenario**,** Compare Scenarios**,** Compare Scenario Methodology**,** Comparison Report**,** Data Sets Methodology**,** Dataset Manager**,** Facility Sets Methodology**,** Scenario Manager Data Sets**,** Scenario Manager Facility Sets**,** Scenario Manager General**,** Scenario Manager Main Dialog Box**,** Scenario Methodology

---

Facility Sets

The facility set defines those network components (pipes, pumps, nodes, etc.) that will be considered during the InfoWater simulation run.

The facility set may include the entire network model or a subset of network components. Only facilities in the active facility set are displayed in InfoWater. Facilities that are not active (i.e., those not in the current facility set) are removed from the map display.

Using Facility Sets in InfoWater

Facilities in InfoWater may be used for the following:

- Use the facility set to define the active network, i.e., the components in your InfoWater project that you want to include in your simulation run.
- Use facility sets to model "what-if" scenarios for operator training, for better understanding your system, for conducting worst case scenario experimentation, for future development planning etc.

Methodology

To associate a facility set with your scenario do the following:

- Launch the Scenario Manager from the **InfoWater Control Center -> InfoWater** button **-> Scenario -> Scenario Manager** command or use the **InfoWater Control Center** Scenario Manager icon .
- Choose the Scenario that you want to create a facility set for by clicking on it. Make sure that this scenario or its children are not presently active. Click here for more information about the Parent-Child relationship for scenarios.
- Click on the **Facility** tab on the right window of the Scenario Manager.
- Choose from among the different facility creation options on the Scenario manager dialog box.
- Once the facility assignment creation process is completed, click on the OK button at the top of your Scenario Manager dialog box to save and exit out of your Scenario manager dialog box.

Note : Active Scenarios or their Parents cannot be customized. To customize the Facility sets for active scenarios you would first need to de-activate the scenario i.e., have another Scenario active.

Other Related Topics - Active Scenario, Compare Scenarios, Compare Scenario Methodology, Comparison Report, Data Sets Methodology, Dataset Manager, General Options Methodology, Scenario Manager Data Sets, Scenario Manager Facility Sets, Scenario Manager General, Scenario Manager Main Dialog Box, Scenario Methodology

Data Sets

A data set is one of three components that comprise a scenario. Data sets provide the capability to take a one-time “snapshot” of data (database information) in the active model and store that information separate from the network itself.

In essence, InfoWater creates separate database tables for each data set created, allowing the user to manipulate the database characteristics of the data set, separate from the "Base" data set. Once new data is stored in a data set, it may be reloaded back into the active model (via a scenario) at any time.

To learn more about data sets, click on the following subjects.

How Do I...

- Create a Dataset?
- Activate a Dataset?
- What Happens when a Dataset is Activated?

Using Datasets in InfoWater

Datasets in InfoWater may be used for the following:

- Data sets store all the modeling information relating to the different Scenarios in InfoWater. For instance the **Demand Set** contains all the Demand informations and so on.
- Different scenarios may have different data sets associated with them. For instance **Scenario 1** may contain **Demand set 1** and **scenario 2** may contain **Demand Set 2** implying that two

scenarios may contain completely different demand values for modeling different situations.

- Similarly each Scenario may be customized to contain different Control Sets, Curve Sets, Demand Set, Energy Sets, Fire Flow Sets, Logic Sets, Pattern Sets, Pipe Sets, Pump Sets, Quality Sets, Reservoir Sets, SCADA Sets, Tank Sets, Valve Sets.

- Customizing the Scenarios allow for various Operator studies, modeling "What-if" scenarios, studying the model under different demand conditions, control sets (initial status and simple controls), logic controls, different pump, pipe, valve, junction, tank and reservoir configurations.

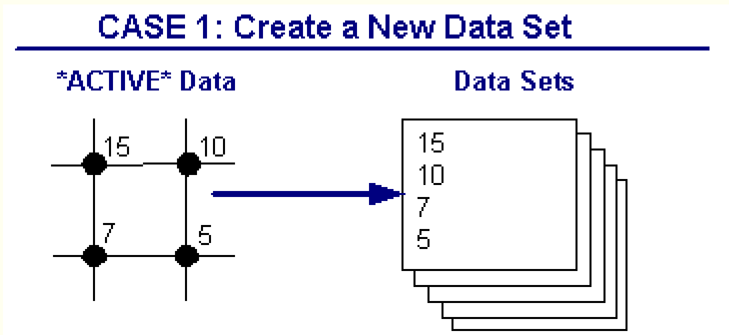
- Scenarios also allow you to create and run different fire-flow simulations, energy management analysis and water quality simulations.

Methodology

Steps to Create a Data Set

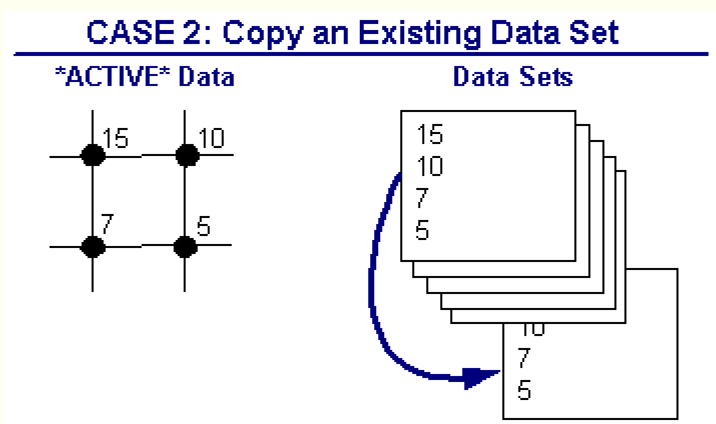
Data sets can be created in one of two fashions:

1. Create a new data set from the current *ACTIVE* or currently loaded data set into a new data set.



   *(Note: For all new data sets (with the exception of nodal demands), creating a new set will copy the original database table of the *ACTIVE* set to the new data set database table. When a new Junction Set is created (not cloned), the database field for system demands will be empty.)*

2. Clone data from a previously-defined data set into a new data set.

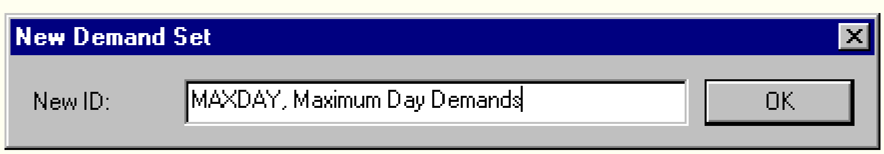


Example Data Set Creation

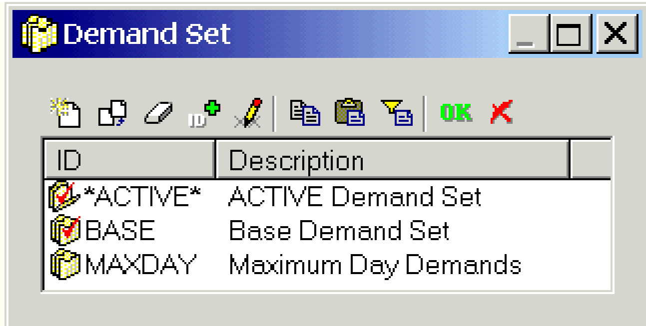
In this example, steps for the creation of a cloned demand set are provided. For a more detailed tutorial, please see Chapter 3 of the User's Guide.

1. From the **InfoWater Control Center -> InfoWater** button **-> Scenario** menu, select **Demand Set**. Once selected, the following dialog box will appear:

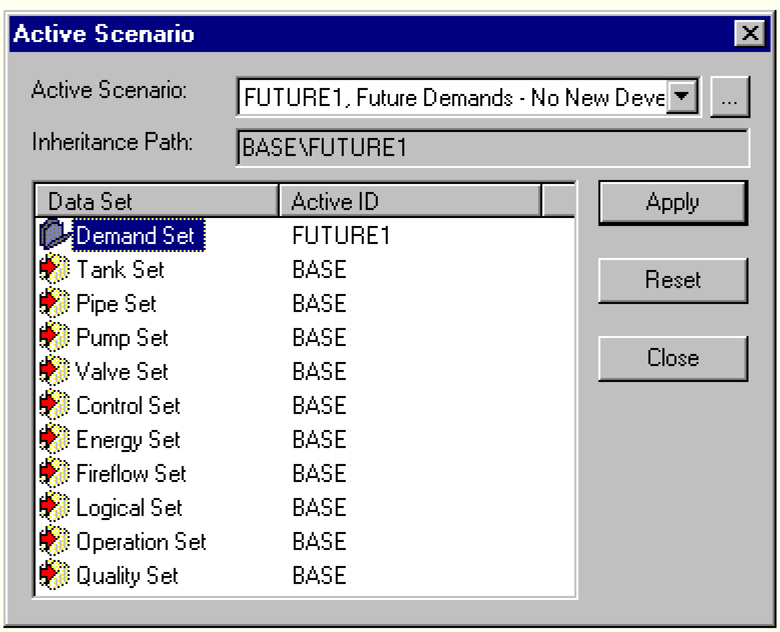
2. The *ACTIVE* demand set represents the Base demand set (in this case). Select the Clone icon  to replicate the existing *ACTIVE* demand set. (By selecting clone, the user is requesting that all information in the *Active* database be copied to the new demand set database - including system demands.) Once selected, the following dialog box will appear:



3. Enter in the unique ID for the demand set (in this case, MAXDAY) and then provide a description for future reference. Once entered, select OK and the following dialog box will appear:



4. Notice that there is no red check mark next to the new MAXDAY demand set. The reason is because the red check mark indicates whether a data set is currently active or not. To edit the data related to the MAXDAY demand set it must first become associated with a scenario and made *active*. (Once a data set becomes active, any edits made to a data set will be saved and stored ONLY in the *active* data set.) The next step is to click OK to exit the demand set dialog box and associate your new data set with a new (or existing) scenario. Go to the **InfoWater Control Center -> InfoWater** button **-> Scenario** menu and select **Active Scenario**. Once selected, the following dialog box will appear:



5. The currently *active* scenario will appear in the Active Scenario drop down box (in the example above, FUTURE1). To create a new scenario, select the [...] icon next to the Active Scenario drop down box to see the Scenario Manager dialog box. Highlight the BASE scenario and click on the "New Child" icon. Specify an ID and description for the new scenario and click the OK button. The user is now able to edit any data set for the new scenario by double-clicking on the Category Name (in our case, the demand set) and selecting the MAXDAY demand set. Once the MAXDAY demand set is associated with your new scenario, select the Activate icon to activate this scenario.

6. At this point, any modifications made to the junction demands will be saved to the MAXDAY demand set (not the original demand set). So, in essence, by creating this new scenario that now contains the new MAXDAY demand set, we have created a new demand database table that is only related to this demand set (and to the new scenario). At any time the user can go back to the Scenario Manager and specify a different demand set (or any other data set). To make it active and edit the database related to the active data set, use the Active Scenario command. Any changes made will be saved and are ready to be "recalled" by the user once any specific scenario is made active.

Activating Another Data Set

There are two methods for loading new data or option sets into the current InfoWater project:

- Associate a data set with a custom scenario and then activate that scenario – The primary method for activating data sets is by associating them with one or more custom scenarios and then activating one of your scenarios. Definition of custom scenarios and related data sets is accomplished using the Scenario Manager command.

- Load a data set at any time – To change a data set for the currently active scenario, from the **InfoWater Control Center -> InfoWater** button **-> Scenario** menu, choose the **Edit Active Scenario** command. You may load one or more data sets into the active scenario. By doing so, the current modeling information in the open InfoWater project will be copied to the active data sets before switching to the new data sets you selected. The contents of the newly-selected data sets are then copied into the open InfoWater project and are immediately available.

The second option is most commonly used when the user has not developed multiple scenarios - yet still wishes to interchange data sets for modeling purposes (trial and error before scenarios are created).

Note: The user cannot switch data sets for the BASE (default) scenario. The user may only switch data sets for custom-developed scenarios.

What Happens During a Data Set Activation

InfoWater activates different data sets in different fashions. The following describes how each data set type is activated and data from those data sets loaded into the *ACTIVE* scenario:

Pipe, Pump, Valve, Tank and Reservoir Sets

When these data sets are activated and where a match exists between a record in the data set and a network component in the activated facility set (the network you see on the map display), the data from the data set overwrite any data currently loaded for those network components. Where there is no match between records in the data set and activated network components, those components retain their current data values.

Control, Energy, Fireflow, Logic, Pattern, Curve and Quality Sets

When these data sets are activated, InfoWater first clears all data (related to these data sets) from network components in the activated facility set (the network you see on the map display). Then, where a match exists between a record in the data set and a network component in the activated facility set, the data from the data set are assigned to those network components. Where there is no match between records in the data set and activated network components, those components remain with zero or null data values.

Demand Sets

By default, demand set activation follows the same rules as Pipe, Pump, Valve, Reservoir and Tank Sets above, where data in the *ACTIVE* data sets are retained and only overwritten if there is a match between records in the data set and activate network component facility set. However, you may specify that demand set activation follow a similar procedure as Control, Energy, Fireflow, Logic, Pattern, Curve and Quality Sets, where data in the *ACTIVE* data sets are first cleared before loading data in the data sets.

The setting of the Auto-Demand Reset preference controls the demand set loading procedure. When ON (checked), demand sets are loaded similar to Pipe, Pump, Valve, Reservoir and Tank Sets. When OFF, demand sets are loaded in a fashion similar to all other data sets.

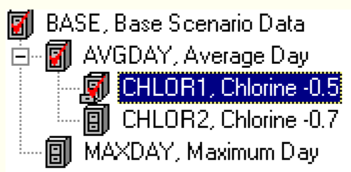
Other Related Topics - Active Scenario, Compare Scenarios, Compare Scenario Methodology, Comparison Report, Dataset Manager, Facility Sets Methodology, General Options Methodology, Scenario Manager Data Sets, Scenario Manager Facility Sets, Scenario Manager General, Scenario Manager Main Dialog Box, Scenario Methodology

---

Scenario ID and Description and Inheritance Tree

The Different Scenario IDs, Descriptions and the Inheritance tree is displayed here.

The relationship between a parent scenario and its child is defined as inheritance. InfoWater shows this relationship in the form of a directory tree. All scenarios are children of the *BASE* scenario until some element of the child is made unique from the parent (such uniqueness comes in the form of data sets, facility sets, or option sets). In the example below, CHLOR1 is a child of AVGDAY and AVGDAY is a child of BASE. When a change is made to a parent, unless some facet of a child is unique, it will inherit the change through inheritance.



For more information on Inheritance click here.

Inheritance

Inheritance refers to the relationship between a parent scenario and one or more of its children. Rather than each scenario existing independently of other scenarios, a scenario may inherit one or more of its properties from a parent scenario. With this capability, there is no need to enter redundant information to numerous scenarios that share the same data. Instead, you simply develop a master scenario (referred to as the parent) and develop one or more scenarios whose properties are dependent on the parent scenario.

For example, supposing that a child has a demand set that is exactly the same as the parent. When

a change is made in the parent demand set the change will also be reflected in the child. To

make the child independent with respect to system demands, create a new demand set specifically for the child and assign the new demand set to the child scenario. In

this case, it will ensure that the child is independent of the parent with respect to its demand set.

Explained further, once a child is created from a parent, the relationship is dynamic, meaning that when some piece of data is changed in the parent (like a control set), the child reflects that change as well. However, where a property is explicitly changed in a child scenario, the inheritance relationship for that property is broken and the change is reflected only in the child.

Compare Scenario Methodology

InfoWater's Compare Scenario command allows the user to monitor the differences between any two scenarios in a model.

Methodology

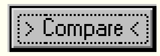
Once the Compare Data Scenarios dialog box appears, you will see that the Active scenario is selected. You can compare the Active scenario and another scenario in a model. Alternatively, you can select another scenario other than the Active scenario to compare to another scenario.

To compare two scenarios, perform the following:

- Select the first scenario you wish to compare from the Scenario #1 drop down list.
- From the Scenario #2 drop down list, select the scenario you wish to compare to the first scenario.
- If you wish to save the location of any elements that have data flagged by the Compare Scenario feature, you can save selection sets to refer to at a later time (i.e. from the Domain Manager). For information on creating a selection set, refer to selection sets.
- You can select an existing Selection Set to use, or you can create a new selection set. To save the Selection Sets, If you wish to save a Selection Set based on the elements that are in Scenario #1, but not in Scenario #2, check the Save elements that exist in Scenario #1 only option.
- Alternately check the Save elements that exist in Scenario #2 only option to save a Selection Set based on the elements that are in Scenario #2, but not in Scenario #1.

- If you wish to save a Selection Set based on elements that differ between the two scenarios, check the Save elements that have different values option.

- To alter the colors selected for each of the Compare Scenario outputs, click on the color swatch you wish to change, and a color pick dialog will appear for you to select a new color. Each color that is selected in this dialog will be reflected in the Data Scenario Comparison Report legend.

- To conduct a comparison of scenarios, press the 

button. Click here to earn more about the Scenarios Comparison report.

Note - you must have different selection sets selected for each comparison result that is to be reported. If you wish to reset an existing Selection Set, ensure that the **Reset SS** option is checked.

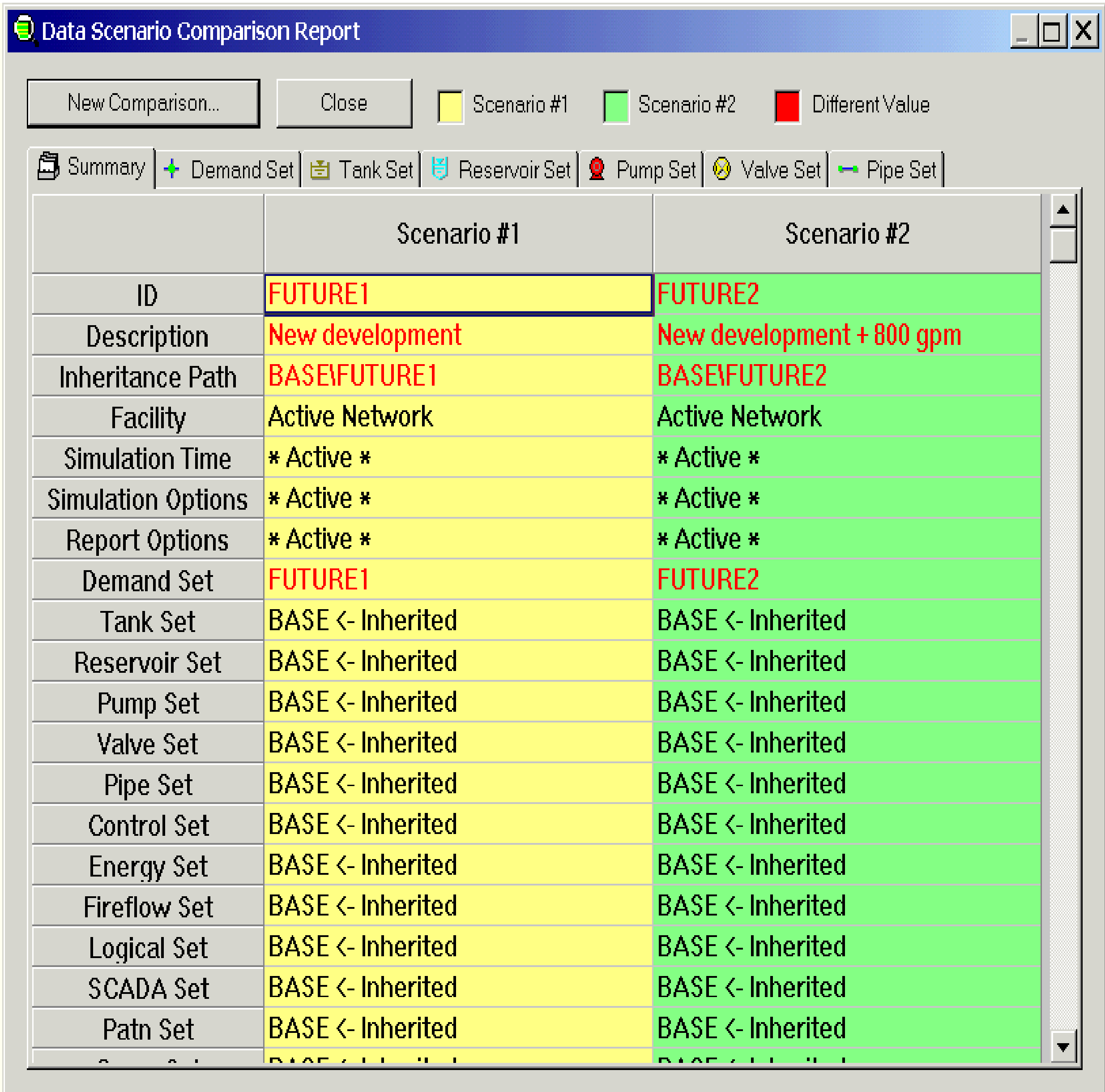
Other Related Topics - Active Scenario**,** Compare Scenarios**,** Comparison Report**,** Data Sets Methodology**,** Dataset Manager**,** Facility Sets Methodology**,** General Options Methodology**,** Scenario Manager Data Sets**,** Scenario Manager Facility Sets**,** Scenario Manager General**,** Scenario Manager Main Dialog Box**,** Scenario Methodology

---

Comparison Report

Compare Scenarios presents the results in a format that allows you to readily compare each scenario’s data. When the comparison analysis is completed, you will be presented with a display similar to the Graph Reports. This provides users with a summary of all the data associated with the scenarios. The following is what each of the summaries describe:

Click on any portion below to learn more.



Other Related Topics - Active Scenario, Compare Scenarios, Compare Scenario Methodology, Data Sets Methodology, Dataset Manager, Facility Sets Methodology, General Options Methodology, Scenario Manager Data Sets, Scenario Manager Facility Sets, Scenario Manager General, Scenario Manager Main Dialog Box, Scenario Methodology

Facility Methodology

The Facility Manager is used to create and maintain the active facility set. The active facility set defines the network components in a current model that will be considered during the next simulation run(s). Facility sets can also be associated with a scenario via the Scenario Manager. To assign a facility set to a scenario, click here.

At any point in time, only the active facility set is displayed on the screen. Inactive components are hidden.

How Do I...

- Create an Active Facility?
- Clear a Facility Set?

Using Facilities in InfoWater

Facilities in InfoWater may be used for the following:

- Choose the elements in your model that you want to activate for running your model. Only these elements will be used while conducting the analysis.

Methodology

Create an Active Facility Set

To create a facility set, do the following:

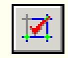
- Launch the **InfoWater Control Center** from the View menu **->** **Toolbars** command.
- From the **InfoWater Control center -> InfoWater** button -> **Tools** menu, select the **Facility Manager** option.
- At the Facility Manager dialog box, decide if the desired active facility set to be created should be done from some sort of query or graphical selection
- Click on the **Add** button to add the facilities to the active facility set.
- At this point, only those elements activated will retain their color, all others will turn grey or turn off (depending on the options specified under **InfoWater Control center -> InfoWater** button **-> Tools -> Project Preferences -> Display Settings** tab). If a model were run at this point, only those elements selected would be modeled (as long as the scenario facility option for the currently *active* scenario under the Scenario Manager was set to "Active Network" - click here to learn more.)

Clear a Facility Set

To clear a Facility, do the following:

- Launch the **InfoWater Control Center** from the **View** menu **->** **Toolbars** command.
- From the **InfoWater Control center -> InfoWater** button -> **Tools** menu, select the **Facility Manager** option.

- With the dialog box open, click on the **Activate All** button. Doing this will activate all elements and return the elements to their normal color.

- Alternatively you may also use the **Activate All** button  from your **InfoWater Edit Network** toolbar to activate all the elements and return them to their normal color.

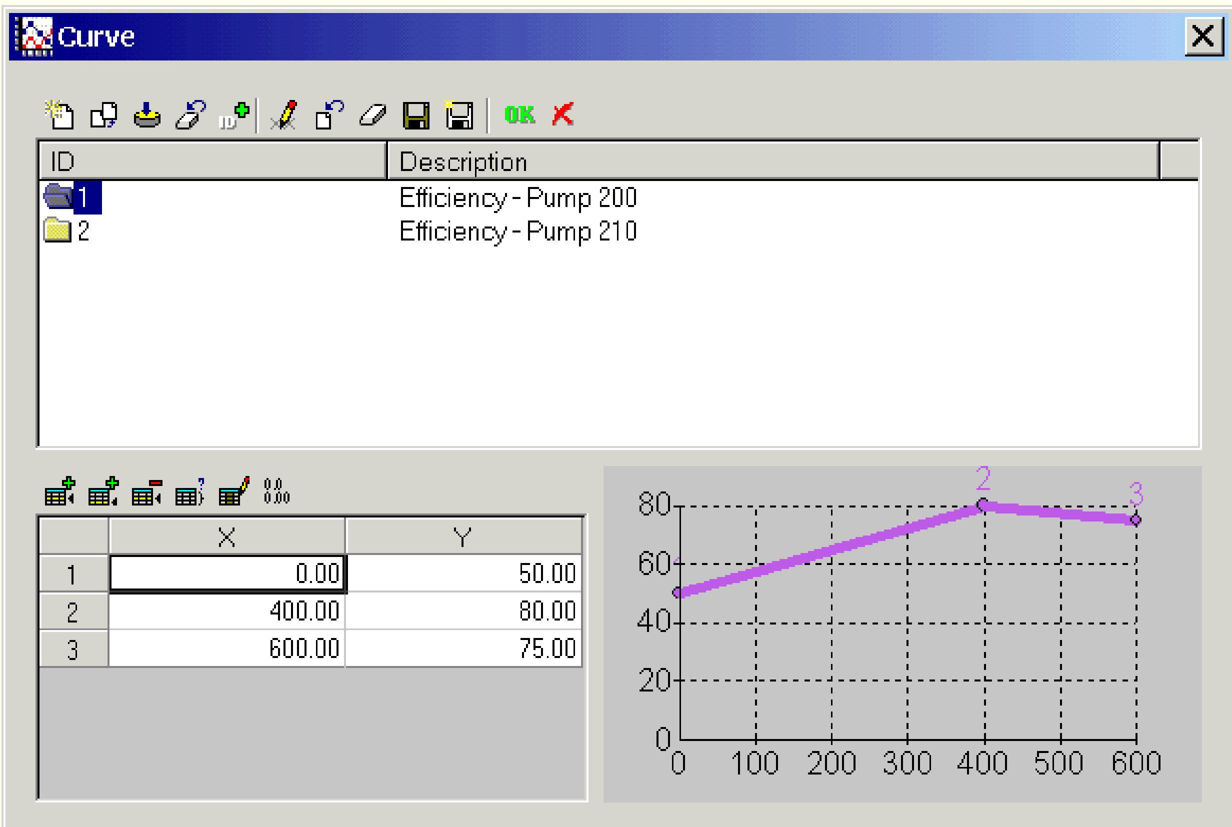
Other Related Topics - Domain Manager**,** Domain Methodology**,** Facility vs Domain**,** Facility Manager

---

Curve Dialog Box

The Curve Dialog Box enables the user to create, edit, clone, and delete curves as necessary. Click here to learn more above the curve creation process, their applicability and use in InfoWater.

Click on any portion below to learn more.

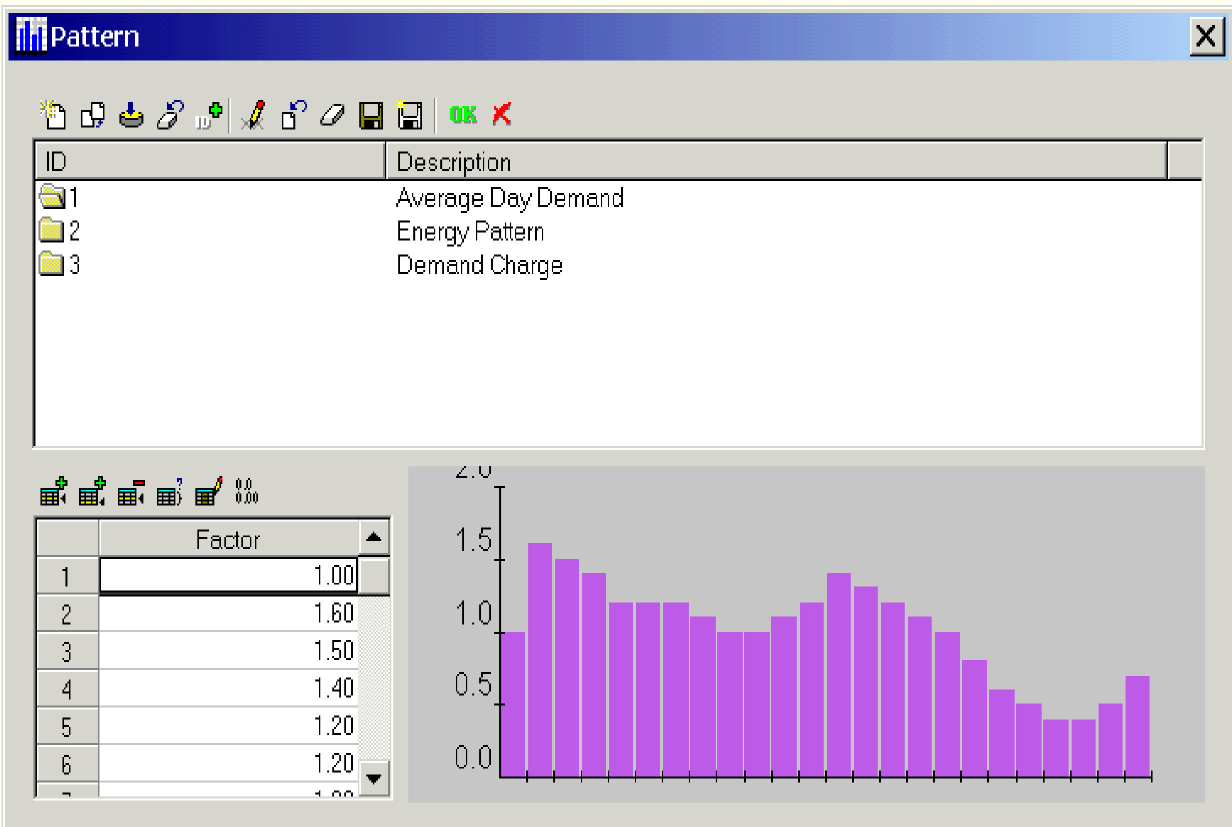


Other Related Topics - Curves, Curves and Patterns, Pattern Dialog Box, Patterns

Pattern Dialog Box

The Pattern Dialog Box enables the user to create, edit, clone, and delete patterns as necessary. Click here to learn more above the pattern creation process, their applicability.

Click on any portion to learn more.



Other Related Topics - Curve Dialog Box, Curves, Curves and Patterns, Patterns.

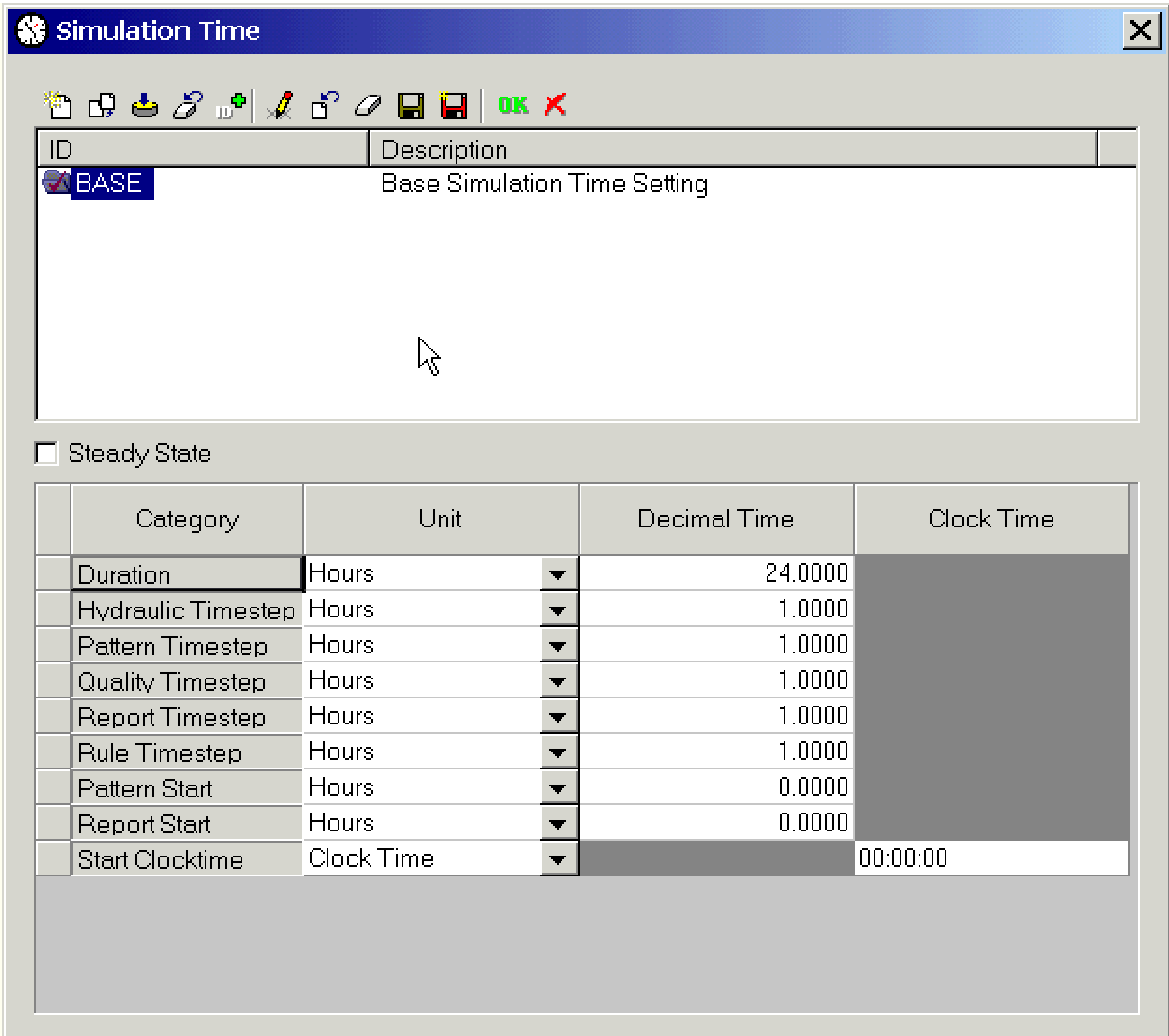
Simulation Time

The Simulation Time feature of InfoWater allows the user to change the time-step at which an extended period simulation is to be analyzed. If the user selects the Steady State check box, then only a steady state model will be analyzed. A simulation time option can be set up for each unique scenario within InfoWater.

An extended period simulation (EPS) is any modeling run that evaluates system hydraulics for a duration greater than a single hydraulic timestep (steady state). The user is provided many tools in InfoWater that allow customization of an EPS simulation.

Many features of InfoWater including SCADA, Energy Management, Water Quality, and Logic Rules are dependent upon a model being analyzed over a series of timesteps (Extended Period Simulation).

Click on any portion below to learn more.

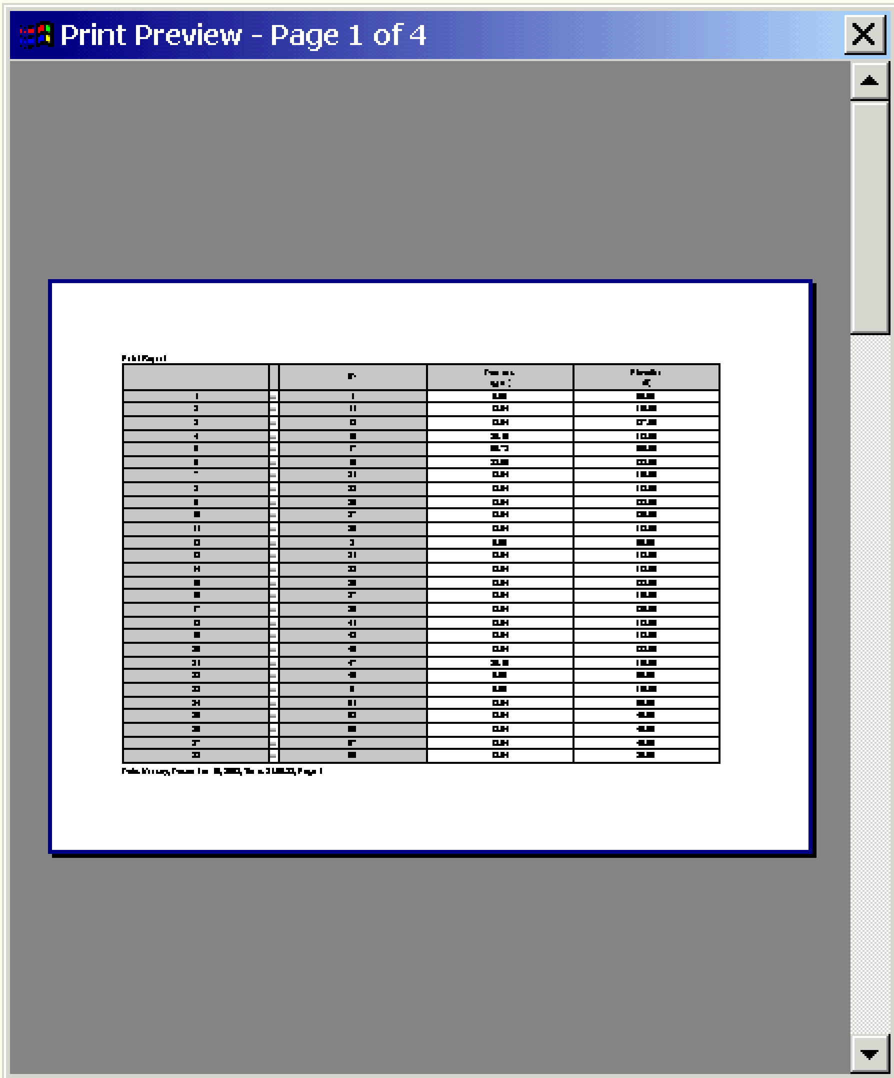


Other Related Topics - Running a Model, Simulation Options Dialog Box, Simulation Options - Demand, Simulation options - Energy, Simulation Options -Quality, Simulation Options Methodology, Simulation Report, Simulation Report Customization Tab, Simulation Report Methodology, Simulation Time Methodology

---

Print Preview

The Print Preview command allows the user to view and make changes and format the InfoWater display prior to plotting.



Graph Settings Methodology

InfoWater graphs allow for may customization options. All the graph modification options on top of the graph in your Output Report Manager provide you with the various tools to customize your graph. However once your graph has been modified you can create a Graph Template file which contains your graph settings. You may launch this file any time to create different graphs with the same exact settings.

Methodology

To create a graph settings template, do the following:

- Launch your **Output Report Manager** from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu **-> Output Report Manager** command. Alternatively you could use the **Output Report Manager**  icon on the **InfoWater Control Center** to launch the **Output Report Manager**.

- Click on the **New** icon to launch the **Output Report & Graph** dialog box. Choose the Output Source from the **Available Output Source** section. Output Sources will only be available after you run a successful model simulation.

- Click on the **Graph** tab and select the graph type and click on the **Open** to select the element that you want to graph. Once your graph is displayed modify your graph using the different graph modification options.

- Click on the **System** icon  to launch the System dialog box.

- Under the **Graph Template** section of the **System** dialog box, check the **Save Data** option 

.

- Click on the **Browse** button to specify the name and location of your Graph Settings Template file (gsp extension).
- Finally click on the **Save** button to save your Graph Settings Template file.
- Once your file has been saved you may launch it at any time by clicking on the **Browse** button and locating the file. Thereafter click on the **Load** button to load your Graph Settings File.
- Close out of your Settings dialog box by clicking on **Cancel** and verify that your presently active graph reflects the settings on your Graph Settings Template file.

Other Related Topics - Print Graph, Print Grid, Printing and Presentation, System Button, Print Preview

---

Building Distribution System Models

InfoWater is developed to take full advantage of the capabilities of ArcGIS and relational database management systems (RDBMS) technology. By integrating and utilizing proven and efficient tools for graphical data entry, maintenance and display, InfoWater provides the user with an efficient easy-to-use vehicle for model construction.

In InfoWater, a water distribution system model consists of two major components:

- **Network schematic** - The network schematic provides a critical role as a model input, providing information on the connectivity, length, and shape of network components and it also plays an important role in the display and analysis of model results, acting as the vehicle for results presentation.
- **Modeling attributes** - Information associated with network elements act as the primary input to the hydraulic and water quality network simulators. This information includes both a set of required attributes describing critical components of system operations and a set of user-defined attributes for integration with other infrastructure management applications.

InfoWater provides the following tools to build and maintain water distribution system models:

Network Components

InfoWater can accommodate a full library of hydraulic components and which can be organized in any topological configuration. Network components can include pipes, single and separate inlet/outlet storage facilities, cylindrical and variable-area tanks, variable-head reservoirs, hydropneumatic tanks, pressure reducing valves, pressure sustaining valves, pressure relief valves, pressure breaker valves, flow control valves, bi-directional flow totalizers (flow meters), throttle control valves, motorized throttled valves, vacuum breaker valves, float valves, check (one way) valves, user-defined (customizable) general purpose valves (any headloss vs. flow relationship), source pumps, booster pumps, variable speed pumps, turbines, booster disinfection stations, and minor loss devices. Pump characteristic curves can be described by multiple data points (head-flow) input.

Importing Existing Networks

The InfoWater network map is an ArcMAP drawing file. An existing map of a water distribution system can be imported from one of many CAD (e.g., DXF, DWG) or GIS (e.g., Shapefile, Generate, MID/MIF, CSV) formats. A highly efficient polyline conversion module automatically converts CAD drawings with curvilinear line representations to an InfoWater working network map. Additionally, existing models can be imported into InfoWater from other water network modeling software (e.g., EPANET).

Data Entry

InfoWater uses extensions to standard ArcMap digitizing tools to create network components including pipes, pumps, valves, junctions, reservoirs and tanks and to simultaneously add the modeling attributes associated with these components. Pipes can be of any shape and their lengths are automatically computed to the scaled drawing length. Other drawings can be easily incorporated into the graphical environment to assist with the data capture process. By digitizing directly on a map display showing streets, parcels, and even other utilities, data can be entered with high positional accuracy.

InfoWater’s unique open-architecture environment enables the user to build the model using other software packages. These applications can be used to populate the database for the various network elements. This data can then be retrieved into InfoWater for further manipulation and customization.

Data Maintenance

The user can move, add, insert, merge, split, or delete network component graphically at any time. Additionally, the user may modify any of the attributes associated with existing network components. Updates to the network map and database records are immediate and automatic.

Database Customization

Although InfoWater requires a prescribed database structure to support the network simulator, the open-architecture environment enables the user to create and manipulate any number of additional attributes. The database tables for pipes, pumps, valves, junctions, tanks and reservoirs can be customized to better organize and manage system data and to support other infrastructure management applications.

Creating Hydraulic and Water Quality Simulations

InfoWater provides unparalleled network analysis and simulation capabilities. With InfoWater, the entire water distribution system or any selected sub-system may be analyzed under steady-state and extended period simulations. InfoWater tracks the flow and velocity of water in each pipe, the pressure and head at each junction node, the height of water in each tank, the discharge flow and pressure for each pump, and the movement and fate (growth or decay) of water quality constituents as they travel through the network during a dynamic simulation period. Specifically, the following types of water quality analyses can be performed:

- **Chemical concentration** - InfoWater computes the concentration of the specified chemical throughout the network during an extended period simulation.
- **Water age** - InfoWater models the changes in age of water over time throughout the network. Under constant (static) hydraulic conditions, the age of water at any location in the network is the time of travel to that location.
- **Source tracing** - InfoWater computes over time the fraction (percentage) of water originating from a specified source.

The dynamic water quality model accurately simulates both conservative (e.g., fluoride) and reactive (e.g., chlorine, THM) species and is equipped to model such phenomena as reactions within the bulk flow, at the pipe wall, in storage tanks, and mass transport between the bulk flow and pipe wall. A global kinetic rate coefficient can be assigned for the entire network or user-specified values can be assigned to selected network components. The pipe wall reaction rate constant can be also expressed as a function of pipe roughness for improved calibration.

Hydraulic Modeling Capabilities

InfoWater provides unparalleled network analysis and simulation capabilities for performing a wide range of essential modeling tasks. It utilizes a full-featured, state-of-the-art hydraulic computational engine that includes the following capabilities:

- Places no limit on the size of the network and number of components that can be analyzed (unlimited pipe version)
- Supports both English and metric (Standard International) units
- Analyzes steady-state and extended period simulations
- Computes friction headloss using the Hazen-Williams, Darcy-Weisbach, or Chezy-Manning formulas
- Includes minor (local) head losses for bends, meters, fittings, etc.
- Models constant or variable speed pumps
- Models turbines and flow totalizers (meters)
- Calculates Net Positive Suction Head requirements (available NPSH, required NPSH, and cavitation index) to ensure cavitation free operation
- Determines system head curves and fire hydrant rating curves
- Calculates available and design fire flows at minimum residual pressures
- Computes pumping energy and cost based on variable electricity and demand charges
- Accommodates various types of static and control valves including shutoff, check, float, pressure regulating, pressure sustaining, pressure beaker, motorized throttle control, general purpose, and flow control valves
- Allows single and multiple inlet/outlet storage tanks to have any shape (i.e., diameter can vary with height)
- Models constant and variable head reservoirs

- considers multiple demand categories at nodes, each with its own pattern of time variation
- Models pressure-dependent demand issuing from emitters (sprinkler heads)
- Simulates leakage in a pipe
- Specifies system operation based on both simple node pressure, tank level, pipe flow, or timer controls and on complex rule-based controls
- Provides SCADA interface with alarms.

Water Quality Modeling Capabilities

InfoWater provides fast, comprehensive and accurate dynamic water quality computations. It:

- Tracks the movement of a non-reactive tracer material (e.g., fluoride) through the network over time
- Models the movement and fate of a reactive material as it grows (e.g., a disinfection by-product) or decays (e.g., chlorine residual) with time
- Calculates the age of water throughout a network
- Computes the percent of flow from a given node reaching all other nodes over time
- Analyzes kinetic reactions both in the bulk flow and at the pipe wall
- Incorporates n-th order kinetics to model reactions in the bulk flow
- Uses zero or first order kinetics to model reactions at the pipe wall
- Accounts for mass transfer limitations when modeling pipe wall reactions

- Allows growth or decay reactions to proceed up to a limiting concentration
- Employs global reaction rate coefficients that can be modified on a pipe-by-pipe basis
- Permits wall reaction rate coefficients to be correlated to pipe roughness
- Allows for time-varying concentration or mass inputs at any location in the network
- Models storage tanks as being either complete mix, plug flow, or two-compartment reactors.

Multi-Level Inheritance Scenario Manager

InfoWater provides a comprehensive tree-type scenario manager with complete project inheritance. Each project "child" scenario inherits the information from its "parent" scenario. You can now maintain a single model of your water system and quickly develop and evaluate numerous modeling alternatives to support your capital improvement decisions. Change a pipe diameter or status, pump power, control or schedule, valve setting, nodal demand, tank level or size, chlorine dosage or any of these combinations, and the changes will cascade through the entire set of projects in an easy-to-use tree-like structure. You can easily switch between scenarios and compare modeling results instantly. You can even directly extract (cut) one or more individual pressure (or boundary) zones from the main simulation model as well as merge (paste) together any number of models for detailed analyses. You also have the ability to automatically generate specialized skeletal models (automated skeletonization) of your water system based on any specified set of parameters (e.g., all pipes with diameter greater than 12 inches).

Analyzing Model Results

InfoWater provides a set of customized mapping functions to greatly accelerate and simplify the presentation of modeling results. With the many data visualization tools available within InfoWater, the user can browse and analyze model results without the laborious and time-consuming process of working with formidable amounts of data and comprehending large volumes of tabular results.

Color Mapping

Color-code any network variable: pipe size, flow rate, velocity, headloss, hydraulic slope, node pressure, grade, demand, elevation, water age, percent source contribution, water quality concentration, and any other database attribute in InfoWater. Numerical ranges for color coding classifications and color selections are user-specified. Use this feature to visually differentiate among pipes of different material, diameter, and date of installation, and among numerous pressure zones by specifying a different color for all pipes in each of the separate zones.

Map Labelling/Annotation

InfoWater provides "what you see is what you get" annotation. You can annotate and view simultaneously modeling data and results for network attributes and for any hydraulic conditions directly on the network map. These dynamic labels automatically update as you switch between hydraulic time steps and scenarios. Annotation of the various element attributes can be directly customized and flow directional arrows can be displayed along with text information.

Graphing

Time-series graphs can be generated for network components and can be used to visualize how user-specified variables change throughout an extended period simulation. Variables such as pipe flow rates, velocities, headlosses, nodal pressures, hydraulic grades, water age, quality concentrations, pump operations, and others can be viewed on time-series graphs. Frequency distribution graphs for any variable can also be generated for any given time period depicting the fraction of nodes and pipes not exceeding specified values. Graphs can be fully customized by the user. The user can superimpose multiple graphs for comparison or display purposes, switch between 2D and 3D display (including several options for each type), change data markers, graph scales, axes, and titles. Observed field data can also be linked to a graph. This proves especially useful for model calibration.

Animation

InfoWater offers a very useful VCR-style animation feature that helps you step through an extended period simulation and instantly visualize and comprehend the dynamic performance of your system such as temporal variations in HGL profile, pump operation, and water quality propagation for the length of the simulation period.

Contouring

For any given simulation run, InfoWater allows you to efficiently generate accurate, smooth (T.I.N.) contours for any numerical database attributes including elevation, pressure, hydraulic grade line, demand, chemical concentration, water age, percent source contribution, fire flow, customer complaint, and other pertinent modeling parameters. Beautiful contours can be generated directly on ArcMap or in a separate graphical window and for the entire network or any user-selected portions of the network. Color grading and coding and numerical interval ranges as well as various useful graphical presentation options are user-specified.

Database Browsing

With InfoWater, the user can browse both model input data and analysis results in a tabular format. The data appears in a spreadsheet format on-screen, accessed directly from the project database tables. The user can customize the database display and may generate database reports using either standard or customized formats. You can also instantly locate network elements and view modeling results in a "point and click" mode for any time period of a dynamic simulation.

Database Queries

Sophisticated query tools let you build complex intelligent queries on any database and modeling attribute (both input and output data) to meet a single criterion or multiple criteria simultaneously. You can isolate those network components that possess a desired property or those that may indicate a specific problem. Such capabilities can greatly assist you in the decision making process for network asset inventory, rehabilitation requirements, and financial planning. Query results can be displayed graphically, in tabular format, printed, or exported to a standard spreadsheet or word processor.

Customized Reports

Comprehensive report generation tools are provided for simultaneous display of input and output variables. Report tables can be completely customized by selecting the variables to display, the order in which they appear along with the desired units, display precision, column width, and format, and the corresponding hydraulic time period. You can include the entire network, any portions or selected elements. Each table can be sorted according to any variable and filtered based on any search criteria. Maximum, minimum and average values for all variables are automatically calculated and reported for EPS runs.

Note - Both Contour and Annotation will work for any user selected DOMAIN.

Note - Color Coding will **NOT** work for a Domain. InfoWater will always color code the **ENTIRE** network.

Integration with Other Applications

Using Windows-standard *copy* and

paste tools, modeling results from InfoWater can be exchanged with other applications including presentation applications, spreadsheets, databases, mapping, and geographic information systems.

Exporting To Water Distribution System Models

InfoWater can write the contents of a network model created in InfoWater to EPANET or GIS formats (e.g., shapefiles) for use in other applications. This will prove useful for sharing your modeling efforts with your organization via enterprise or web (intranet or internet) applications.

Exporting Network Maps

Just as an existing network map can be

imported from another CAD or GIS application, a network map created with InfoWater can be easily exported to other CAD or GIS applications. Any application that can read an EPANET map file, AutoCAD drawing file (DWG), Digital Exchange Format (DXF) file, Generate files, MID/MIF files, or Shapefiles can accept network maps created in InfoWater.

Exporting Database Tables

InfoWater model inputs and outputs can

be exchanged with other applications in four fashions. First, InfoWater attribute data is stored in a dBASE format and is therefore immediately compatible with any other application that can read or write from or to that format. Second, selected database rows or columns can be *copied* to the Windows Clipboard and then *pasted* into other Windows-compatible applications. Third, InfoWater allows database tables to be exported (and imported) in a Windows comma delimited text (CSV) format. Additionally, data can be directly accessed or shared through complete DAO and ODBC

support.

Exporting Graphs

InfoWater graphs can be exported to other

applications via Windows *copy* and *paste* tools. After bringing a graph to the screen and optionally customizing the graph, the user can save the graph to a Windows Metafile (WMF) or Bitmap (BMP) format or export the graph display to word processors, presentation packages, or other Windows-based software applications.

Physical Components

InfoWater conceptualizes a water distribution network as a collection of pipes connected together at their endpoints, called nodes. The nodes represent junctions, tanks, reservoirs, pumps and valves (control valves).

Besides being the junction point between connecting pipes, nodes can serve as:

1. Points of water consumption (demand nodes)

2. Points of water input (source nodes)

3. Locations of tanks or reservoirs (storage nodes)

4. Locations of pumps or valves

How InfoWater models the hydraulic behavior of each of these physical components will be reviewed next. For the sake of discussion we will express all flow rates in cubic feet per second (cfs), although the program can also accept flow units in gallons per minute, million gallons per day (mgd), imperial million gallons per day (imgd), acre-feet per day (afd), cubic meters per hour (cmh), cubic meters per day (cmd), million liters per day (mld), liters per second (lps), or liters per minute (lpm).

Junctions

Junctions are points in the network where pipes join together and where water enters or leaves the network. They are also placed at locations where pipe characteristics change (e.g., diameter, roughness coefficient, material) and at location where pressure, head or water quality calculation is desired. The basic input data required for junctions are:

- Elevation above some reference (usually mean sea level)
- Water demand (rate of withdrawal from the network) and associated pattern
- Initial water quality.

The output results computed for junctions at all time periods of a simulation are:

- Hydraulic head (internal energy per unit weight of fluid)
- Pressure
- Water quality.

Junctions can also:

- Have their demand vary with time
- Have multiple categories of demands assigned to them
- Have negative demands indicating that water is entering the network
- Be water quality sources where constituents enter the network
- Contain emitters (or sprinklers) which make the outflow rate depend on the pressure.

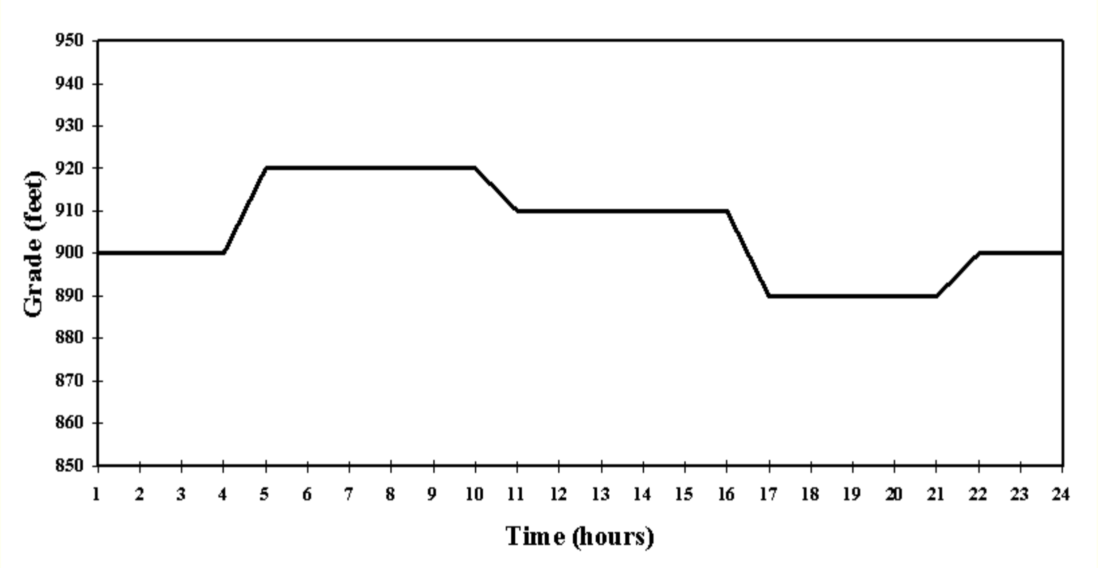
Reservoirs

Reservoirs are nodes that represent an infinite external source or sink of water to the network. They are used to represent external water sources, such as lakes, rivers, groundwater aquifers, and tie-ins to other systems. Reservoirs can also serve as water quality source points.

The primary input properties for a reservoir are its hydraulic head (equal to the water surface elevation if the reservoir is not under pressure) and its initial quality for water quality analysis.

Because a reservoir is a boundary point to a network, its head and water quality cannot be affected by what happens within the network. Therefore it has no computed output properties. Its head can remain constant or can be made to vary with time by assigning a time pattern to it. The former type is called a constant-head reservoir and the latter type is referred to as a variable-head reservoir.

Variable-head reservoirs can be used to explicitly define boundary conditions of varying storage characteristics. These facilities are extremely useful in modeling system partitioning of multiple pressure zones by accurately describing the influence of aqueducts and other facilities which are acting as a source to the modeled area. They are also useful in calibration process whereby selected supply (such as well draw-down) and storage levels (such as storage tanks) are assigned a water level trajectory and are modeled by field data. The variation in the water level with time is defined as a time varying pattern (see TIME PATTERNS below). An example of a variable-head reservoir hydraulic grade pattern is as follows:



Tanks

Tanks are nodes with storage capacity, where the volume of stored water can vary with time during an extended period simulation. The primary input properties for tanks are:

- Bottom elevation (where water level is zero)
- Diameter (or shape if non-cylindrical)
- Initial, minimum and maximum water levels
- Initial water quality.

The principal outputs computed over time are:

- Hydraulic head (water surface elevation)
- Percent full

- Water quality.

Tanks are required to operate within their minimum and maximum levels. InfoWater stops outflow if a tank is at its minimum level and stops inflow if it is at its maximum level. Tanks can also serve as water quality source points.

InfoWater models the change in water level of a tank with the following equation:

$$\Delta y = \frac{q}{A}\Delta t$$

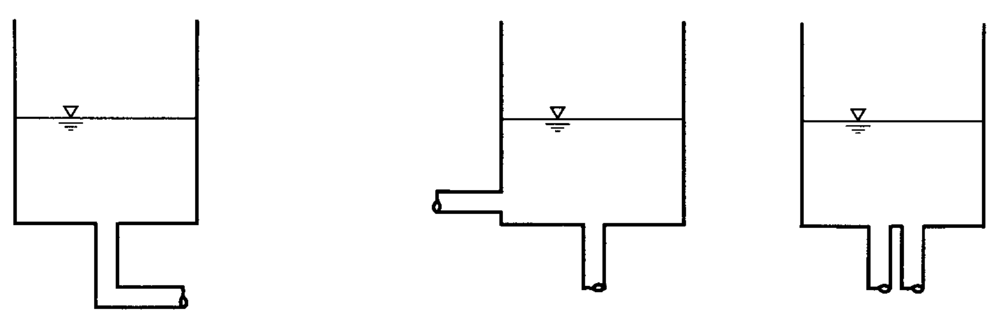
where D *y* = change in water level, ft

q = flow rate into (+) or out of (-) tank, cfs

A = cross-sectional area of the tank, ft^2

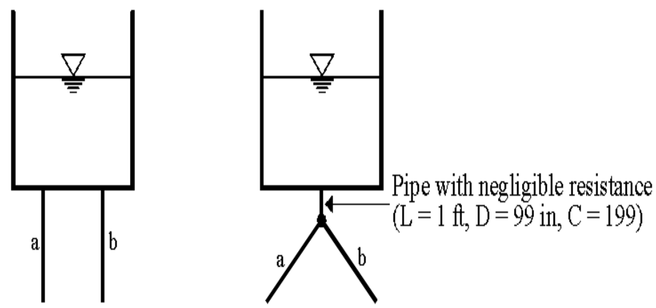
Δt = time interval, sec

A single inlet/outlet or multiple inlets and outlets can be specified for any tank in the distribution system. The number of pipes connecting the tank is used to describe the inlet/outlet configuration. This allows tanks with common or separate inlet/outlet to be accurately modeled. The figure below illustrates this inlet/outlet configuration.



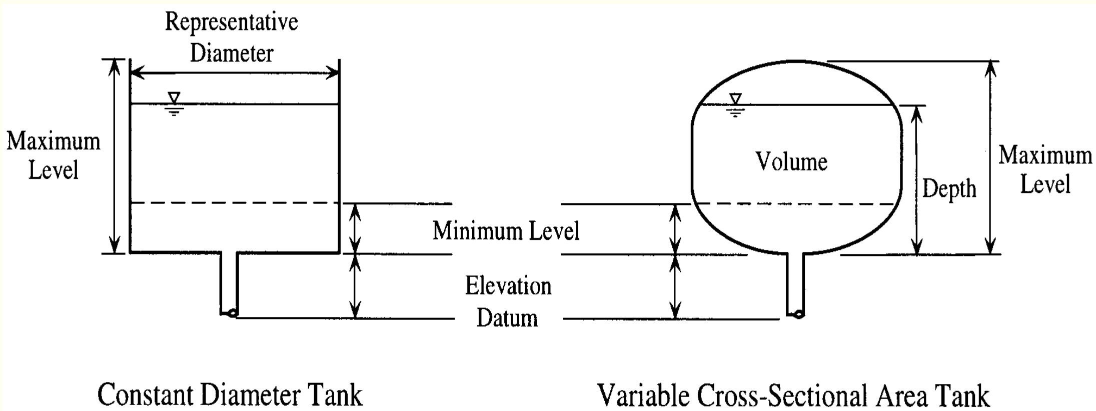
Single Inlet/Outlet Tank   Separate Inlet/Outlet Tanks

Note - Tanks with multiple feeds can also be represented as a single feed tank. Simply place a junction node in the vicinity of the tank, connect all the separate feeds to the junction node, and provide a pipe with negligible resistance (short length + large diameter) connecting the new junction node and the tank. This arrangement is illustrated in the figure below.



InfoWater allows tanks of any shape (variable cross-sectional area) to be accurately modeled. A tank can be modeled as a constant diameter (cylindrical) tank where a representative diameter is specified as the diameter corresponding to the average tank area. If a storage tank can not be adequately described by a representative diameter, InfoWater allows you to define data for the stored volume as a function of depth of water (see CURVE below).

For an extended period simulation, InfoWater accurately calculates the change in water level of a tank. The tank water level is allowed to vary between the user-specified range of minimum and maximum permissible water levels in the tank. If the water level in the tank during a draining cycle drops below the minimum level specified (e.g., tank is empty), the program automatically closes the outgoing pipe and no additional flow can exit the tank. Similarly, if the water level in the tank during a filling cycle reaches the maximum level specified (e.g., tank is full), the incoming pipe is closed and no additional flow from the system can feed the tank. This procedure accurately simulates the action of an altitude valve. The figures below illustrates the characteristics of constant and variable area tanks.



During an extended period simulation, hydropneumatic tanks can also be accurately modeled. A hydropneumatic tank is normally pressurized between a high and low pressure value in a given draining (water released) and filling (water pumped) cycle. This tank is modeled as a cylindrical storage tank where the maximum and minimum water levels correspond to the high and low pressures, respectively. The equivalent tank diameter is calculated based on the volume of water discharged by the hydropneumatic tank between the minimum and maximum water levels during its cycle. Because the cycle time period for a hydropneumatic tank is usually short, a relatively small hydraulic time step should be used in the analysis.

Emitters

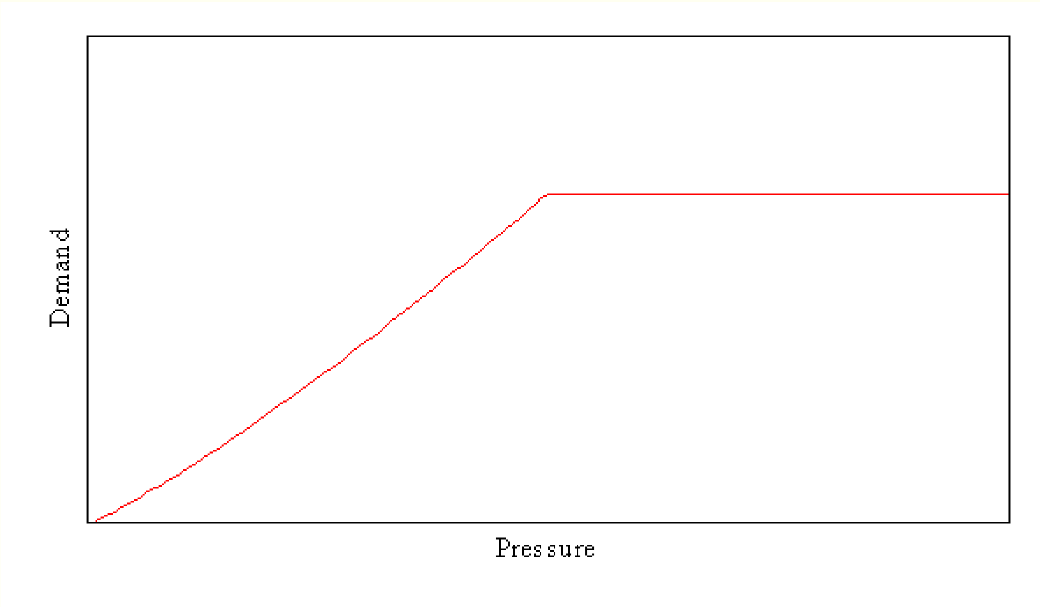
Emitters are devices associated with junctions that model the flow through a nozzle or orifice that discharges to the atmosphere. The flow rate through the emitter varies as a function of the pressure available at the node:

$$q = C\,p^{\gamma}$$

where q = flow rate, p = pressure, C = discharge coefficient, and g = pressure exponent. For nozzles and sprinkler heads g equals 0.5 and the manufacturer usually provides the value of the discharge coefficient in units of gpm/psi$^{0.5}$ (stated as the flow through the device at a 1 psi pressure drop).

Emitters can be used to model flow through sprinkler systems and irrigation networks. They can also be used to simulate leakage in a pipe connected to the junction (if a discharge coefficient and pressure exponent for the leaking crack or joint can be estimated).

A very important characteristic of emitters is their ability to also effectively simulate the effects of pressure-dependent demands. InfoWater allows you to define the data that describes the variation of the external demand as a function of pressure (see CURVE below). An example of a pressure-dependent demand curve is shown below.



Pipes

Pipes convey water from one point in the network to another. InfoWater assumes that all pipes are flowing full at all times. Flow direction is from the end at higher hydraulic head (internal energy per weight of water) to that at lower head. The principal hydraulic input parameters for pipes are:

- Start and end nodes
- Diameter
- Length
- Roughness coefficient (for determining headloss)
- Minor (local) loss coefficient
- Status (open, closed, or contains a check valve)

- Flow totalizer (flow meter).

The status parameter allows pipes to implicitly contain shutoff (gate) valves, and check (non-return) valves which allow flow in only one direction.

The water quality inputs for pipes consist of:

- Bulk reaction coefficient
- Wall reaction coefficient.

These coefficients are explained more thoroughly in Section 4.4 below.

Computed outputs for pipes include:

- Volumetric flow rate
- Velocity
- Headloss
- Headloss per 1,000 unit of length (hydraulic gradient)
- Darcy-Weisbach friction factor
- Average reaction rate (over the pipe length)
- Average water quality (over the pipe length).

The hydraulic head lost by water flowing in a pipe due to friction with the pipe walls can be computed using one of three different formulas:

1 The Hazen-Williams formula

2. The Darcy-Weisbach (Colebrook-White) formula

3. The Chezy-Manning formula.

The Hazen-Williams formula is the most frequently used headloss equation for water distribution systems in the United States. It cannot be used for liquids other than water and was originally developed for turbulent flow only. The Darcy-Weisbach formula applies over all flow regimes and to all liquids. The Chezy-Manning formula is more commonly used for open channel flow.

Each formula uses the following equation to compute headloss between the start and end node of the pipe:

$$h_L = aq^b$$

where h_L is the headloss in feet, q is the flow in cfs, a is a resistance coefficient, and b is a flow exponent. Table 1 lists values of the resistance coefficients and flow exponents for each formula. Note that each formula uses a different pipe roughness coefficient that must be determined empirically. Table 2 lists general ranges of these coefficients for different types of new pipe materials.

Table 1 - Pipe Headloss Formulas

Formula	**Resistance Coefficient (*a*)**	**Flow Exponent (*b*)**
Hazen-Williams	$4.727\ C^{-1.852}\ d^{-4.871}\ L$	1.852
Darcy-Weisbach	$0.0252\ f(e,d,q)\ d^{-5}\ L$	2
Chezy-Manning (full pipe flow)	$4.66\ n^{2}\ d^{-5.33}\ L$	2

Notes: C = Hazen-Williams roughness coefficient

e = Darcy-Weisbach roughness coefficient (ft)

f = friction factor (dependent on e, d, and q)

n = Manning roughness coefficient

d = pipe diameter (ft)

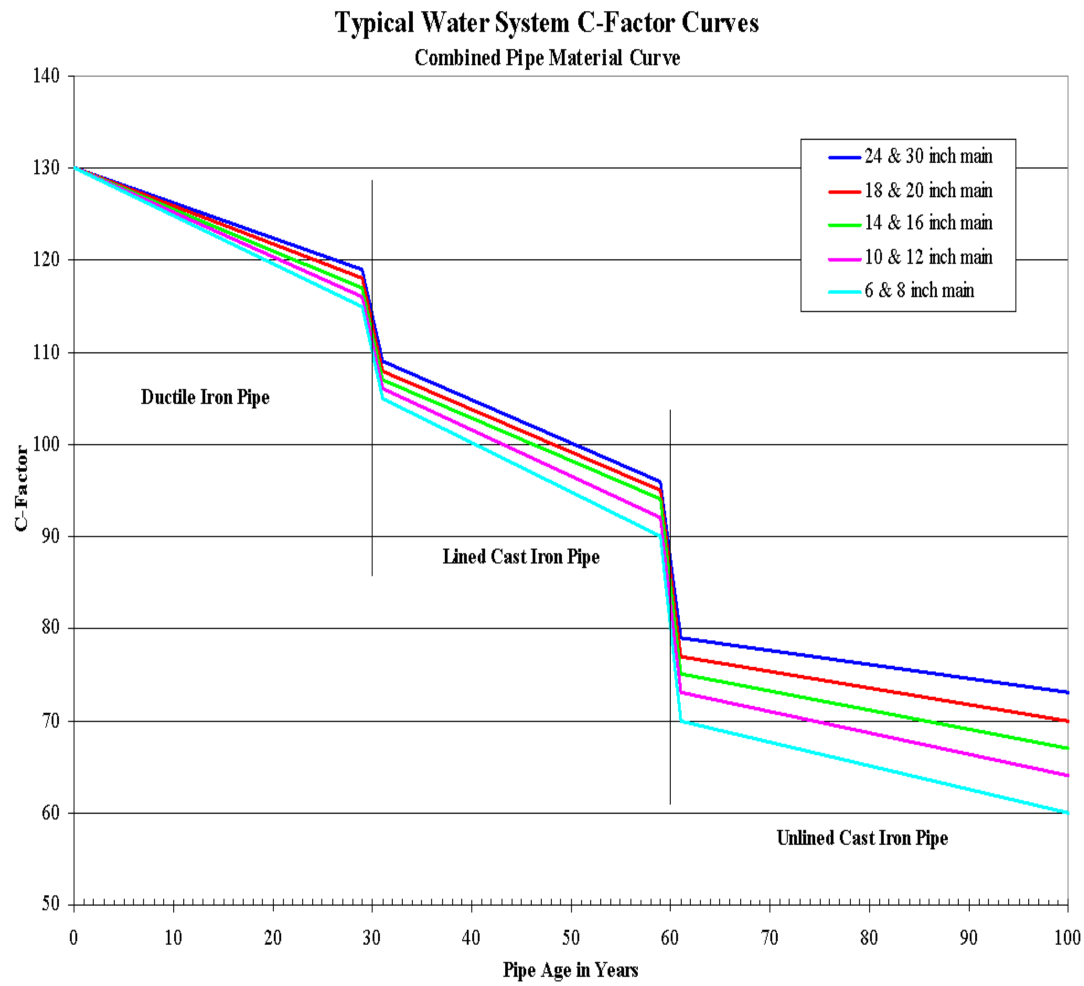
L = pipe length (ft)

Table 2 - Roughness Coefficients for New Pipe

Material	**Hazen-Williams *C***	**Darcy-Weisbach e, millifeet**	**Manning *n***

Asbestos Cement	140	0.005	0.011
Cast Iron	130-140	0.85	0.012-0.015
Concrete or Concrete Lined	120-140	1.0-10	0.012-0.017
Copper	135	0.005	0.011
Galvanized Iron	120	0.5	0.015-0.017
Lead	135	0.005	0.011
Plastic	140-150	0.005	0.011-0.015
Steel	140-150	0.15	0.015-0.017
Wood Stave	120	0.6	0.011-0.013

Be aware that the roughness coefficient depends on type and condition of the pipe and can change considerably with age, diameter, material, soil type, and water quality characteristics. Typical water system C-factor curves are shown below.



With the Darcy-Weisbach formula InfoWater uses different methods to compute the friction factor *f* depending on the flow regime:

The Hagen–Poiseuille formula is used for laminar flow (*Re* < 2,000).

$$f = \frac{64}{Re}$$

The Swamee and Jain approximation to the Colebrook-White equation is used for fully turbulent flow (*Re* > 4,000).

$$f = \frac{0.25}{\left[Ln\left(\frac{\varepsilon}{3.7d} + \frac{5.74}{Re^{0.9}}\right)\right]^2}$$

A cubic interpolation (Dunlop 1991) from the Moody Diagram is used for transitional flow (2,000 < *Re* < 4,000).

$$f = (X1 + R(X2 + R(X3 + X4)))$$

where

$$R = \frac{Re}{2000}$$

$$X1 = 7FA - FB$$

$$X2 = 0.128 - 17FA + 2.5FB$$

$$X3 = -0.128 + 13FA - 2FB$$

$$X4 = R(0.032 - 3FA + 0.5FB)$$

$$FA = (Y3)^{-2}$$

$$FB = FA\left(2 - \frac{0.00514215}{(Y2)(Y3)}\right)$$

$$Y2 = \frac{\varepsilon}{3.7d} + \frac{5.74}{Re^{0.9}}$$

$$Y3 = -0.86859 Ln\left(\frac{\varepsilon}{3.7d} + \frac{5.74}{4000^{0.9}}\right)$$

Pipes can be set open or closed at pre-set times (time switch), when tank water levels fall below or above certain set-points (grade switch), or when nodal pressures (pressure switch) or pipe flow rates (flow switch) fall below or above certain set-points.

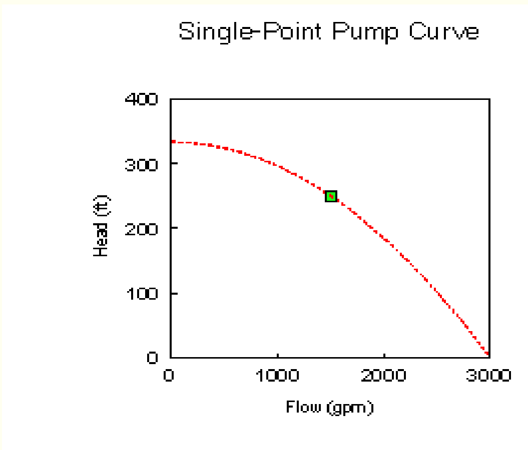
For an extended period simulation, Flow Totalizers can be designated for any pipe in the distribution system. These are flow meters that give bi-directional measurements of the total volume of water passing through the pipe.

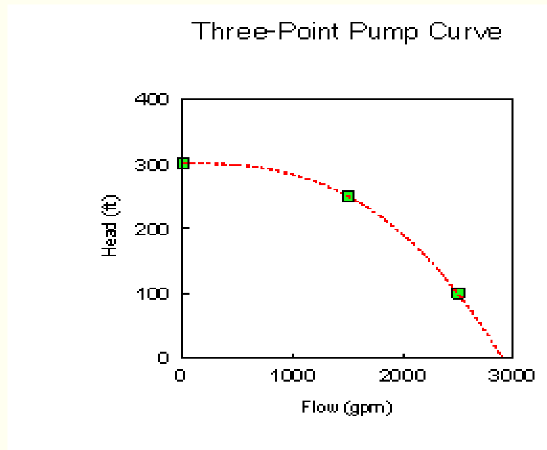
Note - Minor losses (also called local losses) may also occur in pipes by the added turbulence that occurs at bends and fittings when water flows through a pipe. Click here to learn more about minor losses.

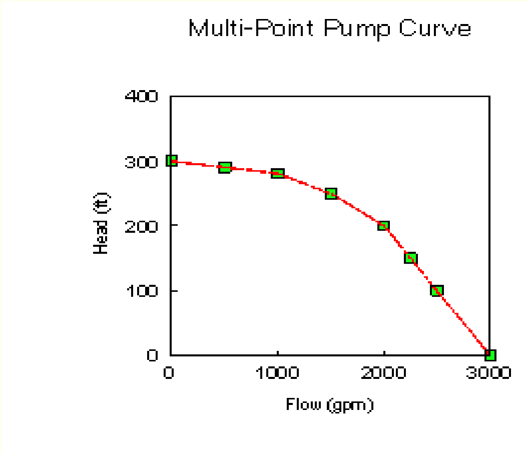
Pumps

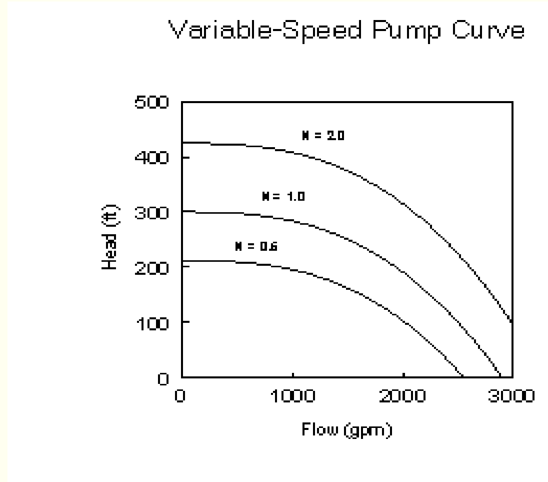
A pump imparts energy to a fluid thereby raising its hydraulic head. The relationship describing the head imparted to a fluid as a function of its flow rate through the pump is termed the pump characteristic curve.

The principal input parameters for a pump are its start and end pipes, inlet pipe diameter, and its characteristic curve. The suction pipe diameter is required only for computing the available NPSH for the pump (see NPSH below). InfoWater uses a different shape of pump curve depending on the number of head-flow data points supplied.









Example Pump Curves

Single-Point Curve - A single-point pump curve is defined by a single head-flow combination that represents a pump's desired operating point (design point). InfoWater adds two more points to the curve by assuming a shutoff head at zero flow equal to 133% of the design head and a maximum flow at zero head equal to twice the design flow. It then treats the curve as a three-point curve.

Three-Point Curve - A three-point pump curve is defined by three operating points: a Low Flow point (flow and head at low or zero flow condition), a Design Flow point (flow and head at desired operating point), and a Maximum Flow point (flow and head at maximum flow). InfoWater tries to fit a continuous function of the form:

$$h_G = h_0 - a q^b$$

where h_G is the head gain imparted by the pump in ft, q is the flow through the pump in cfs, h_o is the shutoff head in feet, a is a resistance coefficient, and b is a flow exponent. By supplying InfoWater with the shutoff head h_o and two other points [(h_1, q_1), (h_2, q_2)] on the pump curve, the program is able to estimate values for a and b from:

$$b = \log\left(\frac{h_0 - h_2}{h_0 - h_1}\right) \Big/ \log\left(\frac{q_2}{q_1}\right)$$

$a = (h_0 - h_1)/q_1^b$

Multi-Point Curve – If the pump characteristic curve is not smooth (irregular shape curve) then additional data points may be required (multiple points). A multi-point pump curve is defined by providing either a pair of head-flow points or four or more such points. InfoWater creates a complete curve by connecting the points with straight-line segments.

For variable speed pumps, the pump curve shifts as the speed changes. The relationships between flow (*q*) and head (*h*) at speeds n_1 and n_2 are

$$\frac{q_1}{q_2} = \frac{n_1}{n_2}$$

$$\frac{h_1}{h_2} = \left(\frac{n_1}{n_2}\right)^2$$

By definition, the original pump curve supplied to the program has a relative speed setting of 1. If the pump speed doubles, then the relative setting would be 2; if run at half speed, the relative setting is 0.5 and so on. The figure above illustrates how changing a pump's speed setting affects its characteristic curve.

Another way to represent a pump when its characteristic curve is unknown is to assume that it adds energy to the fluid at a constant rate. In this case the equation of the pump curve would be

$$h_G = \frac{8.81Hp}{q}$$

where h_G is the head gain imparted by the pump in ft, *Hp* is the pump useful horsepower, and *q* is the flow through the pump in cfs. The latter quantity can be computed based on an initial estimate of the flow and head at which the pump will operate. This type of pump curve is used to represent constant power pumps. It should only be used for steady-state preliminary design studies when the specific operating characteristics of the pump are not known.

Note - Constant power pumps will allow flow reversal and should not be used under a zero flow condition (no flow passing through the pump).

Flow through a pump described by operating data (not described by constant power) is unidirectional and pumps must operate within the head and flow limits imposed by their characteristic curves. If the system conditions require more head than the pump can produce, InfoWater shuts the pump off. If more than the maximum flow is required, InfoWater extrapolates the pump curve to the required flow, even if this produces a negative head. In both cases, a warning message will be issued.

As with pipes, pumps can be turned on and off at preset times, when tank levels fall below or above certain set-points, or when nodal pressures fall below or above certain set-points. A pump's operation can also be described by assigning it a time pattern of relative speed settings.

Net Positive Suction head - Cavitation occurs when the absolute pressure at the pump inlet falls below the vapor pressure and vapor pockets are formed. This phenomenon can reduce pump capacity and efficiency and can cause noise and vibration as well as permanent damage to the pump. The term Net Positive Suction Head (NPSH) is frequently used in the design and evaluation of pumping systems with regard to cavitation avoidance. NPSH is defined as the total head at the suction side of the pump minus the vapor pressure head of the liquid being transported. Mathematically, this can be expressed as:

$$NPSH = \frac{Ps}{\gamma} + \frac{V_s^2}{2g} - \frac{Pv}{\gamma}$$

where

P_s = absolute pressure at pump inlet, psi V_s = velocity at pump inlet, fps

P_v = absolute vapor pressure of water, psi

g = specific weight of water, lb/ft^3

g = acceleration due to gravity, ft/sec^2

The vapor pressure head of water in absolute notation as a function of temperature is given in Table 3. To determine whether cavitation will be a problem, InfoWater will compute the NPSH available and the cavitation index based on the required NPSH. The required NPSH is evaluated from the multiple point curve (NPSH required vs flow) supplied by the user. This curve is based on performance tests and is normally available from the pump manufacturer. The cavitation index is the ratio of the available NPSH to the required NPSH. A cavitation free operation results (theoretically) when the cavitation index is greater than one. When the cavitation index is less than one, cavitation problems would be likely to occur and consequently, either another pump should be chosen or the suction piping system should be altered to increase the available NPSH to an acceptable level.

Note - If the required NPSH curve for a specific pump is not supplied by the user then InfoWater will only compute the available NPSH and the cavitation index will not be reported.

Note - If the suction piping diameter for a specific pump is not supplied by the user, then InfoWater will not compute the available NPSH.

Note - Only pumps described by constant power input (hp or kW) can be connected to pipes with check valves. Pumps described by characteristic curves cannot be connected to pipes with check valves.

Table 3 - Vapor Pressure Head of Water

Temperature (° F)	Vapor Pressure Head (ft)	Temperature (° C)	Vapor Pressure Head (m)
32	0.20	0	0.06
40	0.28	5	0.09
50	0.41	10	0.12
60	0.59	15	0.17
70	0.84	20	0.25

80	1.17	25	0.33
90	1.61	30	0.44
100	2.19	35	0.58
110	2.95	40	0.76
120	3.91	45	0.98
130	5.13	50	1.26
140	6.67	55	1.61
150	8.58	60	2.03

Control Valves

Control valves limit the pressure or flow at a specific point in the network. Their principal input parameters include:

- Start and end pipes
- Diameter
- Setting
- Status.

The computed outputs for a valve are flow rate and headloss.

InfoWater can model nine types of control valves.

Pressure Reducing Valve (PRV) – Pressure regulating valves limit the pressure on their downstream end to not exceed a pre-set value when the upstream pressure is above the setting. InfoWater automatically computes in which of three different states a PRV can be in:

- Partially opened (i.e., active) to achieve its pressure setting on its downstream side when the upstream pressure is above the setting
- Fully open if the upstream pressure is below the setting
- Closed if the pressure on the downstream side exceeds that on the upstream side (i.e., reverse flow is not allowed).

Pressure Sustaining Valve (PSV) – Pressure sustaining valves try to maintain a minimum pressure on their upstream end when the downstream pressure is below that value. InfoWater automatically computes in which of three different states a PSV can be in:

- Partially opened (i.e., active) to maintain its pressure setting on its upstream side when the downstream pressure is below this value
- Fully open if the downstream pressure is above the setting
- Closed if the pressure on the downstream side exceeds that on the upstream side (i.e., reverse flow is not allowed).

Pressure Breaker Valve (PBV) – Pressure breaker valves force a specified pressure drop to occur across the valve. Flow through the valve can be in either direction. PBVs are not true physical devices but can be used to model situations where a particular pressure drop is known to exist.

Flow Control Valve (FCV) – Flow control valves limit the flow through a valve to a specified amount. The program produces a warning message if the flow setting cannot be maintained without having to add additional head at the valve (i.e., the flow cannot be maintained even with the valve fully open).

Vacuum Breaker Valve (VBV) – Vacuum breaker valves are devices that allow air to enter at that location in the system when the pressure drops to atmospheric level. They are generally installed at high points in a pipeline to prevent low pressure (cavitation). When the pressure drops to zero, the valve becomes active and will compute the correct flow rate required to maintain atmospheric condition.

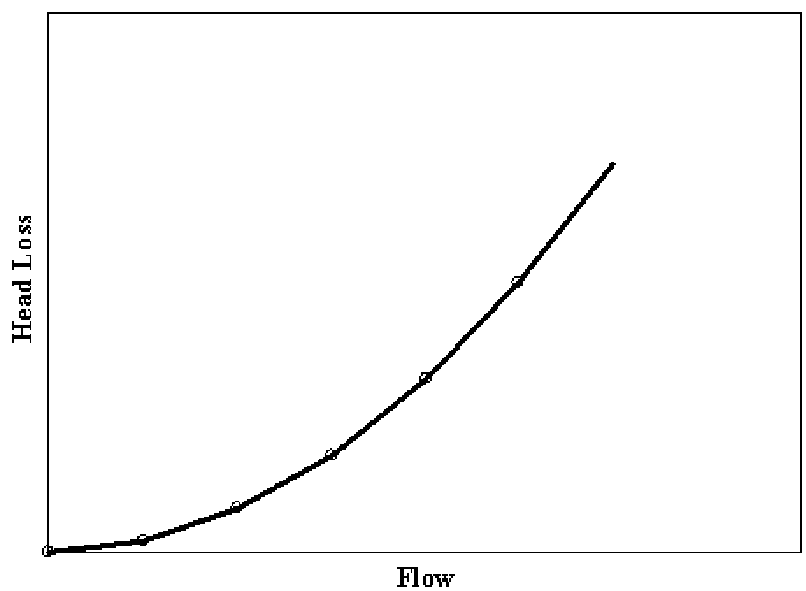
Throttle Control Valve (TCV) – Throttle control valves simulate a partially closed valve by adjusting the minor headloss coefficient (by controlling the percentage open settings or degree opening) of the valve. They are normally used to increase or decrease flows or to control pressures in the system. A relationship between the degree (or percentage) to which a valve is closed and the resulting headloss coefficient is usually available from the valve manufacturer or can be derived from field tests (based on measured flow through the valve, headloss across the valve and percentage (degree) opening setting). Alternately, a curve can be used to define the relationship between valve openings and loss coefficients. In this case, the X values represent the percentage (degree) opening settings and the Y values designate the resulting headloss coefficients. Based on the valve opening settings specified, loss coefficients are directly computed by InfoWater for linearly interpolated values of intermediate percentage (degree) opening settings. When such a curve is used, this type of valves is also referred to as **Motorized Throttled Valves (MTVs)**.

Float Valve (FLV) – Many tanks are fitted with float valves (e.g., ball float valves) on the inlet pipe to control rate of flow and prevent overflow. These valves gradually close (increase the resistance of the inlet pipe) as the water level in the controlling tank rises. The headloss across the valve is modeled based on any user specified pairs of headloss-flow points (a minimum of three headloss-flow data points is required). The status of the float valve depends on the local hydraulic conditions and the previous valve status. The valve is **active** if the water level in the controlling tank is below the lower control level specified or the water level is between the lower and upper control levels specified and the valve is previously active. The valve is **closed** if the water level in the controlling tank is at or above the upper control level specified or the water level is between the lower and upper control levels specified and the valve is previously closed.

A float valve can also be implicitly modeled as follows:

1. The tank is replaced by two tanks. One is placed at the outlet of the actual tank with an HGL set to the water level in the tank, and the second tank is placed on the inlet to the actual tank with an HGL set at some higher level to simulate the effect of the valve.
2. The tank and float valve are modeled as a reservoir and a junction node demand. The network is split at the tank and the valve is simulated by a reservoir at the outlet, set to the appropriate head, and a demand on the inlet, set to the appropriate inflow.
3. A dummy pipe is added at the inlet of the tank and the pipe resistance is set to the resistance imposed by the valve.

General Purpose Valves (GPV) – General purpose valves simulate the headloss across the valve based on user specified pairs of headloss-flow points instead of following one of the standard hydraulic formula. A minimum of three headloss-flow data points is required (see CURVE below). This type of valve can be effectively used to model turbines, well draw-down or reduced-flow backflow prevention valves.



Pressure Relief Valves (PFV) – Pressure relief valves are mainly used to prevent an unexpected rise of line pressure above a predetermined intensity. These valves are normally utilized in pressure zones with no water storage facilities. They are wide open when the upstream pressure exceeds the set point, otherwise they remain closed. In the case the valve is wide open, it usually discharges to atmospheric pressure or to a lower pressure zone. This valve is modeled as an initially *(initial status*) closed pipe of equal diameter (with appropriate minor loss coefficient) connected to a reservoir where the hydraulic grade corresponds to the ground elevation of the upstream node (if the valve discharges to atmosphere) or to the grade of the lower pressure zone (if the valve discharges to a lower pressure zone). The valve is set (*control rules*) to open when the pressure at the upstream node exceeds the opening set pressure and to close when the upstream pressure falls below the closing set pressure.

Shutoff (gate) valves and check (non-return) valves, which completely open or close pipes, are not considered as separate valves but are instead included as a property of the pipe in which they are placed.

Each type of valve has a different type of setting parameter that describes its operating point (e.g., pressure for PRVs, PSVs, and PBVs; flow for FCVs; loss coefficient for TCVs and VBVs; degree of opening for motorized TCVs; and head loss curve for GPVs and FLVs).

Valves can have their operational control status overridden by specifying they be either completely open or completely closed. A valve's status and its setting can be changed during the simulation by using control rules.

Note - Control valves cannot be connected to pipes with check valves.

Non-Physical Components

In addition to physical components, InfoWater employs three types of informational objects to describe the behavior and operational aspects of a distribution system. The three informational objects are curves, patterns, and controls.

Curves

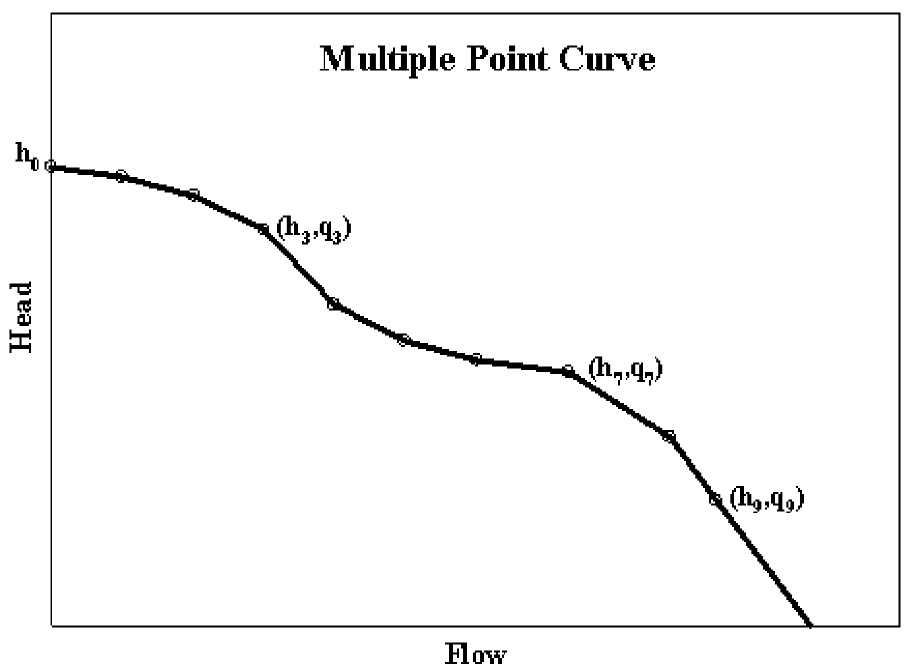
Curves are objects that contain data pairs representing a relationship between two quantities. Two or more objects can share the same curve. An InfoWater model can utilize the following types of curves:

- **Pump Curve**
- **Efficiency Curve**
- **Volume Curve**
- **Headloss Curve**
- **Headloss Coefficient Curve**
- **Pressure-Demand Curve**

Pump Curve

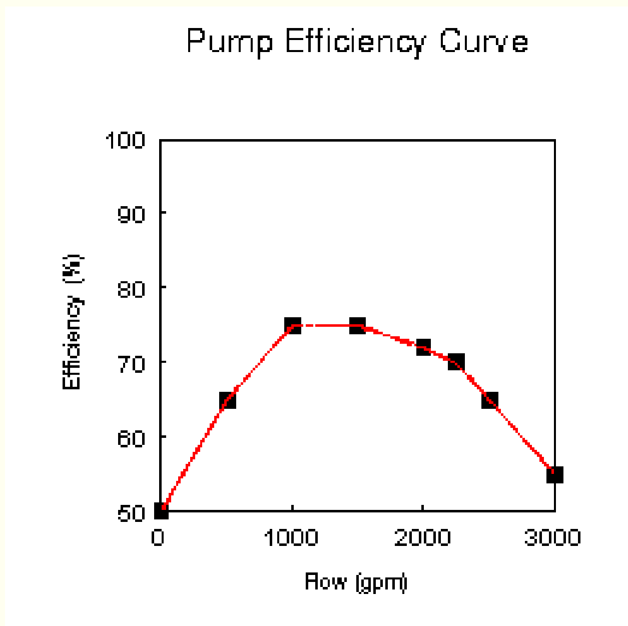
A Pump Curve represents the relationship between the head and flow rate that a pump can deliver at its nominal speed setting. Head is the head gain imparted to the water by the pump and is plotted on the vertical (Y) axis of the curve in feet (meters). Flow rate is plotted on the horizontal (X) axis in flow units. A valid pump curve must have decreasing head with increasing flow.

When four or more data are points are provided for the pump, the resulting head-flow curve is called a Multi-Point Pump Curve. It will be expressed as a piecewise linear curve of up to the last data point entered. This provides an accurate representation of the pump operation over the flow range specified.



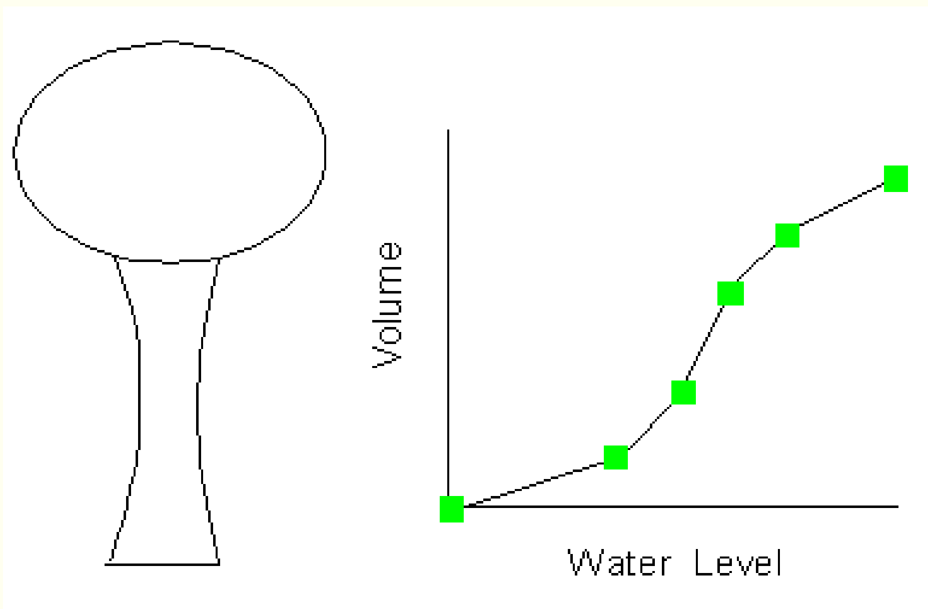
Efficiency Curve

An Efficiency Curve determines pump efficiency (Y in percent) as a function of pump flow rate (X in flow units). An example efficiency curve is shown below. Efficiency should represent wire-to-water efficiency that takes into account mechanical losses in the pump itself as well as electrical losses in the pump's motor. The curve is used only for energy calculations and requires three or more data points. When only three data points are provided, a quadratic fit is used to calculate pump efficiency. When four or more data points are provided, linear interpolation is used to compute pump efficiency.



Volume Curve

A Volume Curve determines how storage tank volume (Y in cubic feet or cubic meters) varies as a function of water level (X in feet or meters). It is used when it is necessary to accurately represent tanks whose cross-sectional area varies with height (e.g., non-circular tanks). The lower and upper water levels supplied for the curve must contain the lower and upper levels between which the tank operates. A valid volume curve must have increasing volume with increasing water height. An example of a tank volume curve is given below.

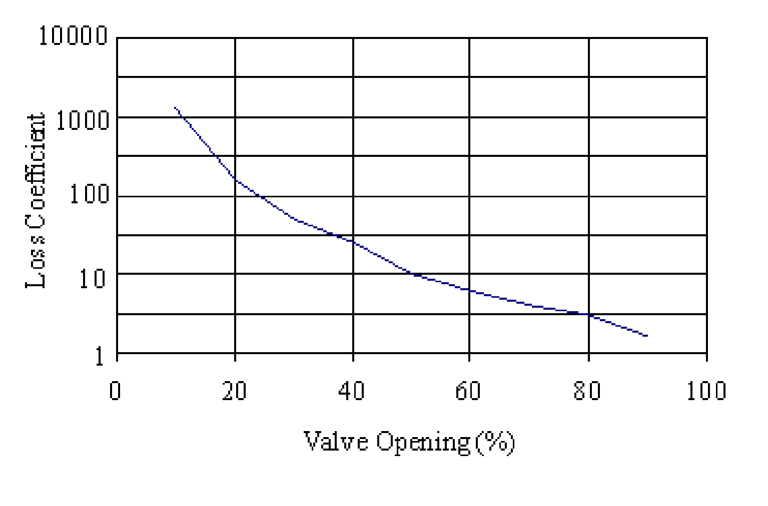


Headloss Curve

A Headloss Curve is used to describe the headloss (Y in feet or meters) through a General Purpose Valve (GPV) as a function of flow rate (X in flow units). It provides the capability to model devices and situations with unique headloss-flow relationships, such as reduced-flow backflow prevention valves, turbines, and well draw-down behavior. A minimum of two data points must be specified.

Headloss Coefficient Curve

For motorized Throttled Control Valves (TCVs), a Headloss Coefficient Curve consists of a collection of points defining the headloss coefficient (Y-axis) as a function of the percentage (degree) opening setting (X-axis). It provides the capability to model valves with unique headloss characteristics such as cone and butterfly valves. A minimum of two data points must be specified.



Coefficients for Butterfly Valve

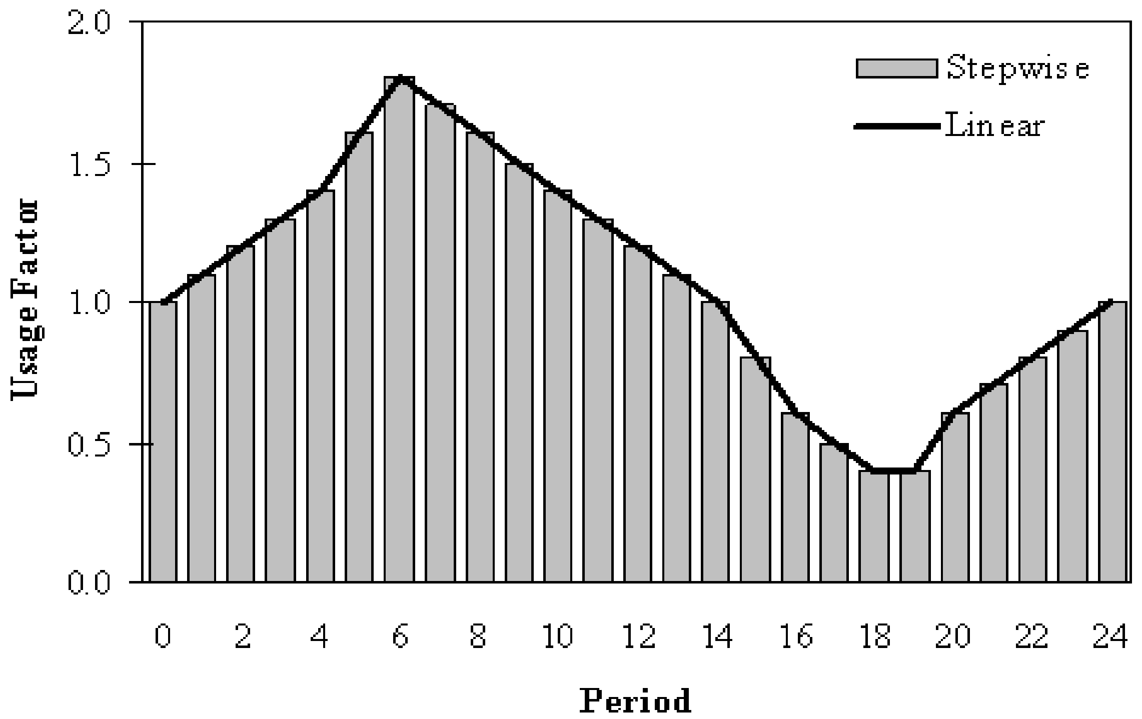
Pressure-Demand Curve

For junctions modeled as emitters, the Pressure-Demand Curve is used to describe how the flow rate (Y in flow units) varies as a function of the pressure (X in psi or m) available at the node. It provides the capability to model situations where pressure-dependent demands are significant. A minimum of two data points must be specified. InfoWater will assume that the flow rate will remain constant for any pressure range after the last data point entered.

Time Patterns

Patterns are used to represent temporal variations within the system. They can consist of a collection of multipliers (multiplication factors) that are applied to a quantity to allow it to vary over time during an extended period simulation or of a time-series of actual values for the quantity and not multipliers of their nominal values. The first type of patterns is called a "multiplication pattern" while the latter type is called a "value pattern". Nodal demands, pump schedules, and water quality source inputs can all have multiplication patterns associated with them. Variable head reservoirs and pump energy consumption rates (energy and demand charges) can all have value patterns associated with them. The time interval used in all patterns is a fixed value set by the user.

Two options are available for representing a multiplication pattern: stepwise and continuous (linear). A stepwise multiplication pattern is one that assumes a constant multiplication factor for each time interval. Within each time interval a quantity remains at a constant level equal to the product of its nominal value and the pattern's multiplier for that time period. A continuous (linear) multiplication pattern is one that linearly interpolates for the multiplication factor between two adjacent time intervals. The figure below illustrates an example multiplication pattern that might apply to daily water demands (diurnal curve), where each time period is of one hour duration.



Different patterns can be applied to individual nodes or groups of nodes to better represent actual water usage categories (e.g., single family residential, multi-family residential, commercial, industrial, and irrigation).

Although all patterns must utilize the same time interval, each can have a different number of periods. When the simulation clock exceeds the number of periods in a pattern, the pattern wraps around to its first period again.

As an example of how multiplication patterns work consider a junction node with an average demand of 10 GPM. Assume that the time pattern interval has been set to 4 hours and a pattern with the following multipliers has been specified for demand at this node:

Period	1	2	3	4	5	6
Multiplier	0.5	0.8	1.0	1.2	0.9	0.7

Then during the simulation the actual demand exerted at this node will be as follows:

Hours	0-4	4-8	8-12	12-16	16-20	20-24	24-28
Demand	5	8	10	12	9	7	5

Controls

InfoWater provides highly sophisticated and comprehensive operational control schemes to accurately simulate the hydraulic behavior of a water distribution system. During an EPS, controls specify the status of selected links as a function of time, flow rates, tank water levels, and pressures at selected points within the network. There are two categories of controls that can be used:

- Simple Controls
- Rule-Based (Logic) Controls

Simple Controls

Simple controls change the status or setting of an element (Pipe, Pump or Valve) based on:

- the water level in a tank,
- the pressure at a junction,
- the flow (pipe, pump or control valve)
- the time into the simulation,
- the time of day.

The simple control statements can be expressed in one of the following formats:

ELEMENT element_id status AT TIME time (Time Control)

ELEMENT element_id status AT CLOCKTIME clocktime **AM/PM** (Time Control)

ELEMENT element_id status IF NODE node_id BELOW value (Node Level Control)

ELEMENT element_id status IF NODE node_id ABOVE value (Node Level Control)

ELEMENT element_id status IF PIPE pipe_id BELOW value (Pipe Flow Control)

ELEMENT element_id status IF PIPE pipe_id ABOVE value (Pipe Flow Control)

where the element_`id` is the id label of the pipe, pump or valve whose status is to be changed; `node_id` is the id label of the controlling node; `status` is the desired change in element status which can be OPEN or CLOSED, a pump speed setting, or a control valve setting; `value` is the control action level which can be either water level above tank bottom if control node is a tank, pressure level if control node is a junction node, or flow rate if control element is a pipe, pump or valve; `time` is the time since the start of the simulation in decimal hours or in hours:minutes format at which change in the element setting applies; and `clocktime` is a 24-hour clock time (hour:minutes).

Some examples of simple controls are:

Open pump 23 when the water level in tank 45 drops below 23 ft and close it when the level rises; ; above 36 ft

```
PUMP 23 OPEN IF NODE 45 BELOW 23
PUMP 23 CLOSED IF NODE 45 ABOVE 36
```

Close pump 245 at 3.2 hours into the simulation

```
PUMP 245 CLOSED AT TIME 3.2
```

Open pump 300 if the flow rate in pipe 100 rises above 1,200 gpm

```
PUMP 300 OPEN IF PIPE 100 ABOVE 1200
```

Pipe 12 is repeatedly closed at 10 AM and opened at 8 PM throughout the simulation

```
PIPE 12 CLOSED AT CLOCKTIME 10 AM
PIPE 12 OPEN AT CLOCKTIME 8 PM
```

The switching mode on the element status is activated when a control statement is satisfied for the corresponding controlled node/pipe.

There is no limit on the number of simple control statements that can be used.

NOTE: Level controls are stated in terms of the height of water above the tank bottom, not the elevation (total head) of the water surface.

NOTE: Using a pair of pressure controls to open and close a pipe, pump or valve can cause the system to become unstable if the pressure settings are too close to one another. In this case using a pair of Rule-Based (Logical) controls might provide more stability.

Rule-Based Controls

With modeling applications becoming ever more sophisticated, there is a need to improve the control schemes for more advanced types of system operational activities (Boulos et al. 1998). InfoWater uses a rule-based language for defining decision rules to permit more flexibility in the ways that operation of water distribution networks can be simulated. Rule-based controls allow element status and settings (for pumps, valves and pipes) to be based on a combination of conditions that might exist in the network after an initial hydraulic state of the system is computed.

Each rule is a series of statements of the form:

```
RULE ruleID
IF condition_1
AND condition_2
OR condition_3
AND condition_4
Etc.
THEN action_1
AND action_2
Etc.
ELSE action_3
```

```
  AND action_4
      Etc.
 PRIORITY value
```

where:

`RuleID`	=	an ID label assigned to the rule
`Condition_n`	=	a condition (premise) clause
`Action_n`	=	an action clause
`Priority`	=	a priority value (e.g., a number from 1 to 5)

Condition Clause Format:

A condition (premise) clause in a Rule-Based Control takes the form of:

```
object id attribute relation value
```

where

`Object`	=	a category of network object

`Id`	=	the object's ID label
`Attribute`	=	an attribute or property of the object
`Relation`	=	a relational operator
`Value`	=	an attribute value

Some example conditional clauses are:

```
JUNCTION 23 PRESSURE > 20
TANK T200 FILLTIME BELOW 3.5
PIPE 44 STATUS IS OPEN
SYSTEM DEMAND >= 1500
SYSTEM CLOCKTIME = 7:30 AM
```

The Object keyword can be any of the following:

JUNCTION

PUMP

VALVE

RESERVOIR

TANK

PIPE

SYSTEM

When **SYSTEM** is used in a condition no ID is supplied.

The following attributes can be used with Node-type objects:

DEMAND

HEAD

PRESSURE

The following attributes can be used with Tanks:

LEVEL

FILLTIME (hours needed to fill a tank)

DRAINTIME (hours needed to empty a tank)

These attributes can be used with Pipes, Pumps and Valves:

FLOW

STATUS (**OPEN**, **CLOSED**, or **ACTIVE**) **SETTING** (pump speed or valve setting)

The **SYSTEM** object can use the following attributes:

DEMAND (total system demand) **TIME** (hours from the start of the simulation expressed either as a decimal number or in hours:minutes format)

CLOCKTIME (24-hour clock time with **AM** or **PM** appended)

Relation operators consist of the following:

= IS

<> NOT

< BELOW

> ABOVE

<= >=

Action Clause Format:

An action clause in a Rule-Based Control takes the form of:

```
object id STATUS/SETTING IS value
```

where

`Object`	=	**`PIPE`**, **`PUMP`**, or **`VALVE`** keyword
`Id`	=	the object's ID label
`Value`	=	a status condition (**`OPEN`** or **`CLOSED`**), pump speed setting, or valve setting

Some example action clauses are:

```
PIPE 23 STATUS IS CLOSED
PUMP P100 SETTING IS 1.5
VALVE 123 SETTING IS 90
```

Remarks:

a. Only the **`RULE, IF`** and **`THEN`** portions of a rule are required; the other portions are optional.
b. When mixing **`AND`** and**`OR`** clauses, the **`OR`** operator has higher precedence than **`AND`**, i.e.,
`IF A or B and C`

is equivalent to

```
IF (A or B) and C.
```

If the interpretation was meant to be

```
IF A or (B and C)
```

then this can be expressed using two rules as in

```
IF A THEN ...

IF B and C THEN ...
```

c. The **PRIORITY** value is used to determine which rule applies when two or more rules require that conflicting actions be taken. A rule without a priority value always has a lower priority than one with a value. For two rules with the same priority value, the rule that appears first is given the higher priority.

Some examples of rule-based controls are:

Example 1:

This set of rules shuts down a pump and opens a by-pass pipe when the level in a tank exceeds a certain value and does the opposite when the level is below another value.

```
RULE 1
IF TANK 1 LEVEL ABOVE 19.1
THEN PUMP 335 STATUS IS CLOSED
AND PIPE 330 STATUS IS OPEN
RULE 2
IF TANK 1 LEVEL BELOW 17.1
THEN PUMP 335 STATUS IS OPEN
```

```
AND PIPE 330 STATUS IS CLOSED
```

Example 2:

These rules change the tank level at which a pump turns on depending on the time of day.

```
RULE 3
IF SYSTEM CLOCKTIME >= 8 AM
AND SYSTEM CLOCKTIME < 6 PM
AND TANK 1 LEVEL BELOW 12
THEN PUMP 335 STATUS IS OPEN
RULE 4
IF SYSTEM CLOCKTIME >= 6 PM
OR SYSTEM CLOCKTIME < 8 AM
AND TANK 1 LEVEL BELOW 14
THEN PUMP 335 STATUS IS OPEN
```

Hydraulic Simulation Model

The hydraulic model used by InfoWater is an extended period hydraulic simulator that solves the following set of equations for each tank in the system:

$$\frac{\partial y_s}{\partial t} = \frac{q_s}{A_s}$$

$$q_s = \sum_i q_{is} - \sum_j q_{sj}$$

$$h_s = E_s + y_s$$

along with the following equations for each pipe (between nodes *i* and *j*) and each node *k*:

$$h_i - h_j = f(q_{ij})$$

$$\sum_i q_{ik} - \sum_j q_{kj} - q_k = 0$$

where the unknown quantities are:

y_s= height of water stored at node *s*, ft

q_s= flow into storage node *s*, cfs

q_{ij}= flow in pipe connecting nodes *i* and *j*, cfs

h_i= hydraulic grade line elevation at node *i* (equal to elevation head plus pressure head), ft

and the known constants are:

A_s= cross-sectional area of storage node *s* (taken as infinite for reservoirs), ft^2

E_s= elevation of node *s*, ft

q_k= flow consumed (+) or supplied (-) at node *k*, cfs

f(.)= functional relation between headloss and flow in a pipe

The Equations above express the conservation of water volume at a storage node and for pipes and junctions. Equations for the energy loss or gain due to flow within a pipe are also shown above. For known initial storage node levels y_s at time zero, the Equations are solved for all flows q_{ij} and heads h_i. This step is called "hydraulically balancing" the network, and is accomplished by using an iterative technique (modified Newton-Raphson) to solve the nonlinear equation set involved. The iterations end when the relative change in flow rates between two successive iterates is less than some specified accuracy – a steady-state network hydraulic solution is reached.

The method used by InfoWater to solve this system of equations is based on the "gradient algorithm" (Todini and Pilati 1987) or the "hybrid method" (Hamam and Brameller 1971; Osiadacz 1991) and has several attractive features. First, the system of linear equations to be solved at each iteration of the algorithm is sparse, symmetric, and positive-definite. This holds for any temporal changes in the system boundary conditions (e.g., pipes, pumps and valves on/off status). This allows highly computationally efficient sparse matrix routines (no pivoting is required) to be used for their solution (George and Liu 1981). Second, the method maintains flow continuity at all nodes after its first iteration. Third, it can readily handle pumps and valves without having to change the structure of the equation matrix when the status of these components changes. Finally, the method combines the good convergence properties of the modified Newton's method, the better conditioning properties and starting values of the loop formulation, and the inherent sparsity of the nodal formulation (Altman and Boulos 1995). The original hybrid method has also been enhanced to explicitly account for analytical gradient derivation of bounded nonlinear elements. As a result, the modified hybrid method exhibits superior convergence characteristics of any other method for nonlinear network analysis.

The extended period simulation (EPS) model involves solving a sequence of steady-state solutions linked by an integration scheme for the differential equation describing the tank dynamics (fill up and depletion). After a network hydraulic solution is obtained, flow into (or out of) each tank, q_s is found from Equation (15) and used in Equation (14) to find new tank water levels after a hydraulic time step *dt*. This process is then repeated for all subsequent time steps for the duration of the extended simulation period. Both inertial and elastic effects are neglected.

The normal hydraulic time step used in InfoWater is one (1) hour, but can be made shorter if more accuracy is needed. Shorter time steps than normal can occur automatically whenever one of the following events occurs:

- The next output reporting time period occurs
- The next time pattern period occurs
- A tank becomes empty or full
- A simple control or rule-based control is activated.

Water Quality Simulation Model

The InfoWater dynamic water quality simulator tracks the movement and fate of a dissolved substance (constituent) flowing through the network over time. It is based on the principles of conservation of mass coupled with reaction kinetics and consists essentially of three processes: advection in pipes, mixing at junctions and tanks, and kinetic reaction mechanism. The following phenomena are represented (Rossman et al. 1993; Rossman and Boulos 1996):

Advective Transport in Pipes

A dissolved substance will travel down the length of a pipe with the same average velocity as the carrier fluid while at the same time reacting (either growing or decaying) at some given rate. Longitudinal dispersion is usually not an important transport mechanism under most operating conditions. This means there is no intermixing of mass between adjacent parcels of water traveling down a pipe. Advective transport within a pipe is represented with the following equation:

$$\frac{\partial C_i}{\partial t} = -u_i \frac{\partial C_i}{\partial x} + r(C_i)$$

where C_i = concentration (mass/volume) in pipe i as a function of distance x and time t, u_i = flow velocity (length/time) in pipe i, and r = rate of reaction (mass/volume/time) as a function of concentration. For pumps and control valves, instantaneous substance advection is assumed.

Mixing at Junctions

At junctions receiving inflow from two or more pipes, the mixing of fluid is taken to be complete and instantaneous. Thus the concentration of a substance in water leaving the junction is simply the flow-weighted sum of the concentrations from the inflowing pipes. For a specific junction node *k* one can write:

$$C_{i|x=0} = \frac{\sum_{j \varepsilon I_k} q_j C_{j|x=L_j} + q_{k,ext} C_{k,ext}}{\sum_{j \varepsilon I_k} q_j + q_{k,ext}}$$

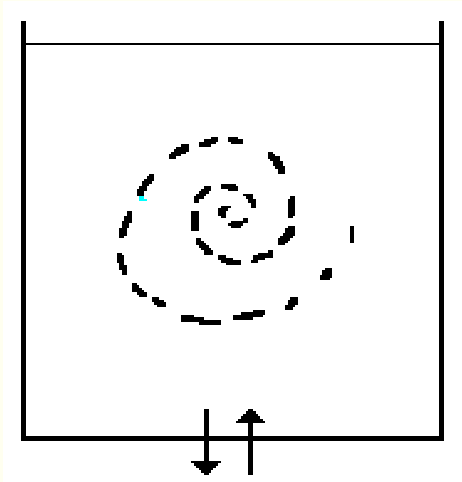
where *i* = pipe with flow leaving node *k*, I_k = set of pipes with flow into *k*, L_j = length of pipe *j*, q_j = flow (volume/time) in pipe *j*, $q_{k,ext}$ = external source flow entering the network at junction node *k*, and $C_{k,ext}$ = concentration of the external flow entering at junction node *k*. The notation $C_{i|x=0}$ is the concentration at the start of pipe *i* while $C_{i|x=L}$ is the concentration at the end of the pipe.

Mixing in Tanks

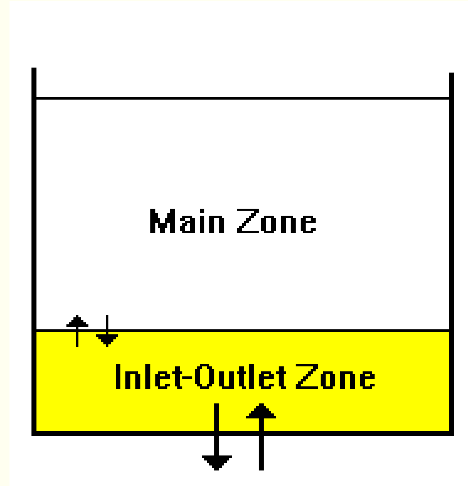
InfoWater can model four different types of mixing characteristics within storage tanks as illustrated in the figure below:

- Complete Mixing
- Two-Compartment Mixing
- FIFO (first-in first-out) Plug Flow
- LIFO (last-in first-out) Plug Flow

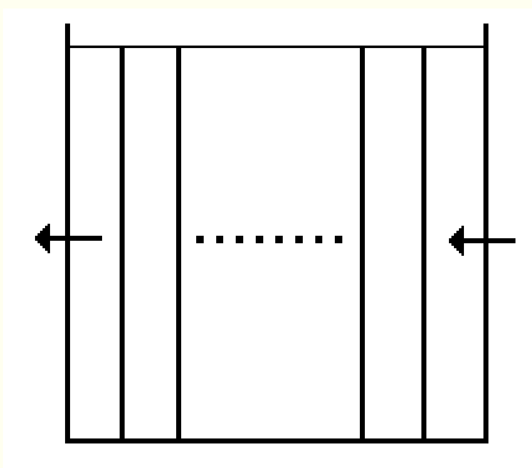
Different models can be used with different tanks within a network.



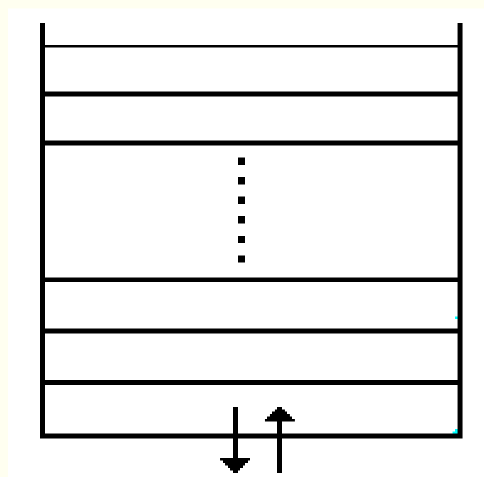
(A) Complete Mixing



(B) Two-Compartment Mixing



(C) Plug Flow – FIFO



(D) Plug Flow - LIFO

The Complete Mixing model (Figure (A)) assumes that all water that enters a tank is instantaneously and completely mixed with the water already in the tank. This is a reasonable assumption for many tanks operating under fill-and-draw conditions providing that sufficient momentum flux is imparted to the inflow. Under completely mixed conditions the concentration throughout the tank is a blend of the current contents and that of any entering water. At the same time, the internal concentration could be changing due to reactions. The following equation expresses these phenomena (Rossman et al. 1993):

$$\frac{\partial(V_s C_s)}{\partial t} = \sum_{i \varepsilon I_s} q_i \, C_{i|x=L_i} - \sum_{j \varepsilon O_s} q_j C_s + r(C_s)$$

where V_s = volume in storage at time t, C_s = concentration within the storage facility, I_s = set of pipes providing flow into the facility, and O_s = set of pipes withdrawing flow from the facility.

The Two-Compartment Mixing model (Figure (B)) divides the available storage volume in a tank into two compartments, both of which are assumed completely mixed. The inlet/outlet pipes of the tank are assumed to be located in the first compartment. New water that enters the tank mixes with the water in the first compartment. If this compartment is full, then it sends its overflow to the second compartment where it completely mixes with the water already stored there. When water leaves the tank, it exits from the first compartment, which if full, receives an equivalent amount of water from the second compartment to make up the difference. The first compartment is capable of simulating short-circuiting between inflow and outflow while the second compartment can represent dead zones. The user must supply a single parameter, which is the fraction of the total tank volume devoted to the first compartment.

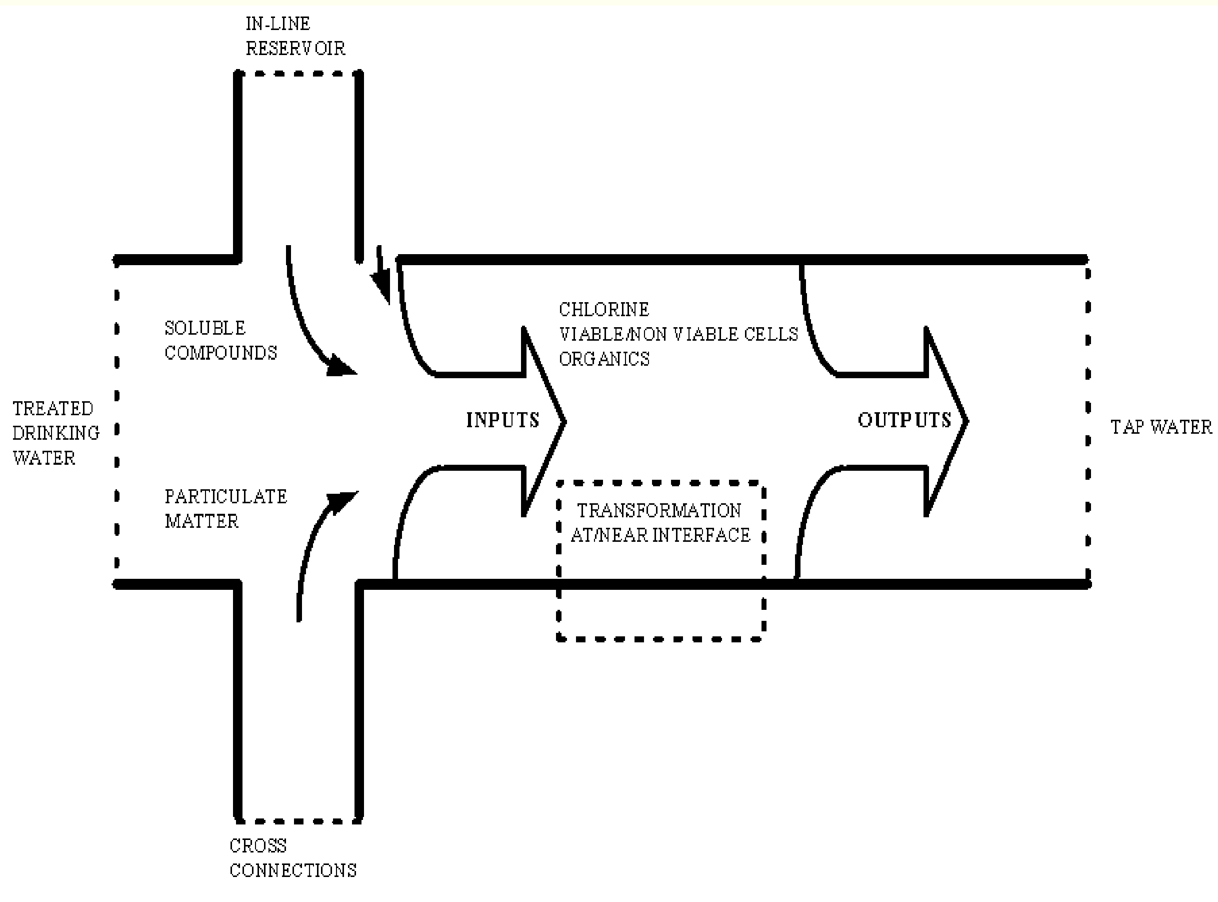
The FIFO Plug Flow model (Figure (C)) assumes that there is no mixing of water at all during its residence time in a tank. Water parcels move through the tank in a segregated fashion where the first parcel to enter is also the first to leave. Physically speaking, this model is most appropriate for baffled tanks that operate with simultaneous inflow and outflow. There are no additional parameters needed to describe this mixing model.

The LIFO Plug Flow model (Figure (D)) also assumes that there is no mixing between parcels of water that enter a tank. However in contrast to FIFO Plug Flow, the water parcels stack up one on top of another, where water enters and leaves the tank on the bottom. This type of model might apply to a tall, narrow standpipe with an inlet/outlet pipe at the bottom and a low momentum inflow. It requires no additional parameters be provided.

For a detailed description of tank mixing models and their verification, the reader is referred to Boulos et al. (1996, 1998), Grayman et al. (1996), Mau et al. (1995, 1996), and Hannoun et al. (1997, 1998).

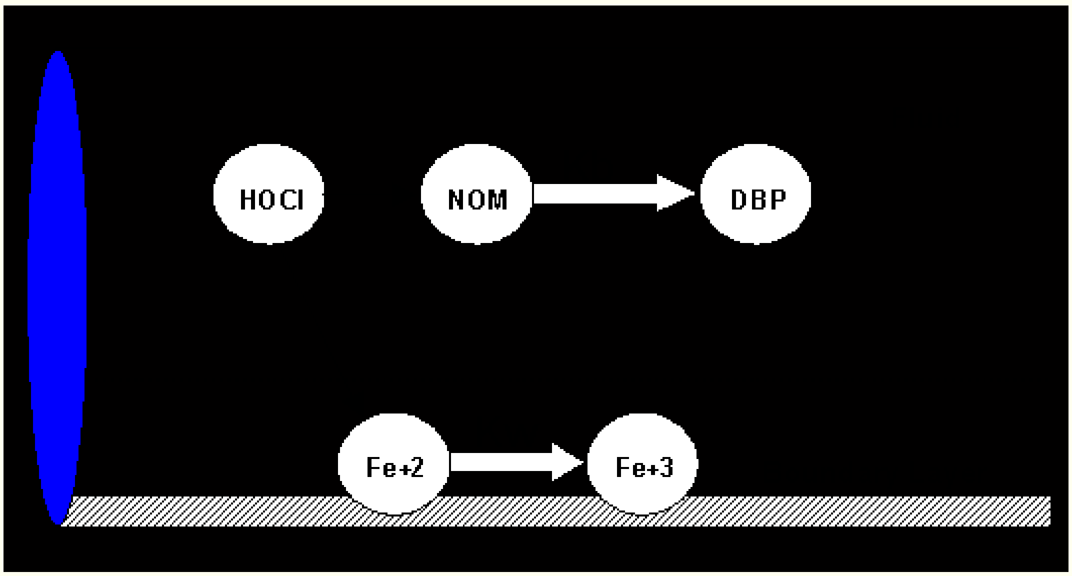
Water Quality Reactions

InfoWater can track the growth or decay of a substance by reaction as it travels through a distribution system and interacts with the various water system components (see figure below).



In order to do this it needs to know the rate at which the substance reacts and how this rate might depend on substance concentration.

Reactions can occur both within the bulk flow and with material along the pipe wall. This is illustrated in the figure below. In this example free chlorine (HOCl) is shown reacting with natural organic matter (NOM) in the bulk phase and is also transported through a boundary layer at the pipe wall to oxidize iron (Fe) released from pipe wall corrosion. Bulk fluid reactions can also occur within tanks. InfoWater allows a modeler to treat these two reaction zones separately.



Reaction Zones Within a Pipe

Bulk Reactions

InfoWater models reactions occurring in the bulk flow with *n*-th order kinetics, where the instantaneous rate of reaction (*R* in mass/volume/time) is assumed to be concentration-dependent according to

$$R = K_b C^n$$

Here K_b = a bulk reaction rate coefficient, C = reactant concentration (mass/volume), and n = a reaction order. K_b has units of concentration raised to the (1-n) power divided by time. It is positive for growth reactions and negative for decay reactions.

InfoWater can also consider reactions where a limiting concentration exists on the ultimate growth or loss of the substance. In this case the rate expression becomes

$R = K_b (C_L - C) C^{(n-1)}$ for $n > 0$, $K_b > 0$

$R = K_b (C - C_L) C^{(n-1)}$ for $n > 0$, $K_b < 0$

where C_L = the limiting concentration. Thus there are three parameters (K_b, C_L, and n) that are used to characterize bulk reaction rates. Some special cases of well-known kinetic models include the following:

Simple First-Order Decay ($C_L = 0$, $K_b < 0$, $n = 1$):

$$R = K_b C$$

The decay of many substances, such as chlorine, can be modeled adequately as a simple first-order reaction.

First-Order Saturation Growth ($C_L > 0$, $K_b > 0$, $n = 1$):

$$R = K_b(C_L - C)$$

This model can be applied to the growth of disinfection by-products, such as trihalomethanes, where the ultimate formation of by-product (C_L) is limited by the amount of reactive precursor present.

Two-Component, Second Order Decay (C_L ¹ 0, $K_b < 0$, $n = 2$):

$$R = K_b C(C - C_L)$$

This model assumes that substance *A* reacts with substance *B* in some unknown ratio to produce a product *P*. The rate of disappearance of *A* is proportional to the product of *A* and *B* remaining. C_L can be either positive or negative, depending on whether either component *A* or *B* is in excess, respectively.

Michaelis-Menton Decay Kinetics ($C_L > 0$, $K_b < 0$, $n < 0$):

$$R = \frac{K_b C}{C_L - C}$$

As a special case, when a negative reaction order *n* is specified, InfoWater will utilize the Michaelis-Menton rate equation, shown above for a decay reaction. (For growth reactions the denominator becomes $C_L + C$.) This rate equation is often used to describe enzyme-catalyzed reactions and microbial growth. It produces first-order behavior at low concentrations and zero-order behavior at higher concentrations. Note that for decay reactions, C_L must be set higher than the initial concentration present.

Koechling (1998) has applied Michaelis-Menton kinetics to model chlorine decay in a number of different waters and found that both K_b and C_L could be related to the water's organic content and its ultraviolet absorbance as follows:

$$K_b = -0.32 UVA^{1.365} \frac{(100 UVA)}{DOC}$$

$$C_L = 4.98 UVA - 1.91 DOC$$

where *UVA* = ultraviolet absorbance at 254 nm (1/cm) and *DOC* = dissolved organic carbon concentration (mg/L).

NOTE: These expressions apply only for values of K_b and C_L used with Michaelis-Menton kinetics.

Zero-Order growth (C_L = 0, K_b = 1, n = 0):

$R = 1.0$

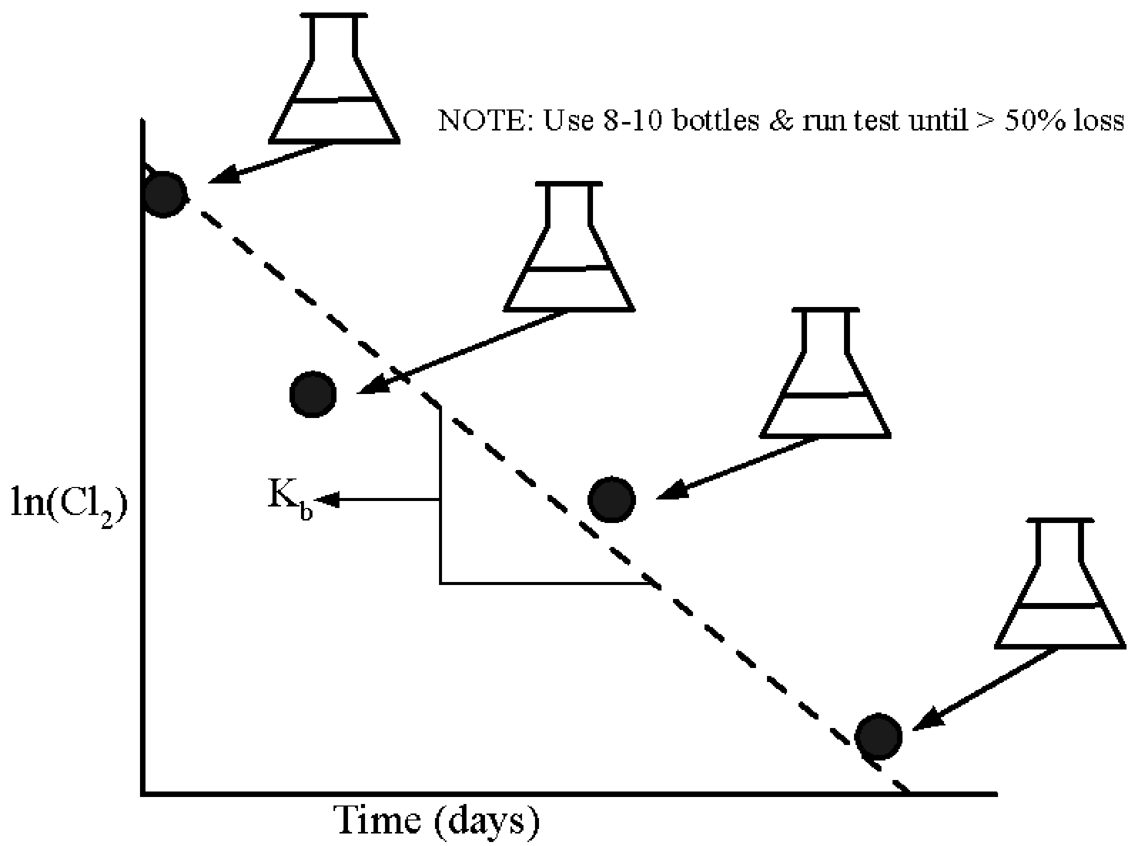
This special case can be used to model water age, where with each unit of time the "concentration" (i.e., age) increases by one unit.

The relationship between the bulk rate constant seen at one temperature (*T1*) to that at another temperature (*T2*) is often expressed using a van't Hoff - Arrehnius equation of the form:

$$K_{b2} = K_{b1} \theta^{T2-T1}$$

where q is a constant. In one investigation for chlorine, q was estimated to be 1.1 when *T1* was 20 deg. C (Koechling, 1998).

The K_b for first-order reactions can be estimated by placing a sample of water in a series of non-reacting glass bottles (make sure the bottles are sealed head-space free and that they not exposed to light) and analyzing the contents of each bottle at different points in time (it is best to obtain the water samples from the entry point of the system being modeled such as from the clearwell of the treatment plant and not the raw water source, or from the pump station feeding the zone being modeled). If the reaction is first-order, then plotting the natural log (C_t / Co) against time should result in a straight line, where C_t is concentration at time t and Co is concentration at time zero. K_b would then be estimated as the slope of this line.

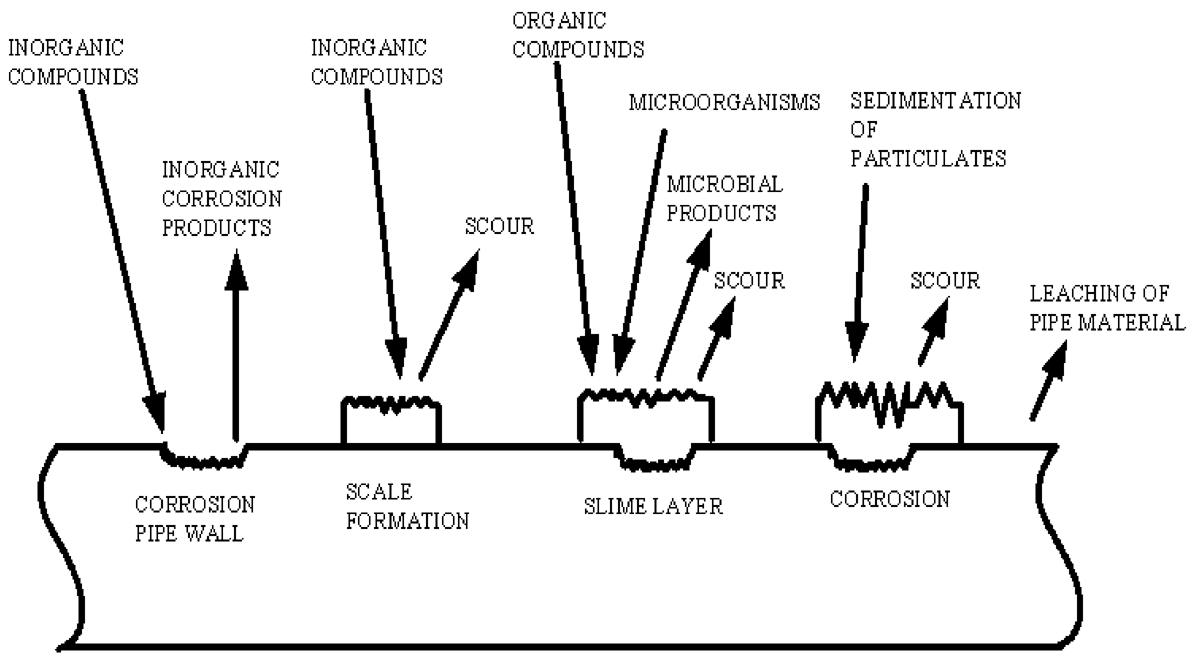


Determination of Chlorine Bulk Decay Coefficient

In general, bulk reaction coefficients will depend on the amount and type of organic matter in the water and on the temperature. Ground waters will typically be less reactive than surface waters (unless there is significant amounts of inorganic reductants such as iron). The coefficients usually increase with increasing temperature. Running multiple bottle tests at different temperatures will provide more accurate assessment of how the rate coefficient varies with temperature. In their AWWARF study, Vasconcelos et al. (1996, 1997) measured bulk decay coefficients for chlorine for 11 different waters and reported values ranging from 0.08/day to 17.7/day.

Wall Reactions

Substantial changes in water quality can take place at or near the pipe wall interface as shown in the figure below.



TRANSFORMATION AT/NEAR INTERFACE

The rate of water quality reactions occurring at or near the pipe wall can be considered to be dependent on the concentration in the bulk flow by using an expression of the form

$$R = (A/V)K_w C^n$$

where K_w = a wall reaction rate coefficient and *(A/V)* = the surface area per unit volume within a pipe (equal to 4 divided by the pipe diameter). The latter term converts the mass reacting per unit of wall area to a per unit volume basis.

InfoWater limits the choice of wall reaction order to either 0 (zero-order kinetics) or 1 (first-order kinetics), so that the units of K_w are either mass/area/time or length/time, respectively. As with K_b, K_w must be supplied to the program by the modeler. First-order K_w values can range anywhere from 0 to as much as 5 ft/day.

K_w should be adjusted to account for any mass transfer limitations in moving reactants and products between the bulk flow and the wall. InfoWater does this automatically, basing the adjustment on the molecular diffusivity of the substance being modeled and on the flow's Reynolds number. The mass transfer coefficient K_f is expressed in terms of a dimensionless Sherwood number (Sh):

$$K_f = Sh \frac{D}{d}$$

in which D = the molecular diffusivity of the species being transported (length^2/time) and d = pipe diameter. In fully developed laminar flow, the average Sherwood number along the length of a pipe can be expressed as

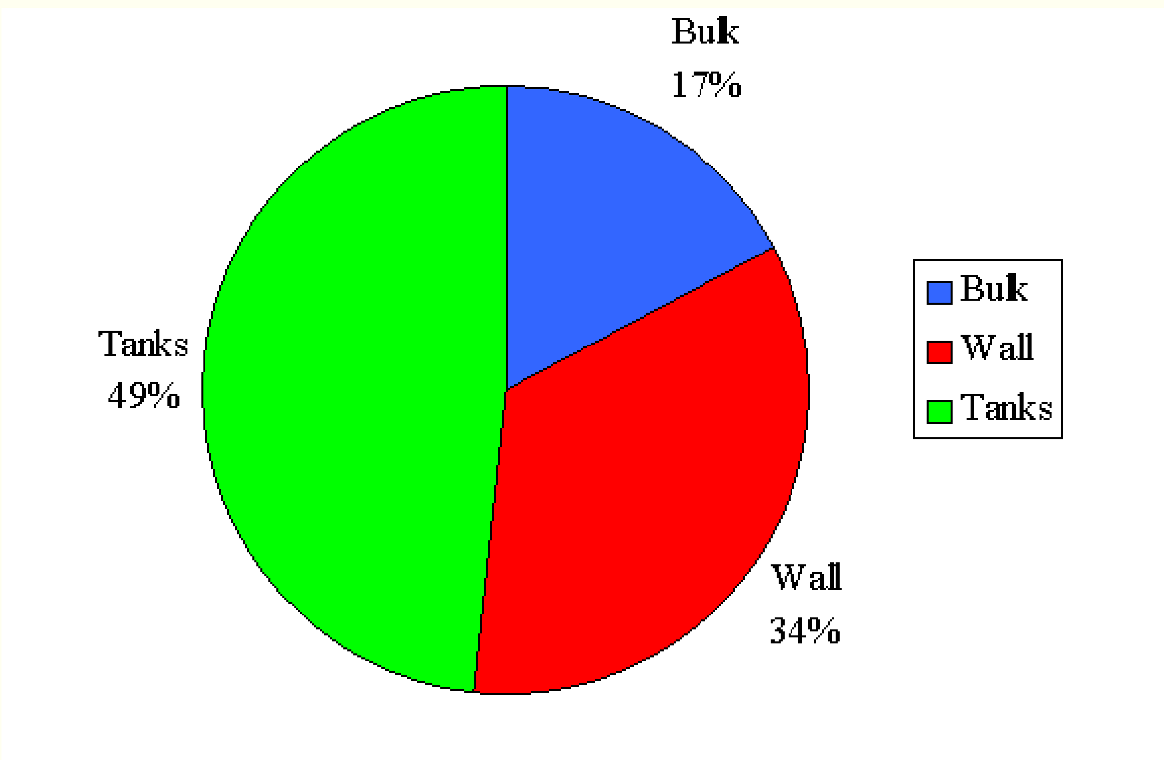
$$Sh = 3.65 + \frac{0.0668(d/L)ReSc}{1 + 0.04\left[(d/L)ReSc\right]^{2/3}}$$

in which Re = Reynolds number and Sc = Schmidt number (kinematic viscosity of water divided by the diffusivity of the chemical) (Edwards et al. 1976). For turbulent flow the empirical correlation of Notter and Sleicher (1971) can be used:

$$Sh = 0.0149 Re^{0.88} Sc^{1/3}$$

Mass transfer effects can be neglected by setting the molecular diffusivity to zero (K_f = 0).

To summarize, there are three coefficients used by InfoWater to describe reactions within a pipe. The pipe's bulk rate constant K_b and its wall rate constant K_w must be determined empirically and supplied as input to the model, while the mass transfer coefficient K_f is automatically calculated by the program. When modeling the fate of a reactive water quality constituent, InfoWater will determine the overall average reaction rates occurring throughout the network in the bulk flow, the pipe wall, and in storage tanks. The example pie chart below depicts the percentage of the overall reaction rate occurring in each location. This information can be very useful since it readily identifies what mechanism is responsible for the majority of growth or decay of a substance in the network.



The wall reaction coefficient can depend on temperature and can also be correlated to pipe age and material. It is well known that as metal pipes age their roughness tends to increase due to encrustation and tuburculation of corrosion products on the pipe walls. This increase in roughness produces a lower Hazen-Williams *C*-factor or a higher

Darcy-Weisbach roughness coefficient, resulting in greater frictional head loss in flow through the pipe.

There is some evidence to suggest that the same processes that increase a pipe's roughness with age also tend to increase the reactivity of its wall with some chemical species, particularly chlorine and other disinfectants (Vasconcelos et al. 1996, 1997). InfoWater can make each pipe's K_w be a function of the coefficient used to describe its roughness. A different function applies depending on the formula used to compute headloss through the pipe:

Headloss Formula	*Wall Reaction Formula*
Hazen-Williams	$K_w = F / C$
Darcy-Weisbach	$K_w = -F / \log(e/d)$
Chezy-Manning	$K_w = F\, n$

where C = Hazen-Williams C-factor, e = Darcy-Weisbach roughness coefficient, d = pipe diameter, n = Manning roughness coefficient, and F = wall reaction - pipe roughness coefficient. The coefficient F must be developed from site-specific field measurements and will have a different meaning depending on which head loss equation is used. The advantage of using this approach is that it requires only a single parameter, F, to allow wall reaction coefficients to vary throughout the network in a physically meaningful way.

NOTE: InfoWater *requires that water be flowing in a pipe for a wall reaction to occur. Pipes with no flow have no computed wall reaction.*

Water Age and Source Tracing

In addition to chemical transport, InfoWater can also model the changes in the age of water throughout a distribution system. Water age is the time spent by a parcel of water in the network. New water entering the network from reservoirs or source nodes enters with age of zero. Water age provides a simple, non-specific measure of the overall quality of delivered drinking water. Internally, InfoWater treats age as a reactive constituent whose growth follows zero-order kinetics with a rate constant equal to 1 (i.e., each second the water becomes a second older).

InfoWater can also perform source tracing. Source tracing tracks over time what percent of water reaching any junction node in the network had its origin at a particular source node. The source node can be any tank or reservoir in the network. Internally, InfoWater treats this source node as a constant source of a non-reacting constituent that enters the network with a concentration of 100. Source tracing is a useful tool for analyzing distribution systems drawing water from two or more different raw water supplies. It can show to what degree water from a given source blends with that from other sources, and how the spatial pattern of this blending changes over time.

System of Equations

When applied to a network as a whole, the advective transport equation for pipes and the mixing equations at junctions and tanks represent a coupled set of differential/algebraic equations with time-varying coefficients that must be solved for C_i in each pipe *i* and C_s in each storage facility *s*. This solution is subject to the following set of externally imposed conditions:

- initial conditions that specify C_i for all *x* in each pipe *i* and C_s in each storage facility *s* at time 0,

- boundary conditions that specify values for $C_{k,ext}$ and $q_{k,ext}$ for all time t at each node k which has external mass inputs
- hydraulic conditions which specify the volume V_s in each storage facility s and the flow q_i in each pipe i at all times t.

Lagrangian time-driven Transport Algorithm

InfoWater uses the Lagrangian Time-Driven Method (TDM) of Rossman and Boulos (1996) for dynamic water quality simulation, which is a hybrid of the Eulerian Discrete Volume Method (DVM) of Rossman et al. (1993) and the Lagrangian Event-Driven Method (EDM) of Boulos et al. (1995). TDM tracks the concentration and size of non-overlapping segments of water that fill each pipe of the network as they move along pipes and mix together at junctions between fixed-length time steps. These water quality time steps are typically much shorter than the hydraulic time step (e.g., minutes rather than hours) to accommodate the short times of travel that can occur within pipes.

As time progresses, the size of the most upstream segment in a pipe increases as water enters the pipe while an equal loss in size of the most downstream segment occurs as water leaves the pipe. The size of the segments in between these remains unchanged (see figure below).

The following steps occur at the end of each such time step:

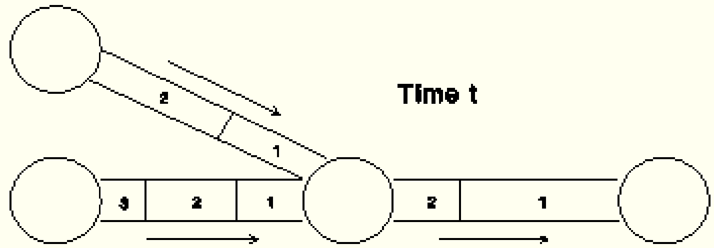
1. The water quality in each segment is updated to reflect any reaction that may have occurred over the time step.
2. The water from the leading segments of pipes with flow into each junction is blended together to compute a new water quality value at the junction. The volume contributed from each segment equals the product of its pipe's flow rate and the time step. If this volume exceeds that of the segment then the segment is destroyed and the next one in line behind it begins to contribute its volume.

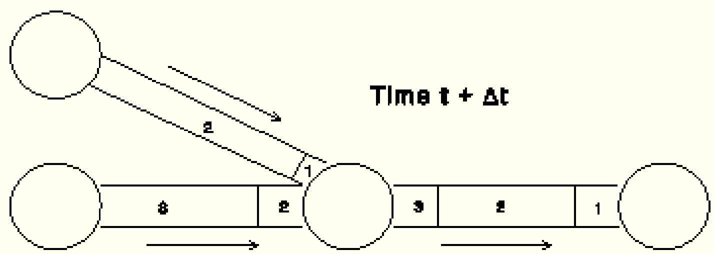
3. Contributions from outside sources are added to the quality values at the junctions. The quality in storage tanks is updated depending on the method used to model mixing in the tank (see below).
4. New segments are created in pipes with flow out of each junction, reservoir, and tank. The segment volume equals the product of the pipe flow and the time step. The segment's water quality equals the new quality value computed for the node.

To cut down on the number of segments, Step 4 is only carried out if the new node quality differs by a user-specified tolerance from that of the last segment in the outflow pipe. If the difference in quality is below the tolerance then the size of the current last segment in the outflow pipe is simply increased by the volume flowing into the pipe over the time step.

This process is then repeated for the next water-quality time step. At the start of the next hydraulic time step the order of segments in any pipes that experience a flow reversal is switched. Otherwise, no other adjustment is necessary. Initially each pipe in the network consists of a single segment whose quality equals the initial quality assigned to the upstream node.

The Lagrangian nature of TDM avoids any numerical dispersion within the interior length of pipes. However, some artificial mixing between segments can be introduced at downstream nodes when more than the leading segment in the pipe is consumed during a time step. The accuracy of the method depends on the choice of a time step and the concentration tolerance used to limit the generation of new segments.





Behavior of Segments in the Lagrangian TDM Solution Method

Fireflow Simulation Model

Water mains are normally designed on the basis of meeting fire demands. Such design practice is based on the fact that fire flow requirements usually exceed the normal domestic, industrial, and other demands imposed on the water system. Fire flow is defined as the rate of water flow, at a specified residual pressure, typically 20 psi (138 kPa), and for a specified duration that is necessary to control a major fire in a specific structure (AWWA M31). This definition is consistent with what the Insurance Services Office (ISO) calls "distribution capability" in their Fire Suppression Rating Schedule (1980). Available fire flow can also be limited by supply capability and hydrant location. Fire flows are normally imposed on the distribution system loaded for maximum-day demand.

Fire flow studies are modeling applications that impose fire flow demands on the network model. In a network representation of a water distribution system, the water demand (or load) specified at any node of the network is the flow available at that location. The mathematical relation between water demand and hydraulic head at a node can be approximated by attaching a low resistance pipe from the node to a fictitious reservoir whose water level equals the elevation head at the node. The flow in the pipe to the reservoir equals the demand at the node.

Under this condition and for any junction node designating a hydrant location, the fire flow available q_a at a target pressure p_a can be computed from (Boulos et al 1997):

$$q_a = q_f \left[\frac{p_s - p_a - c\left(p_f - p_a\right)}{p_s - p_f} \right]^{\frac{1}{n}}$$

where

$$c = (q_s / q_f)^n$$

Here, q_s designates the static demand at the node, p_s is the static pressure, q_f is the normal fire flow demand, p_f is the pressure at the normal fire demand, and n is a flow exponent that is dependent on the headloss expression used ($n \varepsilon [1.85, 2.0]$). The above equation represents the exact analytical solution of the basic pressure-flow equilibrium relationship and is applicable to any system of consistent units.

The redistribution of pressures throughout a network as flow demands and operating conditions are changed is directly accounted for by the program. InfoWater implicitly accounts for all hydraulic changes of the distribution system that are required in getting water to the hydrants. This results in efficient and accurate modeling of fire flows throughout the distribution system and for any set of boundary conditions.

InfoWater computes distribution main fire flows for all junction nodes (hydrant locations) or any user-specified network domain in the water system. The network domain can designate any portion of the distribution system for fire flow computation. It can comprise a single junction node, a collection of junction nodes, or all the network junction nodes. The target pressure (P_a) is user-specified and is defined as the minimum residual pressure allowed at each hydrant in the selected domain during a fire. The default minimum residual pressure (target pressure) is 20 psi. The computed available flow which can be delivered to the fire is compared with the required fire flow to determine the adequacy of the overall system. InfoWater also identifies the critical node with the minimum pressure, for each fire flow calculation within the selected domain. This information is necessary to determine system integrity for fire fighting needs. It is important to note that a properly calibrated hydraulic model of the water distribution system is required for fire flow modeling applications.

A unique feature of InfoWater is its capability to explicitly compute design flows at junction nodes (hydrant locations) based on a user-specified minimum design pressure to be maintained throughout the selected network domain. The design flow represents the maximum flow available at a hydrant location such that the system pressure anywhere within the selected domain does not drop below the minimum design pressure specified. Based on the critical node identified (junction node with minimum pressure), InfoWater explicitly determines the design flow for each junction node in the selected domain needed to maintain the minimum system design pressure.

Complete flexibility is allowed for specifying fire flow distribution throughout the system. Network nodes, designating hydrant locations, can be lumped in various groups, each group representing a specific type of structure (e.g., single family, industrial, etc.), and a separate fire demand can be assigned to each group. Computations can be carried out for a steady-state simulation or for any time step during an EPS run. This allows direct evaluation of system performance under a wide range of demand loading and operating conditions (e.g., maximum hour, seasonal change).

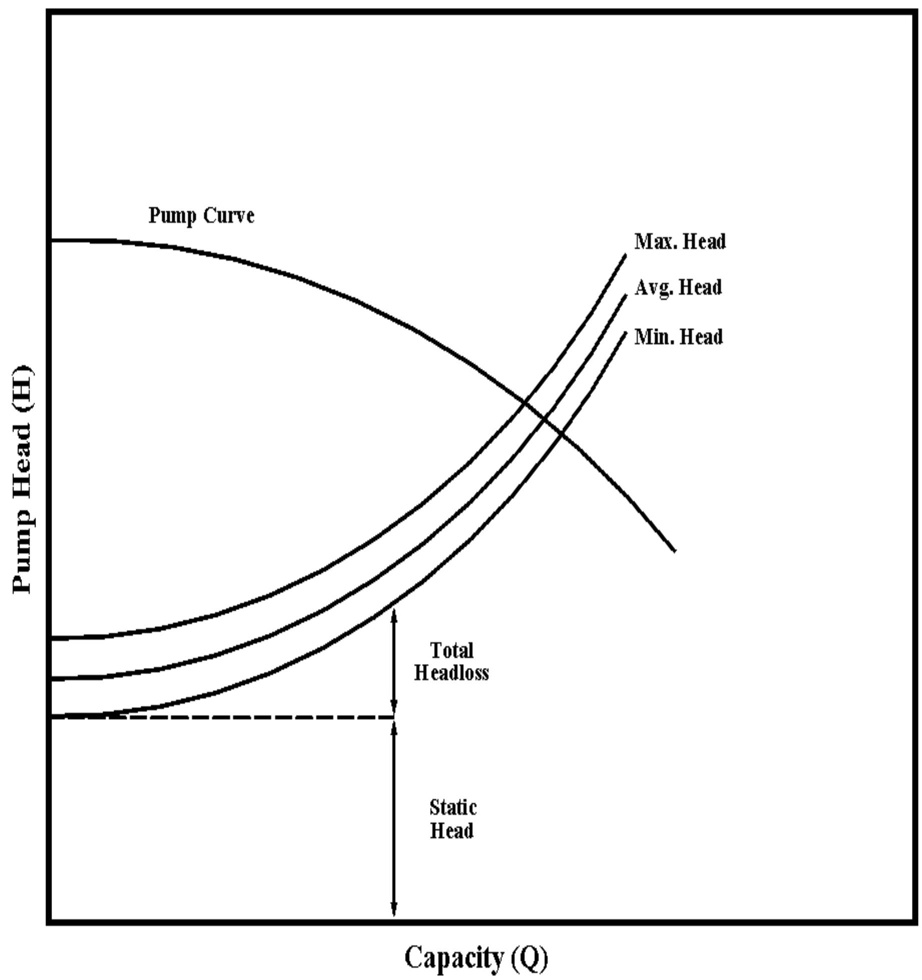
NOTE: The analytical equation used by InfoWater to compute available fire flows will always compute the exact solution. If the static pressure at a junction is less than the specified minimum residual pressure then a negative available flow will be returned.

System Head Curves

The ability to determine system head curves (sometimes called head capacity curves) is an essential task in the design of pumping systems. These curves are useful in determining the rating of the pumps and in assessing the number of pumps required and the type of drive to be used. Each curve describes the relationship between system head and capacity under identified conditions. More specifically, the curve represents the variation in total dynamic head against which the pumps will be required to operate under various flow conditions. The intersection of the system head curve (the resistance the pump must overcome) and the pump characteristic curve (head developed by the pump as a function of capacity) defines the point at which the pump will operate.

To establish typical pump operating conditions, it is useful to represent the total range of pumping conditions with system head curves. A typical system head curve graph is normally constructed to represent three operating conditions: minimum head curve, median head curve, and maximum head curve.

InfoWater automatically generates system head curves for pumps for any flow condition specified. For this application, the pump location is modeled as a junction node and the desired capacity is specified as an inflow (negative demand) at this node. The hydraulic analysis then determines the system head associated with the inflow specified. By specifying a range of flow rates (inflows) and calculating the corresponding system head, the system head curve can be constructed. A system head curve graph is determined by repeating this process for different hydraulic time steps of an EPS run. Each hydraulic time step designates a specific operating condition. Up to three sets of system boundary conditions can be simultaneously specified (e.g., low head, medium head, and high head) and the graphs can be constructed for any junction location in the system. The figure below is an example of system head curves for maximum, average, and minimum heads.



InfoWater computes system head curves in two principal fashions. A system head curve can be described by a curve fit to points of operating data (multi-point curve). Up to 51 data points can be specified for any curve. For this application, system head calculations are performed for each data point over the maximum flow range specified.

Alternatively, InfoWater can represent system head curves using an exponential function of the form:

$$h_s = h_0 + \alpha q^{\beta}$$

where h_s is the system head, h_0 is the system head at zero flow, q is the capacity, α is a resistance coefficient and β is a flow exponent. By supplying InfoWater with the maximum operating flow (q_2), the program is able to directly compute values for α and β from:

$$\beta = \log\left(\frac{h_2 - h_0}{h_1 - h_0}\right) \Big/ \log\left(\frac{q_2}{q_1}\right)$$

$$\alpha = \left(h_1 - h_0\right) / q_1^{\beta}$$

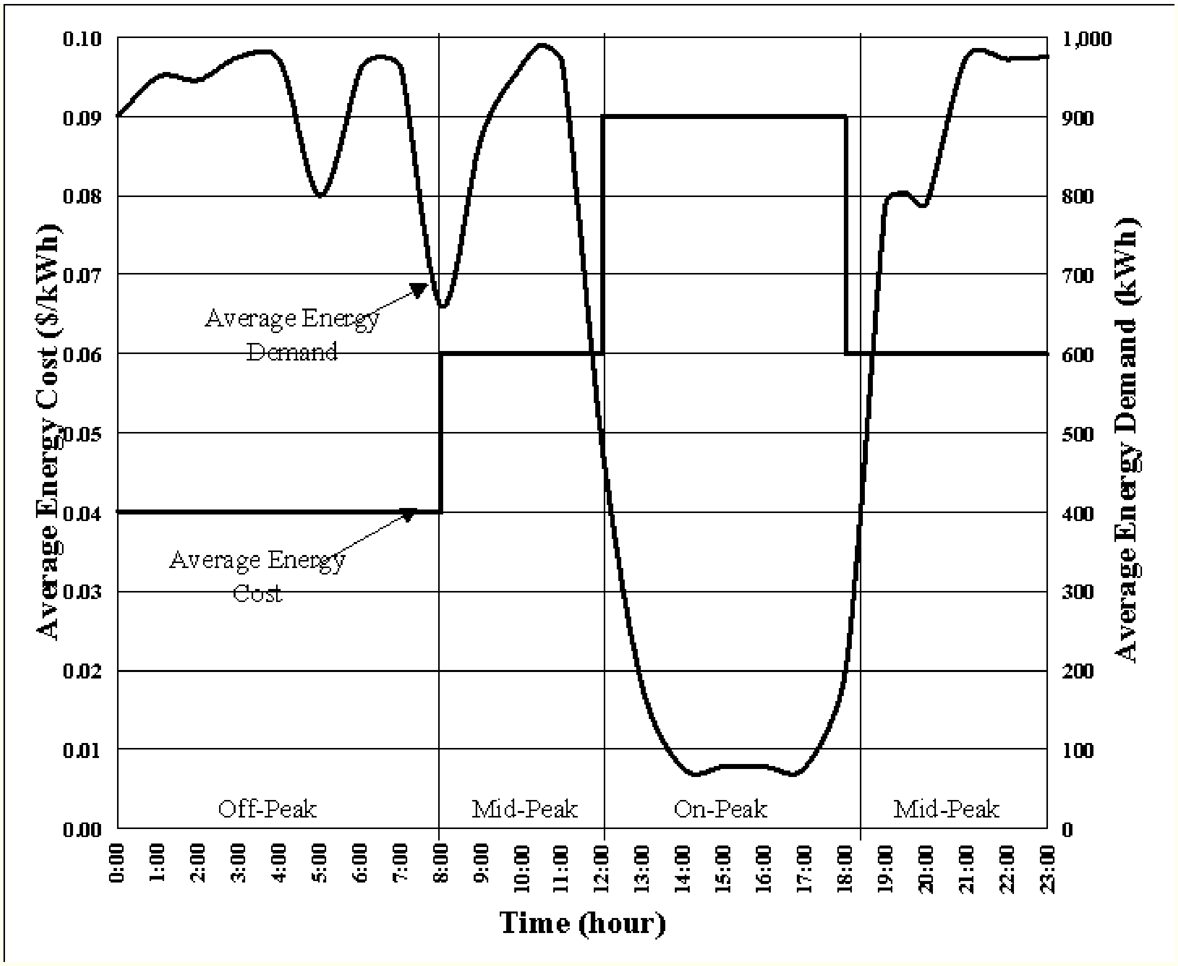
where q_1 is half the maximum flow ($q_2/2$), h_1 is the system head at q_1, and h_2 is the system head at q_2. Only the maximum flow q_2 needs to be specified and all other data points are directly determined by the program. This method of representing system head curve is particularly useful for a preliminary analysis or design.

Energy Management Model

The InfoWater energy management simulator calculates pump energy charges based on user-specified energy charge rates, one-time demand charges for maximum power consumption and pump efficiencies for any given hydraulic conditions within the network. Such calculations can be used to establish energy-efficient and cost-effective pump operation.

The energy charge rate is defined as the charging rate per unit of energy usage ($/kWh). Demand (or capacity) charge is defined as the cost associated with the maximum power consumed within the charging period ($/max. kW). This is a one-time charge due to peak power used during each billing period. Both charging rates are represented as a time-varying pattern (constant or variable rate schedule) for the simulation period. The pump efficiency data describe the relationship between the overall pump efficiency as a function of flow and are expressed as a percentage. The overall pump efficiency is the wire-to-water (total) efficiency, which is the product of the pump efficiency and driver efficiency. This overall efficiency is used in calculating the energy cost of pumping water.

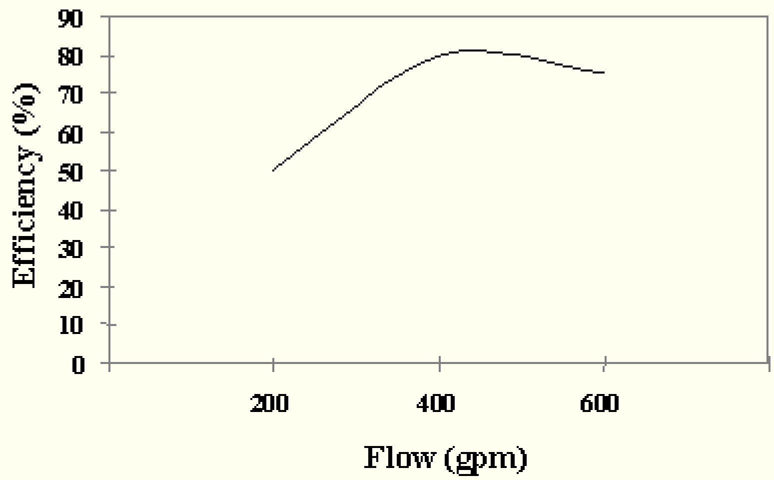
The following figure illustrates an example relationship between average energy cost and energy usage during a 24-hour extended period simulation:



Two options are available for specifying pump efficiency data. A constant efficiency or multiple efficiency-flow operating data points can be specified for any pump. In the latter case, when only three data points are provided, InfoWater fits a quadratic curve to this data of the form: $e = a + bq + cq^2$

where the coefficients *a*, *b*, and *c* are determined by the program. When more than three points of efficiency-flow data are provided, InfoWater uses linear interpolation to calculate pump efficiency.

If the pump is operating outside the flow range specified (above the largest flow data point or below the smallest flow data point), then a constant efficiency is used for all cost calculations in this flow region. This efficiency is set equal to the efficiency associated with the closest operating flow. This gives a realistic description of the pump operation in the extended flow range. The figure below illustrates this situation.



Since pumps can operate at different operating points during the course of a simulation period, InfoWater calculates the power used at each operating point (simulation time step) to determine power cost. At any operating point, the useful power, p_u (hp), can be computed from:

$$p_u = \frac{qh}{8.81}$$

where q is the flow rate (cfs), h is the pump head (ft). The required power consumption, p_c (kW), is determined from:

$$p_c = \frac{p_u}{1.341e}$$

where e is the overall pump efficiency for the operating point in fractional form (<1.0). For each pump, the cost of operation can then be computed for each hydraulic time step of the simulation period.

SCADA Interface

A Supervisory Control and Data Acquisition system (SCADA) stores many information useful for network modeling applications. This system compiles both real-time and historical operational data for local and remote facility sites. Such information is useful to define input data conditions for InfoWater including loading conditions in temporal format for pressure zone and system demands, boundary conditions for reservoir and tank water levels, as well as pipe status and pump and control valve settings. The SCADA output for pressure and flow measurements can also be used in model calibration to verify the reliability of model results. The ability to interface with SCADA systems can offer a number of benefits including at a minimum (Orr et al. 1999):

- Confirmation of normal system performance
- Real-time calibration
- System trouble-shooting
- Projection of operating scenarios
- Performing "what-if’ scenarios
- Training for emergency response
- Testing of case studies
- Improvement of overall operations.

These benefits can only be realized if both systems communicate properly with one another, so that information gathered by the SCADA system can be shared with InfoWater and could be used in simulation runs. The intent is to update InfoWater with the most recent SCADA data. The goal is to monitor as opposed to control (one way communication). The new or most recent SCADA values would replace the existing boundary data in the active InfoWater project.

InfoWater provides the capability to extract pertinent modeling data from SCADA in ASCII format. All communications between InfoWater and the SCADA system are file-based. The ASCII file can reside anywhere on the computer systems, including network path. The key to the successful transfer of information is the format of the file. It must adhere to a specific format so that a correct exchange of information between the two platforms is obtained. The data consist of tank water levels, pipe status, pump status and settings, control valve status and settings, and three demand scaling options. This information is used to update the InfoWater boundary conditions. Recorded pressure and flow measurements can also be imported for comparison (calibration/verification) purpose. Alarm settings for tank levels (minimum/maximum), node pressures (minimum/maximum) and pipe velocities (maximum) can also be specified. The alarm is activated when the modeled results are above or below the maximum or minimum specified settings, respectively.

An initialization phase must first be carried out for proper system configuration and to match SCADA IDs with model IDs (one-to-one match) for any system element, to define the format and units of measurement data, and to optionally specify the lower and upper limits for alarm settings. Configuration data is required only for those network elements whose settings or status are to be updated from the SCADA system and/or a comparison between modeled and measured data is desirable.

It is expected that the SCADA system is able to extract and write the relevant data in an ASCII format readable by InfoWater. The SCADA system must also save this interface file within the computer network system accessible by InfoWater. Upon activation of the SCADA option, InfoWater locates the interface file and updates its network model definition accordingly.

This section summarizes the data information required to update the network model and outlines the interface data file format for SCADA system reference.

Measurement Input File Organization

InfoWater can receive input measurement data from SCADA via an ASCII file. This data is used to update model boundary (tank water levels) and status (pump on/off status) conditions, to define measurement reference for a direct comparison with model results, and to optionally provide system flow measurement to update the overall system demands.

The SCADA Measurement Input file contains eight different sections. Each section begins with a specific keyword in brackets. The keywords and the categories of input data they represent are:

[TIME]

[TANK DATA]

[RESERVOIR DATA]

[PUMP DATA]

[VALVE DATA]

[PRESSURE MEASUREMENTS]

[PIPE FLOW MEASUREMENTS]

[PUMP FLOW MEASUREMENTS]

[VALVE FLOW MEASUREMENTS]

[SYSTEM FLOW MEASUREMENTS]

[END]

Keywords can appear in mixed lower and upper case. The only mandatory sections are **[TIME]** and **[END]**. The order of sections is not important, except for the **[TIME]** and **[END]** sections which must begin and end the input file, respectively. All sections must be separated by at least one blank line. Each section can contain one or more lines of data, however, blank lines are not allowed within a section. Data items can appear in any column of a line, but a line cannot contain more than 80 characters. Data items in any line are separated with a colon (:).

A detailed description of the data in each section of the SCADA Measurement Input file is given below. Optional items are shown in parentheses.

Section [TIME]

Formats:

TIME : *time*

Parameters:

time actual clock (military) time with SCADA readings; this time will be used as the reference time for InfoWater model simulation

Section [TANK DATA]

Formats:

meterID : level

Parameters:

meterID meter reference name from SCADA

level tank water level; InfoWater will update the corresponding tank water level

Section [RESERVOIR DATA]

Formats:

meterID : level

Parameters:

meterID meter reference name from SCADA

level reservoir water level; InfoWater will update the corresponding reservoir water level

Section [PUMP DATA]

Formats:

pumpID : status

pumpID : setting (speed)

Parameters:

pumpID meter reference name from SCADA

status On, True, Up, 1 - if pump is open

Off, False, Down, 0 - if pump is closed

setting measured flow value

speed optional pump speed value

Remarks:

The first format is used for pumps specified by ON/OFF status while the second format is used for pumps recording flow values. InfoWater will update all corresponding pump operating status and settings accordingly.

Section [VALVE DATA]

Formats:

valveID : status

valveID : setting

Parameters:

valveID meter reference name from SCADA

status Open - if valve is open (valve acts as an open pipe)

Closed - if valve is closed

setting measured valve setting for active valves (pressure for PRVs, PSVs, and PBVs; flow for FCVs; loss coefficient for TCVs and degree of opening for MTVs).

Remarks:

Only PRVs, PSVs, PBVs, FCVs, TCVs and MTVs are supported. InfoWater will update all corresponding values specified for the valve operating status and setting accordingly.

Section [PRESSURE MEASUREMENTS]

Formats:

meterID : *setting*

Parameters:

meterID meter reference name from SCADA

setting measured junction node pressure

Remarks:

This data will be used by InfoWater to provide a direct comparison between measured and modeled pressures at the selected junction nodes.

Section [PIPE FLOW MEASUREMENTS]

Formats:

meterID : *setting*

Parameters:

meterID meter reference name from SCADA

setting measured pipe flow value

Remarks:

Pipe flow readings are used by InfoWater to provide a direct comparison between measured and modeled flows for the selected pipes.

Section [PUMP FLOW MEASUREMENTS]

Formats:

meterID : *setting*

Parameters:

meterID meter reference name from SCADA

setting measured pump flow value

Remarks:

Pump flow readings are used by InfoWater to provide a direct comparison between measured and modeled flows for the selected pumps.

Section [VALVE FLOW MEASUREMENTS]

Formats:

meterID : *setting*

Parameters:

meterID meter reference name from SCADA

setting measured control valve flow value

Remarks:

Control valve flow readings are used by InfoWater to provide a direct comparison between measured and modeled flows for the selected control valves.

Section [SYSTEM FLOW MEASUREMENTS]

Formats:

meterID : *setting*

Parameters:

meterID meter reference name from SCADA

setting measured inflow and outflow value (in flow unit)

Remarks:

System inflow and outflow readings are used by InfoWater to update the overall system demands. This option applies only when the %Inflow/Outflow Demand Adjustment Method is selected from the InfoWater Run Manager SCADA Tab.

Example SCADA Measurement Input File:

[Time]

Time : 18:00

[Tank Data]

tank2 : 90.0

[Pump Data]

pump9 : 1100.0 950.0

[Pressure Measurements]

junction11 : 125.0

junction23 : 90.0

[Pipe Flow Measurements]

pipe10 : 450.0

pipe12 : 612.0

pipe122 : 135.0

[End]

References

Altman, T. and Boulos, P.F. (1995). "Convergence of Newton Method in Nonlinear Network Analysis." *Journal Mathematical and Computer Modelling*, 21(4), pp. 35-41.

American Water Works Association (1989). *Distribution System Requirements for Fire Protection*. Manual M31. Denver, CO.

Boulos, P.F., Altman, T., Jarrige, P.A., and Collevati, F. (1995). "Discrete Simulation Approach for Network Water Quality Models." *Journal Water Resources Planning and Management* ASCE, 121(1), pp. 49-60.

Boulos, P.F., et al. (1997). "Automated Computation of Fire Flows with Network Models." *Journal American Water Works Association*, 89(2), pp. 51-56.

Boulos, P.F., et al. (1998). "Application of Rule-Based Logical Controls in Network Hydraulic Models." In *Proceedings of the AWWA Annual Conference*, Dallas, Texas, June.

Boulos, P.F., Mau, R.E., and Clark, R.M. (1998). "Analytical Equations of Storage Reservoir Water Quality. " *Journal Civil Engineering and Environmental Systems*, 15, pp. 171-186.

Dunlop, E.J. (1991). *WADI Users Manual*. Local Government Computer Services Board, Dublin, Ireland.

Edwards, D.K., Denny, V.E., and Mills, A.F. (1976). *Transfer Processes*. McGraw-Hill, New York, NY.

Fire Suppression Rating Schedule (1980). Insurance Services Offices, New York, NY.

George, A. and Liu, J. (1981). *Computer Solution of Large Sparse Positive Definite Systems*. Prentice-Hall, Inc., Englewood Cliffs, NJ.

Grayman, W.M., et al. (1996). "Water Quality and Mixing Models for Tanks and Reservoirs." *Journal American Water Works Association*, 88(7), pp. 60-73.

Koechling, M.T. (1998). *Assessment and Modeling of Chlorine Reactions with Natural Organic Matter: Impact of Source Water Quality and Reaction Conditions*, Ph.D. Thesis, Department of Civil and Environmental Engineering, University of Cincinnati, Cincinnati, Ohio.

Hamam, Y.M. and Brameller, A. (1971). "Hybrid Method for the Solution of Piping Networks." In *Proceedings IEE*, 118(11), pp. 1607-1612.

Hannoun, I.A. and Boulos, P.F. (1997). "Optimizing Distribution Storage Water Quality: A Hydrodynamic Approach." *Journal Applied Mathematical Modelling*, 21, pp. 495-502.

Hannoun, I.A., Boulos, P.F., and List, E.J. (1998). "Using Hydraulic Modeling for CT Compliance." *Journal American Water Works Association*, 90(8), pp. 77-87.

Mau, R.E., et al. (1995). "Explicit Mathematical Models of Distribution Storage Water Quality." *Journal Hydraulic Engineering ASCE*, 121(10), pp. 699-709.

Mau, R.E., Boulos, P.F. and Bowcock, R. (1996). "Modeling Distribution Storage Water Quality: An Analytical Approach." *Journal Applied Mathematical Modelling*, 20, pp. 329-338.

Notter, R.H. and Sleicher, C.A. (1971). "The Eddy Diffusivity in the Turbulent Boundary Layer Near a Wall", *Chem. Eng. Sci.,* Vol. 26, pp. 161-171.

Orr, C.H, Boulos, P.F., Stern, C. and Liu, P. (1999). "Developing Real-Time Models of Water Distribution Systems". Water Industry Systems: modeling and optimization applications, eds. D. Savic and G. Walters, Vol. 1, pp. 175-185.

Osiadacz, A.J. (1991). *Simulation and Analysis of Gas Networks*. E&F Spon Ltd., London, UK.

Rossman, L.A., Boulos, P.F., and Altman, T. (1993). "Discrete Volume Element Method for Network Water Quality Models." *Journal Water Resources Planning and Management* ASCE, 119(5), pp. 505-517.

Rossman, L.A. and Boulos, P.F. (1996). "Numerical Methods for Modeling Water Quality in Distribution Systems: A Comparison." *Journal Water Resources Planning and Management* ASCE, 122(2), pp. 137-146.

Rossman, L.A., Clark, R.M., and Grayman, W.M. (1994). "Modeling Chlorine Residuals in Drinking-Water Distribution Systems." *Journal Environmental Engineering* ASCE, 120(4), pp. 803-820.

Rossman, L.A. (2000). *EPANET 2 Users Manual*, EPA-600/R-94/057, Environmental Protection Agency, Risk Reduction Engineering Laboratory, Cincinnati, OH.

Todini, E. and Pilati, S. (1987). "A Gradient Method for the Analysis of Pipe Networks." In *Proceedings of the International Conference on Computer Applications for Water Supply and Distribution*, Leicester Polytechnic, UK, September 8-10.

Vasconcelos, J.J., et al. (1996). *Characterization and Modeling of Chlorine Decay in Distribution Systems.* American Water Works Association Research Foundation, Denver, CO.

Vasconcelos, J.J., et al. (1997). "Kinetics of Chlorine Decay." *Journal American Water Works Association*, 89(7), 54-65.

Additional References:

Altman, T. and Boulos, P.F. (1995). "On Solving Flow Constrained Networks: The Inverse Problem." *Journal Hydraulic Engineering ASCE*, 121(5), 427-431.

Bhave, P.R. (1991). *Analysis of Flow in Water Distribution Networks*. Technomic Publishing Co., New York, NY.

Boulos, P.F. and Altman, T. (1991). "A Graph-Theoretic Approach to Explicit Nonlinear Pipe Network Optimization." *Journal Applied Mathematical Modelling*, 15, pp. 459-486.

Boulos, P.F. and Altman, T. (1993). "Explicit Calculation of Water Quality Parameters in Pipe Distribution Networks." *Journal Civil Engineering Systems*, 10, pp. 187-206.

Boulos, P.F. and Altman, T. (1993). "An Explicit Approach for Modeling Closed Pipes in Water Networks." *Journal Applied Mathematical Modelling*, 17, pp. 437-443.

Boulos, P.F., et al. (1994). "An Explicit Algorithm for Modeling Distribution System Water Quality with Applications." In *2nd BHR International Conference on Water Pipeline Systems*, D.S. Miller ed., Edinburgh, Scotland, pp. 405-423.

Boulos, P.F., Altman, T., and Liou, C.P.J. (1993). "On the Solvability of Water Distribution Networks With Unknown Pipe Characteristics." *Journal Applied Mathematical Modelling*, 17, pp. 380-387.

Boulos, P.F., Altman, T., Jarrige, P.A., and Collevati, F. (1994). "An Event-Driven Method for Modeling Contaminant Propagation in Water Networks." *Journal Applied Mathematical Modelling*, 18, pp. 84-92.

Boulos, P.F., Altman, T., and Sadhal, K.S. (1992). "Computer Modelling of Water Quality in Large Multiple Source Networks." *Journal Applied Mathematical Modelling*, 16, pp. 439-445.

Boulos, P.F. and Ormsbee, L.E. (1991)."Explicit Network Calibration for Multiple Loading Conditions." *Journal Civil Engineering Systems,* 8, pp. 153-159.

Boulos, P.F., Rossman, L.A., and Vasconcelos, J.J. (1994). "A Comparison of Methods for Modeling Water Quality in Distribution Systems." In *Proceedings of the AWWA 1994 Annual Conference,* New York, New York, pp. 523-541.

Boulos, P.F. and Wood, D.J. (1990). "Explicit Calculation of Pipe-Network Parameters". *Journal Hydraulic Engineering* ASCE, 116(11), pp. 1329-1344.

Boulos, P.F. and Wood, D.J. (1991). "An Explicit Algorithm for Calculating Operating Parameters for Water Networks." *Journal Civil Engineering Systems,* 8, pp. 115-122.

Boulos, P.F., et al. (1996). "Hydraulic Mixing and Free Chlorine Residual in Reservoirs." *Journal American Water Works Association*, 88(7), pp. 48-59.

Boulos, P.F., et al. (1997). "Water Distribution Network Management: A Fully Integrated Approach." *Journal New England Water Works Association*, 111(2), pp. 180-188.

Boulos, P.F., et al. (1997). "Complete Automation of Water Distribution Facilities Management." In *Proceedings of the AWWA IMTech Conference*, Austin, Texas, pp. 73-94.

Cesario, A.L. (1995). *Modeling, Analysis, and Design of Water Distribution Systems*. American Water Works Association, Denver, CO.

Clark, R.M., et al. (1994). "Managing Water Quality in Distribution Systems: Simulating TTHM and Chlorine Residual Propagation." *Journal Water SRT-AQUA*, 43(4), pp. 182-191.

Clark, R.M., et al. (1996). "Mixing in Distribution System Storage Tanks: Its Effect on Water Quality." *Journal Environmental*

Engineering ASCE, 122(9), pp. 814-821.

Heath, J.E., et al. (1997). "Automated Calculation and Distribution of Water Demands in Distribution Network Models." In *Proceedings of the AWWA IMTech Conference*, Austin, Texas, pp. 219-231.

Jeppson, R.W. (1981). *Analysis of Flow in Pipe Networks*. 5th printing, Ann Arbor Science, Ann Arbor, MI.

Mau, R.E., et al. (1996). "Advanced Network Modeling Applications: Dynamic Design and SCADA Interface." In *Proceedings of the AWWA Engineering and Construction Conference*, Denver, Colorado, pp. 223-250.

Walski, T.M. (1984). *Analysis of Water Distribution Systems*. Van Nostrand Reinhold Co., New York, NY.

Create New InfoWater Project

When creating a new InfoWater project you can define the project's coordinate system here. InfoWater provides the following tools to specify a project's coordinate system:

ArcGIS Default - Use the Infowater default coordinate system. This default spatial reference system follows ArcGIS's default settings. It's extent (or XY domain) is:

Minimum X: -10000

Maximum X: 11474.83645

Minimum Y: -10000

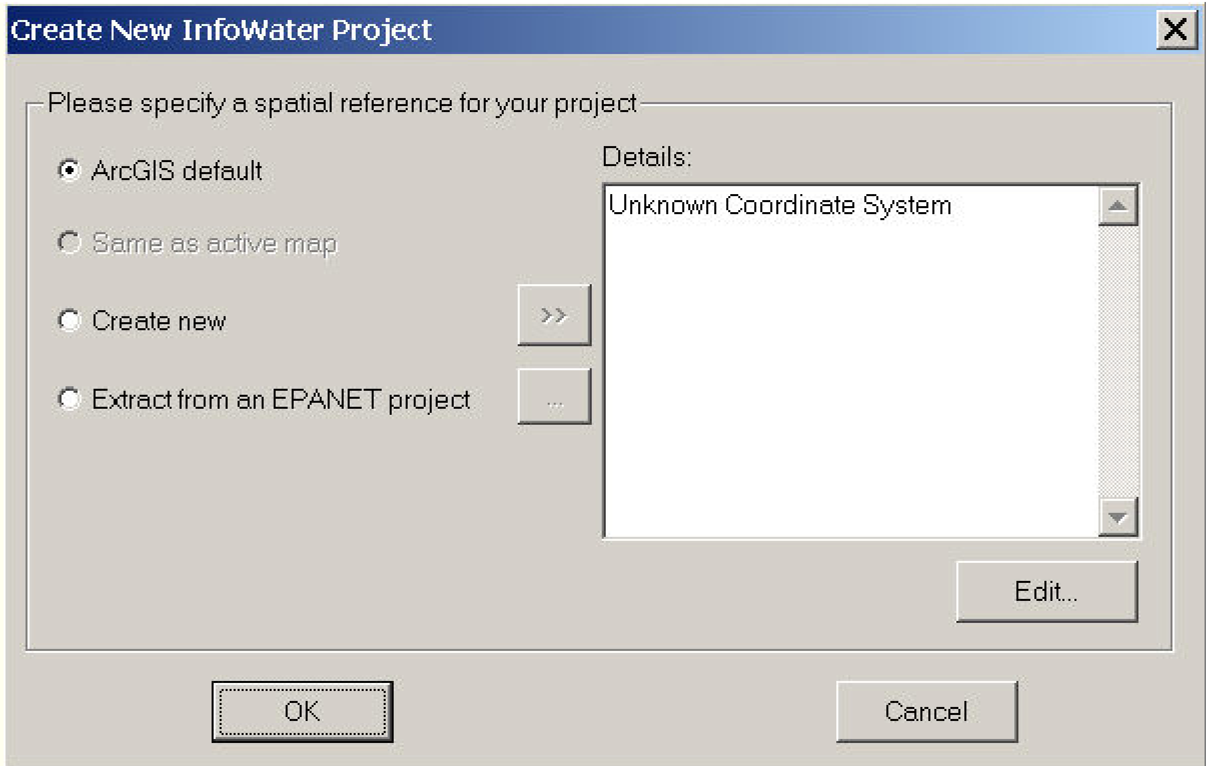
Maximum Y: 11474.83645

The Same as the Active Map - Use this to assign the presently active map's background layer's coordinate system. This option is enabled only if you have a background layer associated with your project.

Create New - Use this to either define a new custom coordinate system or use an existing available coordinate system from the ArcGIS library. Click here to learn more.

Extract from an EPANET project - Use this to extract a coordinate system from an existing EPANET project. In addition you can import EPANET v2.x data files using this option. Click here to learn more about the importing process.

Click on any section below to learn more:



Other Related Topics - Change Project Spatial Reference, Degree of Resolution, New Spatial Reference Dialog box, Spatial System Coordinate Range

---

Output Report Methodology

The Output Report manager provides you with a means to access all your Output variables in report and graph form for all your simulations.

Methodology

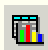
Reports

To access Output Reports do the following:

- Launch the InfoWater Control Center toolbar from the ArcMap View menu -> Toolbars command.
- Select Output Report Manager from the InfoWater button -> Tools sub menu. Alternatively click on the Output Report Manger icon on the InfoWater Control Center to launch the Output Report Manager.
- Click on New to launch the Output Report & Graph dialog box.
- Choose the Output Source from the list displayed under the Available Output Sources section. Only after the particular simulation has been run will the Output Source appear.
- Select the Report Type you want to see and click on Open at the bottom of the dialog box to launch the Output Report Manager to display the output results in report format. Click here to view all the different available Report types.

Graphs

To access Output Graphs do the following:

- Launch the InfoWater Control Center toolbar from the ArcMap View menu -> Toolbars command.
- Select Output Report Manager from the InfoWater button -> Tools sub menu. Alternatively click on the Output Report Manger icon  on the InfoWater Control Center to launch the Output Report Manager.
- Click on New to launch the Output Report & Graph

  dialog box.
- Click on the Graph Tab to display all the Available Output Sources and the different graph types available.
- Choose the Output Source from the list displayed under the Available Output Sources section. Only after the particular simulation has been run will the Output Source appear.
- Select the Graph Type you want to see and click on Open at the bottom of the dialog box to launch the Output Report Manager to display the output results in graph format.

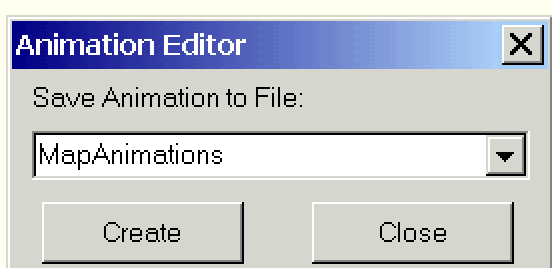
Animation Viewer Methodology

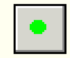
The Animation Viewer is used to retrieve and display map animations created and saved through the Animation Editor found under the **InfoWater Control Center -> InfoWater** button **-> Tools -> Animation Editor**. Map animations display simulation results for EPS timesteps in succession, thereby creating a "movie" effect that allows the user to see saved and view simulation results at any point in the future. Map Animations can be created only after a successful **EPS** run.

Methodology

To create a New Animation do the following:

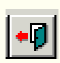
- Create and run an Extended Period Simulation. Click here to learn more about running Extended simulations.
- Launch the Map Display manager by clicking on the **Map Display** icon  on your **InfoWater Control Center**. Specify your map display options (the Data Source and Data Field and the color ranges). Refer to the section on Map Display Methodology to learn more about customizing your map display.
- Launch the **Animation Editor** dialog box from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu -> **Animation Editor** command. Specify a name for the animation as shown below and click on the **Create** button.



- Launch the **InfoWater Output** tool bar from your **View** menu **-> Toolbars** command and click on the **Output Status** icon . This will record your animations to your specified file (in this case MapAnimations).

- Once you have successfully recorded your animations launch the **Animation Viewer** from your **InfoWater Control Center -> InfoWater** button **-> Tools** menu. Select the File that you want to view at the bottom your **Animation Viewer** dialog box.

- Choose the time steps and evaluate your animations. You may choose to copy, go to the first slide, fast backward, go backward, go forward, fast forward or view the last animation slide.

- Finally exit from the **Animation Viewer** dialog box by clicking on the **Exit** icon .

Other Related Topics - Animation Editor**,** Animation Viewer**,** Elevation Profile**,** Hydraulic Calculator

---

Map Display Methodology

Map Display may be used to either color code and/or vary symbol sizes of your data elements such as Junctions, Pumps, Valves, Tanks , Reservoirs and Pipes based on either their input or output variables. You may also annotate your InfoWater map with the actual data value. You can choose the data breaks, the different colors and the symbol sizes. This provides a very dynamic and powerful way of categorizing your entire project visually based on your defined criteria.

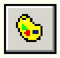
Using Map Display in InfoWater

Some of the different uses of Map Display are as follows.

- Color Coding and/or using different symbol sizes for your nodes based on elevations will give you a quick idea of the topology of your area. You may use this feature to help in identifying zonal separations.
- Color coding and/or using different symbol sizes for your pipes based on diameters for instance will provide you with a time saving tool to determine the correctness of your digitization process. You may also gauge the water levels in your tanks and reservoirs by color coding and/or using different symbol sizes for them based on water level.
- Color coding and/or using different symbol sizes for your elements with the output data will provide a quick means of visually gauging the correctness of your model simulation. Junction pressure color coding will right away tell you about your system deficiencies and inadequacies.

Methodology

To color code and/or use different symbol sizes for your data elements using the InfoWater Map rendering feature do the following

- Launch the **Map Display Manager** by clicking on the **Map Display**  icon on your InfoWater **Control Center**.
- Choose the **Element Type** and the **Data Source** for the annotation.
- Specify the **Data Field** that you want to use to annotate your map with and the number of classes. Alternatively for advanced classification options click on the **Classify** button.
- Choose the **Rendering Method** that you want to use. You may select between annotating your data element based on color or based on symbol size.
- Specify the data breaks and either manually specify the colors and the symbol size breaks or use the **From - To** option and the Ramp Color button to automatically assign the symbol sizes and the colors to the data elements.
- You may also add text to your annotation. Choose the Label Properties tab if you want to annotate your model with labels. Specify the different options available including the placement and the size of the label. Also specify if you want the units displayed on the map. To learn more about the different options available to annotate your map with labels click here.
- Once you have entered all the Map Rendering data to your satisfaction click on **OK** to save, apply and exit from the Map Display tab. Choose to click on **Apply** if you want to keep the Map Display Manager open and yet annotate your Map at the same time.

- Finally choose the **Reset Display** icon  to clear your map.

Note: Color Coding will **NOT** work for a domain. InfoWater will always color code the **ENTIRE** network.

Other Related Topics - Map Display - Label Properties, Map Display Classification, Map Display Main Dialog Box

---

Annotation

The annotation feature of InfoWater allows you to annotate your InfoWater project with data values. Such annotations can consist of any input or output data as provided in the InfoWater project and model databases.

Methodology

To annotate your InfoWater project do the following

- Launch the **Table of Contents** dialog box from the InfoWater **View** menu.
- Choose the **Operation Data** tab of your InfoWater **Table of Contents** dialog box by clicking on it and then right click and choose the **New** command.
- Suggest an ID and a Description for your Annotation. This will launch your Annotation dialog box.
- In your Annotation dialog box, choose the element, the **Data Source** and the **Data category**. Also specify the **Scope** and the simulation **Time** for EPS models.
- Choose the text siize and the number of decimal places that you want to see. You may also specify a layer that you want to contain this data.
- Then specify your **Data Anchor** options and then click on **Create** to create your Annotations.
- Choose **Clear Layer** at any time to clear the InfoWater layer or you may choose to hide the layer if you want to save the annotations and retrieve it at a later time.

Other Related Topics - Annotation Dialog Box**,** Contour-Labelling**,** Contour-Level**,** Contour Dialog Box**,** Contour Methodology**,** Contour Options**,** Operation Data**,** Table of Contents

---

Contour

The Contour section of the **Table of Contents** is used to generate, edit and delete contours. Contours can be displayed at any time before or after a simulation run and are saved as a permanent part of the InfoWater projects in layers. Contours can be generated for any numeric **Junction**-based model input field or simulation result variable

Methodology

To create Contours in your InfoWater project do the following

- Run a successful hydraulic simulation in InfoWater if you want to contour result variables.
- launch the InfoWater **Table of Contents** from the **View** menu.
- Choose the **Contour** tab of your InfoWater **Table of Contents** by clicking on it and then right click and choose the **New** command.
- Suggest an ID and a Description for your Contour. This will launch your Contour dialog box.
- In your Contour dialog box, Source tab choose the **Data Source** and the **Data Category**. Also specify the Resolution, the **Smooth Method** and simulation **Time** for EPS models
- Click on the Level tab and specify the contour interval and colors for your InfoWater contours.
- Finally specify your label options by clicking on the Label tab. Here you may specify the different label settings including the size, font, color and placement of your contour labels.
- Once you have specified all your Contour options click on the **OK** button at the bottom of the Contour dialog box to create your Contour.

Other Related Topics - Annotation Dialog Box, Annotation Methodology, Contour-Labelling, Contour-Level, Contour Dialog Box, Contour Options, Operation Data, Table of Contents

---

Working with ESRI® Geodatabases

InfoWater allows you to work with both Personal Geodatabases as well as SDE Geodatabases served by enterprise ArcSDE servers. To import/export data from/to Geodatabases use the **GIS Gateway** tool (**InfoWater Control Center -> InfoWater** button **-> Exchange** pull down menu **-> GIS Gateway** command). Click here to learn more about Geodatabases.

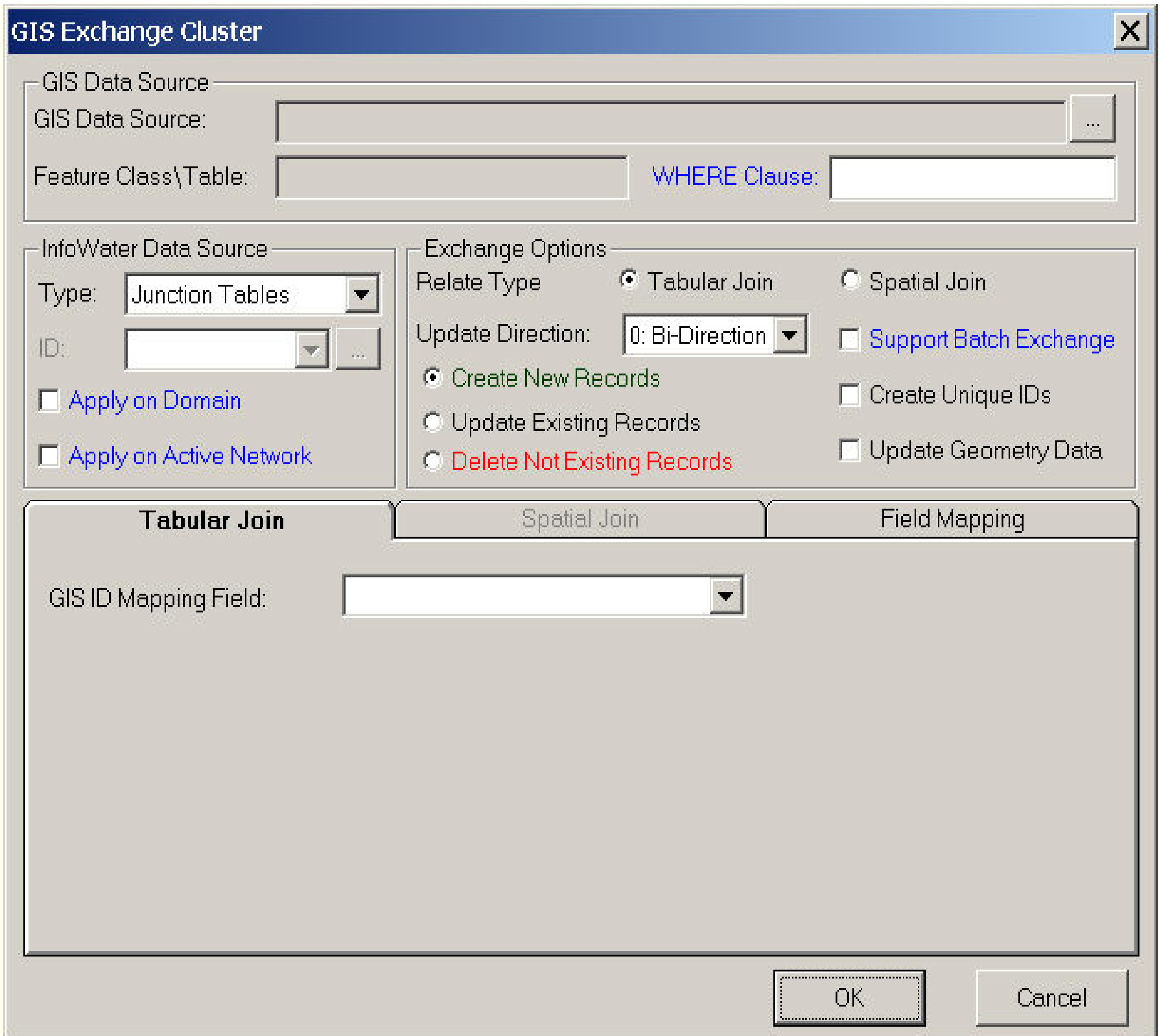
GIS Gateway utilizes GIS Exchange Clusters to perform data exchange with various GIS data sources supported by ESRI including personal and enterprise Geodatabases. A GIS Exchange Cluster defines a linkage toward the desired GIS data source and a set of data exchange parameters that allows the user to quickly load and save data when the cluster is applied.

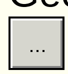
Define a GIS Exchange Cluster

For defining a new GIS Exchange Cluster for the desired Geodatabase do the following:

- From the **InfoWater Control Center -> InfoWater button -> Exchange** pull-down menu, choose the **GIS Gateway** command. This launches the GIS Gateway dialog box.
- Click on the **Add** button on the **GIS Gateway** dialog box and the **Identification** dialog box appears as shown below. Enter the new GIS Cluster ID and description as shown below.

- Click on **OK** to launch the **GIS Exchange Cluster** dialog box as shown below.

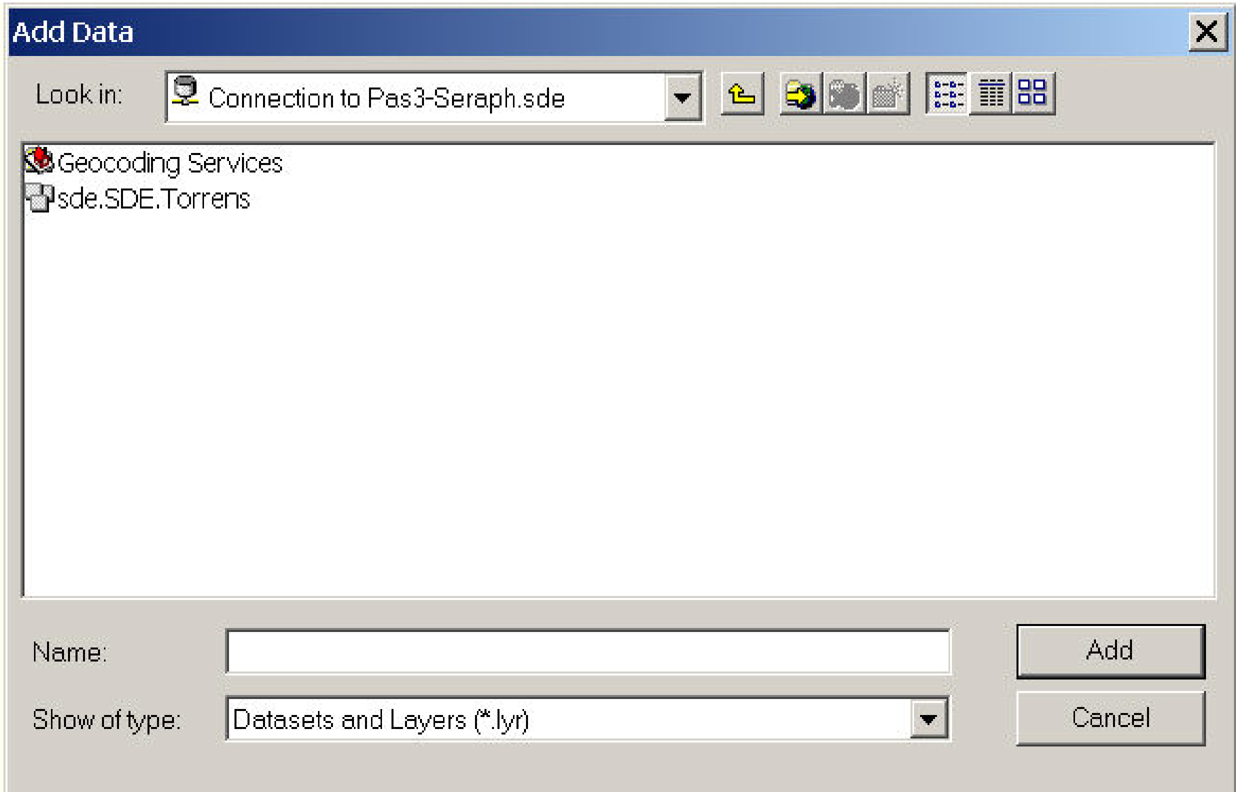


- On the **GIS Exchange Cluster** dialog box specify the GIS Data Source to connect the desired Geodatabase to your project by clicking on the **Browse** button 

 next to the 

 field.
- Locate your Geodatabase using the **Select a Table or Layer** dialog box and click on the **Add** button 

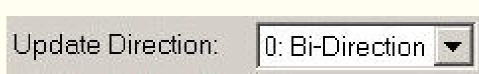
 as shown below.



- From the **GIS Exchange Cluster** dialog box, select the appropriate Exchange type form the **InfoWater Data Source Type:** InfoWater Data Source section of the dialog box. InfoWater provides 6 different data element types including Junctions, Tanks, Reservoirs, Pumps, Valves and Pipes as well as any other model attribute.

- Specify the **Relate Type** in the Relate Type section of the **GIS Exchange Cluster** dialog box. You may choose either Tabular Join or Spatial Join depending on the type of join that you want to create.

- Choose the appropriate Update Direction. You may choose *0: Bi-Direction*, *1: Load Only* or *2: Save Only* as shown below. To load data from Geodatabase choose either *0: Bi-Direction* or *1: Load Only* option. To update Geodatabase choose either *0: Bi-Direction* or *2: Save Only* option.



- Depending on the **Relate Type** that you specified choose either the Tabular Join or the Spatial Join tab.

- For Tabular join, specify the GIS ID Mapping Field. InfoWater will assume that this field contains the unique element IDs for hydraulic modeling.

- Choose **Create New Records** or **Update Existing Records** to create or update InfoWater data.

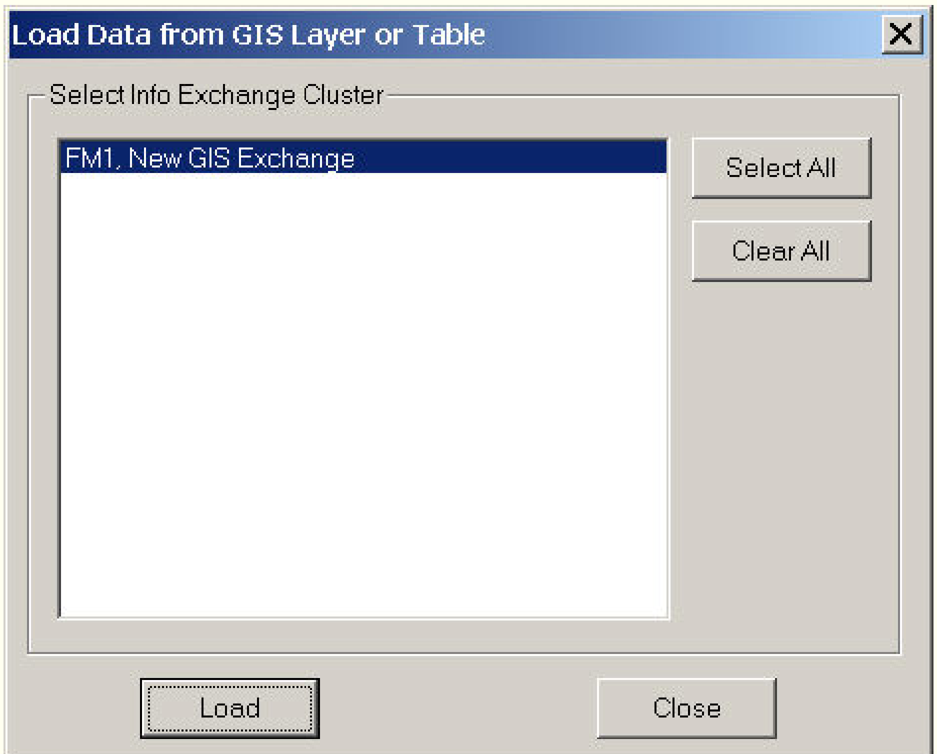
- Depending on the Geodatabase that you are loading you can choose to check the Update Geometry Data option. If this option is checked then InfoWater will update the element geometry as well. Elements will then be updated with the new coordinate geometry.

- Click on the Field Mapping tab and map the appropriate GIS data fields with the InfoWater fields and finally click on **OK**. This will create a new GIS Exchange Cluster definition.

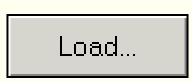
Load Data from Geodatabases

You can update InfoWater data directly from your Geodatabases. Once the GIS Exchange Clusters have been defined and linked with the desired Geodatabases, click on the **Load** button 

on your **GIS Gateway** dialog box to load data from the desired Geodatabases.



For **Loading Geodatabases** perform the following:

- Launch the **GIS Gateway** dialog box from the **InfoWater Control Center -> InfoWater** button **-> Exchange** pull down menu.
- Click on the **Load** button 

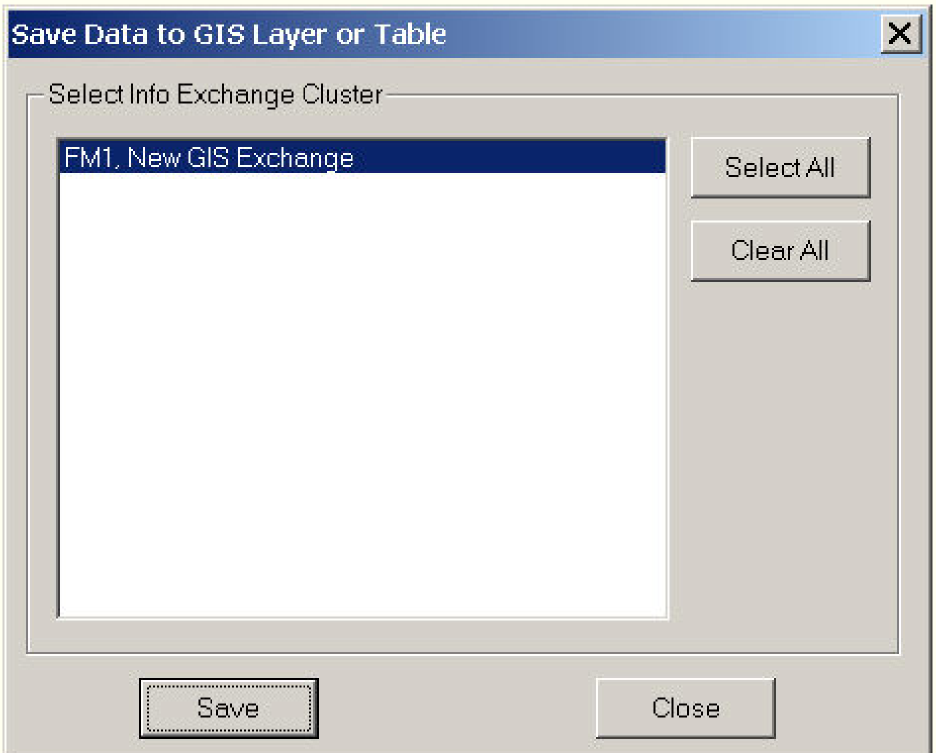
to launch the **Load Data from GIS Layer or Table** dialog box and select the desired GIS Exchange Cluster(s). Click on the **Load** button (from the **Load**

Data to GIS Layer or Table dialog box) at the bottom to read data from the specified Geodatabases.

Save Data to Geodatabases

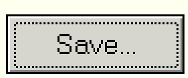
You can use InfoWater to make any desired changes to your Geodatabases. Once the GIS Exchange Clusters have been defined and linked with the desired Geodatabase, click on the **Save** button 

from **GIS Gateway** dialog box to update the desired Geodatabases.



For **saving Geodatabases** perform the following:

- Launch the **GIS Gateway** dialog box from the **InfoWater Control Center -> InfoWater** button **-> Exchange** pull down menu.

- Click on the **Save** button 

to launch the **Save Data to GIS Layer or Table** dialog box and select the desired GIS Exchange Cluster(s). Click on the **Save** button (from the **Save Data to GIS Layer or Table** dialog box) at the bottom to update the specified Geodatabases.

Note: You can only save to a geodatabase that has been loaded in InfoWater.

Other Related Topics - Geodatabase Methodology**,** Geodatabase Theory**,** GIS Cluster -Spatial Join**,** GIS Exchange Cluster**,** GIS Exchange Cluster Tabular Join**,** GIS Field Mapping**,** GIS Gateway

Warning Messages

The following is a listing on non-fatal warning messages that might be encountered during a successful analysis run. When these warnings occur, the Run Status Indicator (stop-light) on the Run Manager dialog box indicates yellow.

Format:

Warning Message Suggested Action

- **Pump cannot deliver head**. Use a pump with a larger shutoff head.
- **Pump cannot deliver flow.** Use a pump with a larger flow capacity.
- **Flow control valve cannot deliver flow.** Reduce the flow setting on the valve or provide additional head at the valve.
- **Following nodes are disconnected at timestep [timestep]: [list of nodes]** One or more pipes are closed at the given timestep, therefore shutting off all flow to a portion of the network. Ensure that this action is appropriate and if not, change the controls or status associated with the pipes and re-run the analysis. It should be noted that not all pipes connecting a junction node can be closed simultaneously. Each junction node must have at least one connecting pipe open. For example, a pipe connecting a dead-end junction node cannot be closed. InfoWater will identify all junction nodes that are disconnected and this data must be corrected before an analysis can be made.
- **System unbalanced.** This condition typically occurs when a pump or valve keeps switching its status back and forth between

successive iterations of solving the hydraulic equations for the network, thus causing the system to fail to converge to a solution. Possible causes for this include a pair of pressure controls that turn a pump on and off whose pressure settings are too close together, or a collection of pressure regulating valves whose pressure settings influence the status of one another. You can specify a FULL system status output report with the Edit Report Option command to identify those pipes that might be behaving in this manner and then modify their pressure or flow settings to avoid this situation. Another option would be to use a slightly larger value for the model ACCURACY parameter, which can be set using the Edit Simulation Options command.

- **Negative Pressures exist at Node(s)**. InfoWater will issue a warning message when it encounters negative pressures at junctions that have positive demands. This usually indicates that there is some problem with the way the network has been designed or operated. Negative pressures can occur when portions of the network can only receive water through pipes that have been closed off. In such cases an additional warning message about the network being disconnected is also issued.

Error Codes & Messages

The following is a list of the InfoWater network simulator error messages and suggested solutions. When one or more of these errors occur, the Run Status Indicator (stop-light) on the Run Manager dialog box indicates red. It is possible for several of these conditions to exist at the same time.

Format

Error Number Description, **Suggestion for Solution**

101 - An analysis was terminated due to insufficient memory available. May be caused by the hard disk drive being full. Clear as much hard disk space as possible and re-run the analysis.

110 - An analysis was terminated because the network hydraulic equations could not be solved. Check for portions of the network not having any physical links back to a tank or reservoir or for unreasonable values for network input data.

200 - One or more errors were detected in the input data. The nature of the error will be described by the 200-series error messages listed below.

201 - There is a syntax error in a line of the input file created from your network data. This is most likely to have occurred in the input text created by a user outside of InfoWater while importing an EPANET .INP file.

202 - An illegal numeric value was assigned to a property.

203 - An object refers to undefined node.

204 - An object refers to an undefined pipe.

205 - An object refers to an undefined time pattern.

206 - An object refers to an undefined curve.

207 - An attempt is made to control a check valve. Once a pipe is assigned a Check Valve status, its status cannot be changed by either simple or rule-based controls.

208 - Reference was made to an undefined node. This could occur in a control statement for example.

209 - An illegal value was assigned to a node property.

210 - Reference was made to an undefined pipe. This could occur in a control statement for example.

211 - An illegal value was assigned to a pipe property.

212 - A source tracing analysis refers to an undefined trace node.

213 - An analysis option has an illegal value (an example would be a negative time step value).

214 - There are too many characters in a line read from an input file. The lines in the input file are limited to 255 characters. This is most likely to have occurred in the input text created by a user outside of InfoWater while importing an EPANET .INP file

215 - Two or more nodes or pipes share the same ID label.

216 - Energy data were supplied for an undefined pump.

217 - Invalid energy data were supplied for a pump.

221 - A rule-based control contains a misplaced clause.

223 - There are not enough nodes in the network to analyze. A valid network must contain at least one tank/reservoir and one junction node.

224 - There is not at least one tank or reservoir in the network.

225 - Invalid lower/upper levels were specified for a tank (e.g., the lower lever is higher than the upper level).

226 - No pump curve or power rating was supplied for a pump. A pump must either be assigned a curve ID in its Pump Curve property or a power rating in its Power property. If both properties are assigned then the Pump Curve is used.

227 - A pump has an invalid pump curve. A valid pump curve must have decreasing head with increasing flow.

230 - A curve has non-increasing X-values.

233 A node is not connected to any pipes.

302 - The system cannot open the temporary input file. Make sure that the InfoWater Temporary Folder selected has write privileges assigned to it.

303 - The system cannot open the status report file. See Error 302.

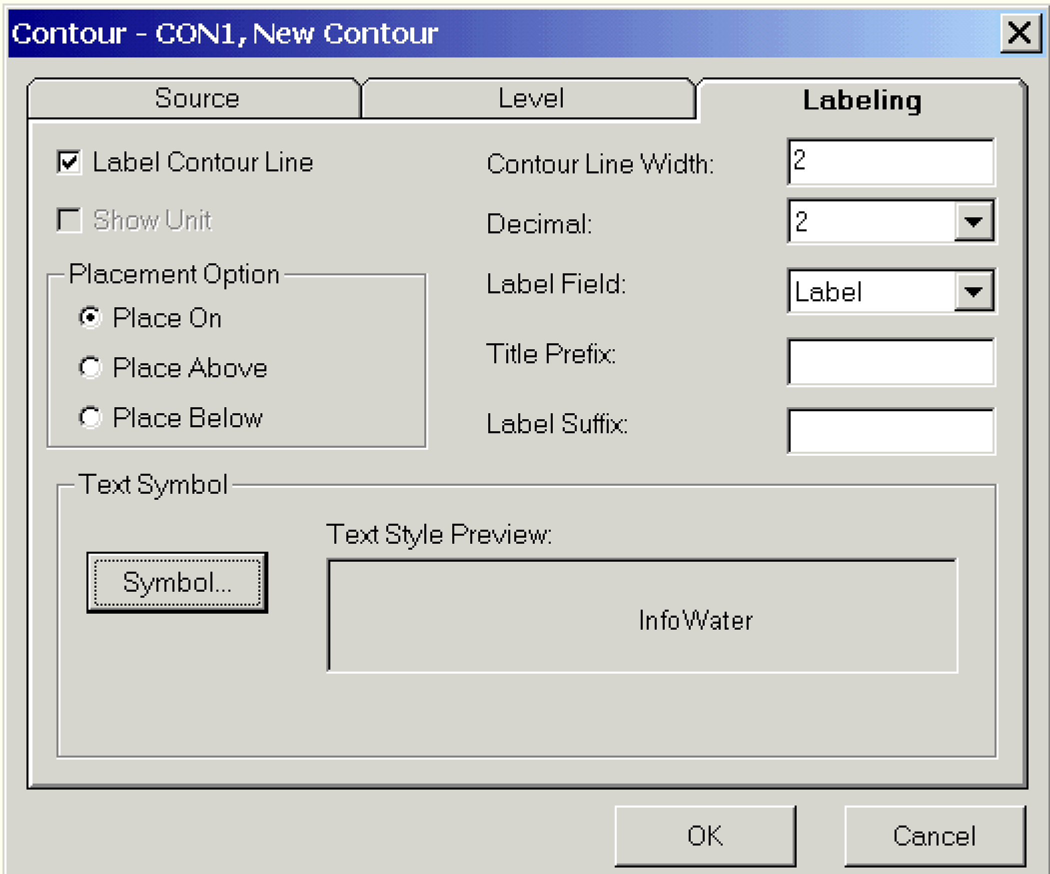
304 - The system cannot open the binary output file. See Error 302.

308 - Could not save results to file. This can occur if the disk becomes full.

309 - Could not write results to report file. This can occur if the disk becomes full.

Contour-Labelling

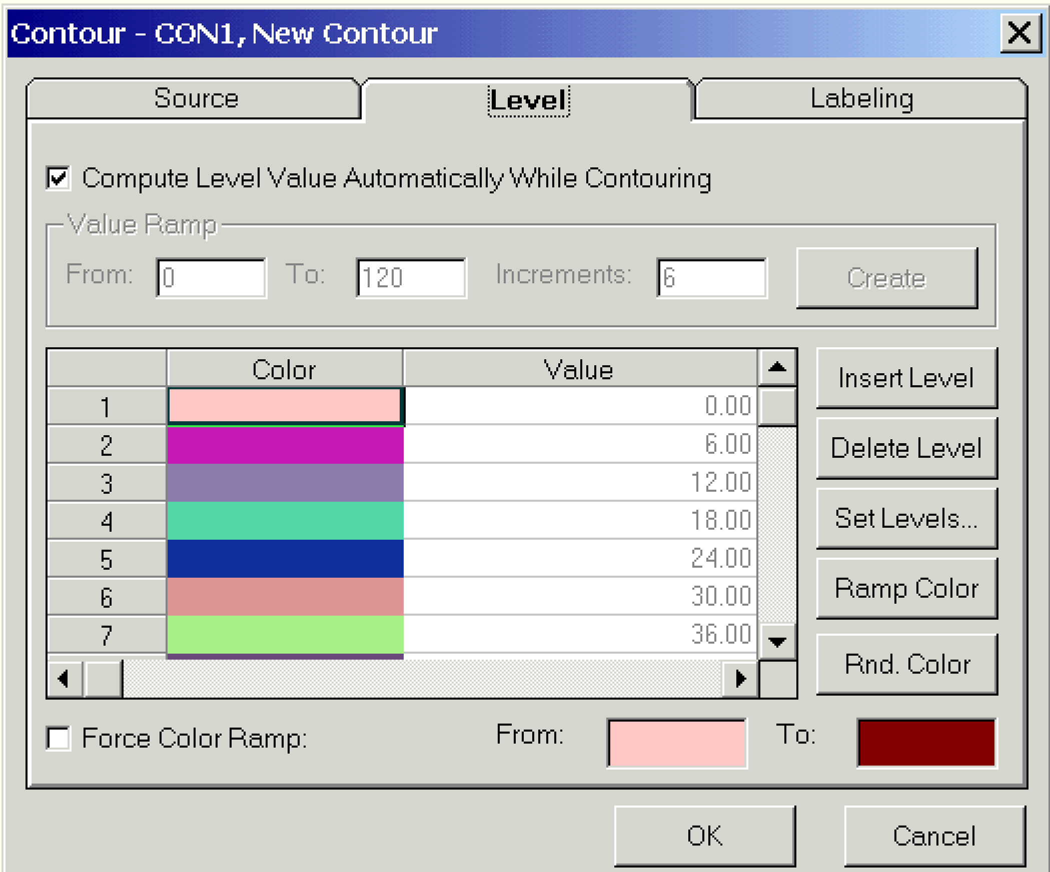
Specify your contour labeling and text placement options here. Click on any section below to learn more:



Other Related Topics - Annotation Dialog Box, Annotation Methodology, Contour-Level, Contour Dialog Box, Contour Methodology, Contour Options, Operation Data, Table of Contents

Contour -Level

Specify contour interval and colors here. Click on any portion below to learn more:



Other Related Topics - Annotation Dialog Box, Annotation Methodology, Contour-Labelling, Contour Dialog Box, Contour Methodology, Contour Options, Operation Data, Table of Contents

Contour Options

New - Create a new contour layer. Enter a unique ID and description when the new dialog box appears.

Edit - Edit the input data related to the contour layer. When initialized, the Contour Edit dialog box will appear and enable the user to change the features related to the contour layer. When the changes are made, click the OK button to see the changes.

Delete - Delete the selected contour layer.

Description - Edit the description of the selected contour layer.

Simulation Report Methodology

A simulation report can be created as part of a scenario or simulation run and is required whenever you wish to have the native analysis model report exported to a specific ASCII text file that differs from the default file.

While you can easily copy the contents of any report, customized report or query report to the Windows Clipboard to be viewed in any third party package, a simulation report contains the results from the model run itself, which contain model specific data. Model specific data consists of items like error messages, pump status, trial runs, element actions at timesteps, connectivity problems, etc.

Using Simulation Reports in InfoWater

Simulation Reports may be used for the following purposes:

- Use the Simulation Report to look at the hydraulic status during your simulation. Details for every time step and iteration is included here.
- The Simulation report may be used to trouble shoot your model, track warning messages and locate problem areas in your network.

Methodology

To create a New Simulation Report, perform the following procedure:

- From the **Table of Contents -> Operation** tab launch the **Simulation Report** dialog box. You may also click on the **Browse** icon next to the Simulation Reports section on the **Run Manager** dialog box to launch the Simulation Report dialog box. The Simulation Report dialog box may also be launched from the **General** tab of the **Scenario Manager** dialog box. Check the check box next to the Simulation Reports section and click on the **Browse** icon.

- Click on the **New** icon and specify an ID and a Description for the new Simulation Report (InfoWater IDs may contain up to 20 characters, no spaces and no funny characters while a Description may contain up to 60 characters). You may also clone an existing Simulation Report by selecting it and clicking on the **Clone** icon.

- Choose the different options available under the General tab, specify the title, size, hydraulic, pipe and the node status and choose to look at the warning messages.

- Click on the Customization tab if you want to customize it further. Choose the different criteria that you want to view and click on the Add button to add it to the report.

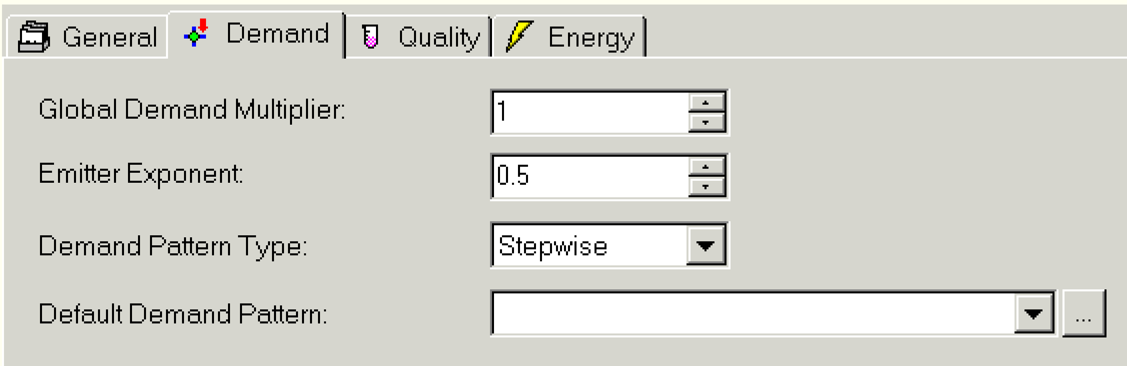
- Run your InfoWater analysis using the **Run Manager**.

- After a successful or unsuccessful simulation, view the Simulation report by clicking on the **Report** icon on top of the Run Manager dialog box. This will launch your Simulation report.

Other Related Topics - Running a Model**,** Simulation Options Dialog Box**,** Simulation Options - Demand, Simulation options - Energy, Simulation Options -Quality, Simulation Options Methodology, Simulation Report, Simulation Report Customization Tab, Simulation Time, Simulation Time Methodology

Demand

Use the Demand tab of the Simulations Options dialog box to set certain global values. Click on any section for more informations.



Other Related Topics - Running a Model**,** Simulation Options Dialog Box**,** Simulation options - Energy, Simulation Options -Quality, Simulation Options Methodology, Simulation Report, Simulation Report Customization Tab, Simulation Report Methodology, Simulation Time, Simulation Time Methodology

Simulation Options Methodology

The Simulation Options dialog box is used to adjust some facet of a model prior to a model run. You may create and customize the simulation Options for each scenario.

Using Simulation Options in InfoWater

Simulation Options may be used for the following purposes:

- Specify Modeling options. You may specify the InfoWater Data units, the equation to calculate head loss and other modeling defaults to be used during a simulation.
- From the Demand tab you may assign a Global Demand multiplier for all your InfoWater demand. You may also specify Emitter Exponents, demand Pattern type and a default demand pattern.
- From the Quality tab of the Simulation Options dialog box, you may specify the type of Water quality analysis that you want to run.
- Use the Energy tab to define and set energy management options. You would need to check the **Run Energy Management Simulation** option to run your Energy Analysis.

Methodology

To create a New Simulation Option, perform the following procedure:

- From the **Table of Contents -> Operation** tab launch the **Simulation Options** dialog box. You may also click on the **Browse** icon next to the Simulation Options section on the **Run Manager** dialog box to launch the Simulation Options dialog box. The Simulation options dialog box may also be launched from the **General** tab of the **Scenario Manager** dialog box. Check the check box next to the Simulation Options section and click on the **Browse** icon.

- Click on the **New** icon and specify an ID and a Description for the new Simulation Options (InfoWater IDs may contain up to 20 characters, no spaces and no funny characters while a Description may contain up to 60 characters). You may also clone an existing Simulation Option by selecting it and clicking on the **Clone** icon .

- Customize each of the General, Demand, Quality and Energy tabs to reflect your modeling preferences and click on the OK button to save and exit out of the Simulation Options dialog box.

- To associate the newly created Simulation Options with your presently active scenario, from the **Run manager** dialog box, click on the down arrow next to the Simulation Options section and select the newly created option. A better way to associate the simulation Options would be from the **Scenario -> Scenario Manager**, **General** tab where you can customize each scenario with a different Simulation Option.

Other Related Topics - Running a Model**,** Simulation Options Dialog Box**,** Simulation Options - Demand, Simulation options - Energy, Simulation Options -Quality, Simulation Report, Simulation Report

Customization Tab, Simulation Report Methodology, Simulation Time, Simulation Time Methodology

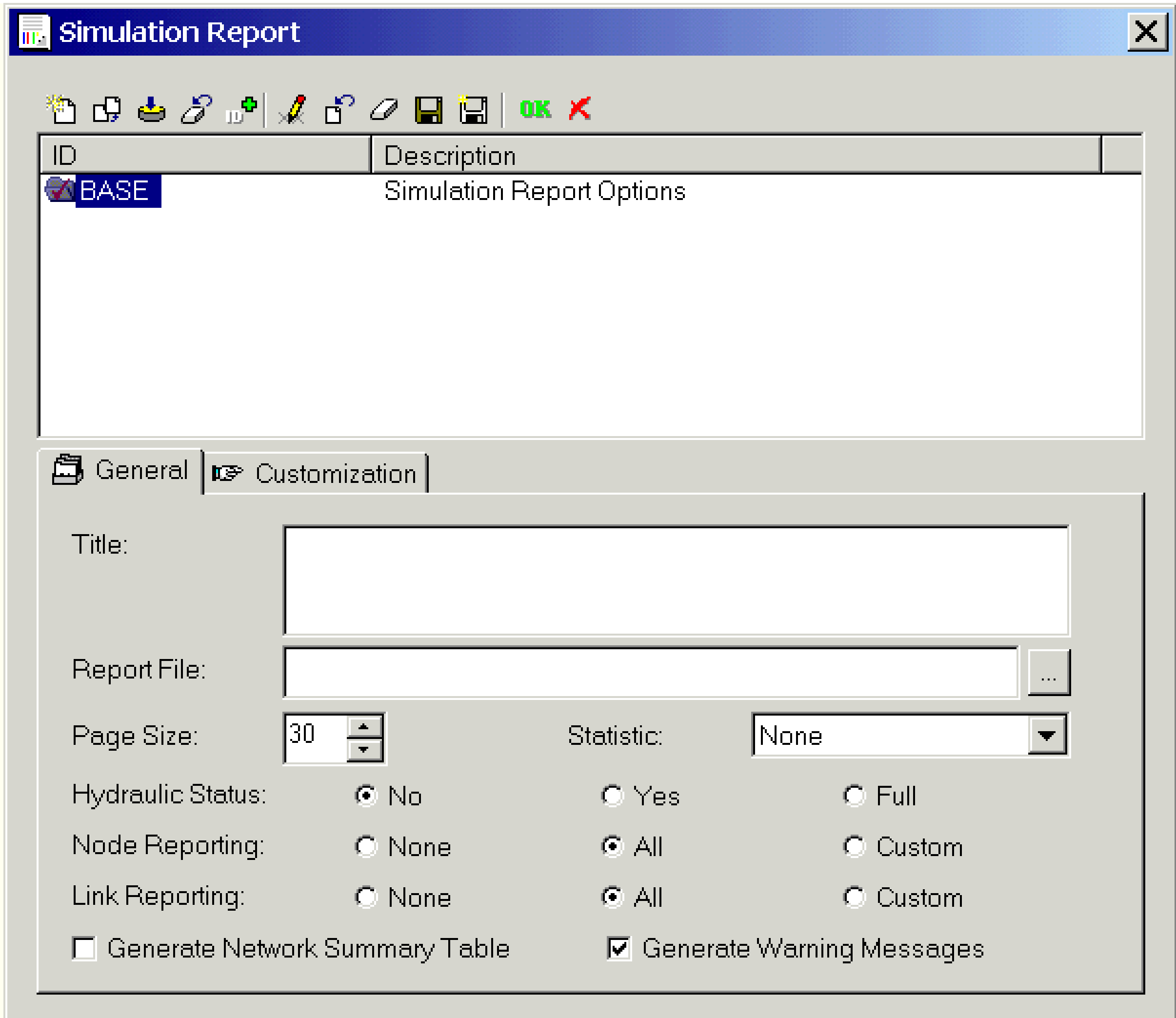
Simulation Report

A simulation report can be created as part of a scenario or simulation run and is required whenever you wish to have the native analysis model report exported to a specific ASCII text file that differs from the default file.

While you can easily copy the contents of any report, customized report or query report to the Windows Clipboard to be viewed in any third party package, a simulation report contains the results from the model run itself, which contain model specific data. Model specific data consists of items like error messages, pump status, trial runs, element actions at timesteps, connectivity problems, etc.

To create a simulation report, from the **Operation** tab of the **Table of Contents**, click on the **Simulation Report** .

Click on any portion below to learn more.



Other Related Topics - Running a Model, Simulation Options Dialog Box, Simulation Options - Demand, Simulation options - Energy, Simulation Options -Quality, Simulation Options Methodology, Simulation Report Customization Tab, Simulation Report Methodology, Simulation Time, Simulation Time Methodology

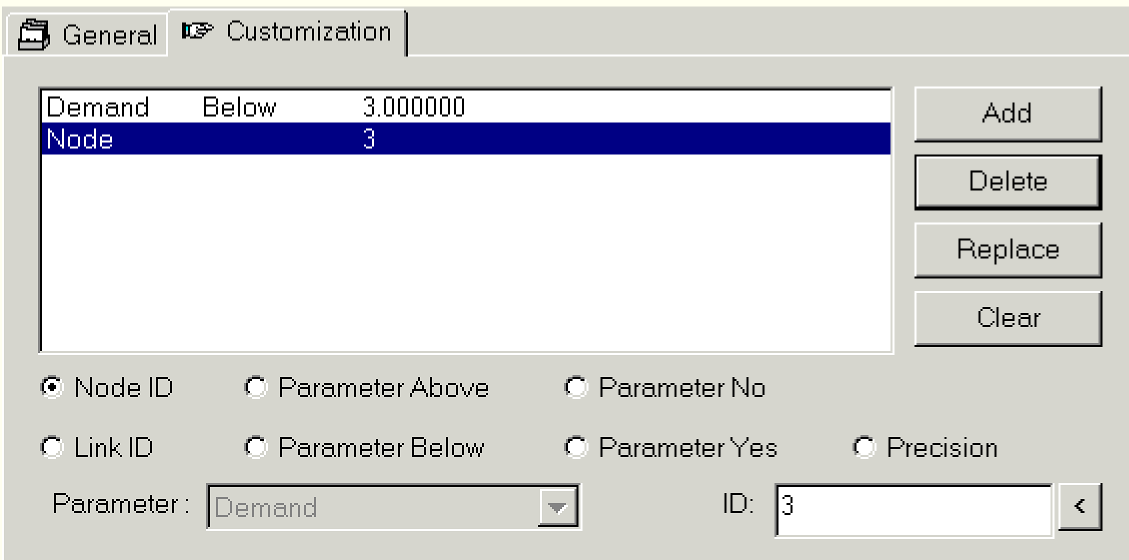
---

Simulation Report Customization Tab

The customization tab allows the user to specify a set of conditions (criteria) that are desired for the simulation report.

Once all options are specified for a simulation report, click on the **OK** button to close the dialog box. The new report can now be tied to a specific scenario through the Scenario Manager.

Click on any portion below to learn more.



Other Related Topics - Running a Model**,** Simulation Options Dialog Box**,** Simulation Options - Demand, Simulation options - Energy, Simulation Options -Quality, Simulation Options Methodology, Simulation Report, Simulation Report Methodology, Simulation Time, Simulation Time Methodology

---

Simulation Time Methodology

The Simulation Time feature of InfoWater allows the user to change the time-step at which an extended period simulation is to be analyzed. If the user selects the Steady State check box, then only a steady state model will be analyzed. A simulation time option can be set up for each unique scenario within InfoWater.

Many features of InfoWater including SCADA, Energy Management, Water Quality, and Logic Rules are dependent upon a model being analyzed over a series of timesteps (Extended Period Simulation).

Using Simulation Time in InfoWater

Simulation Time may be used for the following purposes:

- Choose a Steady State run by checking the steady state option.
- Uncheck the steady state option to run an Extended Period Simulation (EPS) and specify the duration of the run.
- Additionally specify the Hydraulic time step, pattern, report, quality, rule time step etc.
- You may create and customize the time setting for each scenario thereby letting you choose the different time options in InfoWater.

Methodology

To create a New Simulation Time set, perform the following procedure:

- From the **Table of Contents -> Operation** tab launch the **Simulation Time** dialog box by clicking on **Time**. You may also click on the **Browse** icon next to the Simulation Time section on the **Run Manager** dialog box to launch the Simulation Time dialog box. The Simulation Time dialog box may also be launched from the **General** tab of the **Scenario Manager** dialog box. Check the check box next to the Simulation Time section and click on the **Browse** icon.

- Click on the **New** icon and specify an ID and a Description for the new Simulation Time set (InfoWater IDs may contain up to 20 characters, no spaces and no funny characters while a Description may contain up to 60 characters). You may also clone an existing Simulation Time set by selecting it and clicking on the **Clone** icon.

- Choose from among the different time setting options to customize your run. Check the Steady State option to specify a Steady State run or specify the different options for your Extended Period run.

- To associate the newly created Simulation Time set with your presently active scenario, from the **Run manager** dialog box, click on the down arrow next to the Simulation Time section and select the newly created time set. A better way to associate the simulation time set would be from the **Scenario -> Scenario Manager**, **General** tab where you can customize each scenario with a different time set.

Other Related Topics - Running a Model**,** Simulation Options Dialog Box**,** Simulation Options - Demand, Simulation options - Energy, Simulation Options -Quality, Simulation Options Methodology, Simulation Report, Simulation Report Customization Tab, Simulation Report Methodology, Simulation Time

Fire Flow Explanation

Fire-Flows are an integral part of the Hydraulic modeling process and provide a means to calibrate your model. You may use the InfoWater Fire Flow module to analyze your existing hydrants or provide recommendations for future build-outs.

To learn more about conducting Fire Flow analyses click here.

In InfoWater, there are three constraints that are applied while conducting Fire-Flows.

Constraint 1- The Fire Flow demand imposed on each of your hydrants.

Constraint 2- The Residual Pressure that you impose on the Hydrant through the InfoWater Run Manager dialog box. This is the minimum pressure allowed at the hydrant.

Constraint 3- The **Minimum Design Pressure** that you impose on the junctions that may be affected when the Hydrant supplies water during a Fire-Flow Simulation. You may include these junctions by assigning them to your Critical Node Searching Range.

InfoWater will run your Fire-Flow analysis and provide you with two reports. The Fire-Flow Report or the Hydrant report will display the hydrant results. This report includes static demand, static pressure, fire flow demand, residual pressure, available flow at the hydrant and pressure at the available flow. The Fireflow design report displays the final design flow. The contents of this report are different, depending on whether the Minimum Design Pressure at the bottom of the Fireflow tab of the Run Manager dialog box is checked or not checked.

Fire Flow Report Variables

ID - Hydrant (Junction node) identifier.

Static Demand - This field displays the Static or the Basic Demand imposed on the Junction (200 gpm). Refer to #**1** in the Hydrant Graph below.

Static Pressure - This represents the static pressure corresponding to the static demand of 200gpm (126.72 psi). Refer to #**1** in the Hydrant Graph below. This pressure indicated the pressure at the hydrant when it only supplies the **Static Demand.** The Fire Flow Demand for the hydrant is not assigned while calculating the Static Pressure.

Fire Flow Demand - This is the Fire Flow demand imposed by you. To learn more about assigning Fire Flow demands click here.

Residual Pressure - This represents the resulting pressure because of a sum total of the Static Demand and the Fire Flow demand (1,200 gpm, 120 psi). Refer to #**2** in the Hydrant Graph below.

Available Flow @ Hydrant - The maximum flow that is available while maintaining the user-specified minimum residual pressure at the current node, assuming that only this hydrant (node) is flowing (6,013 gpm). Refer to #**3** in the Hydrant Graph below.

Available Flow Pressure - Residual pressure calculated for the available flow at the current hydrant (node). This value should equal the residual pressure specified by the user (20 psi). Refer to #**3** in the Hydrant Graph below.

Fire Flow Design Report

The Fireflow Design Report summarizes fire-flow simulation results such as the Total Demand, Critical Node 1 ID, Critical Node 1 Pressure, Adjusted Fire-Flow, Available Flow at the Hydrant, Critical Node 2 ID, Critical Node 2 Pressure, Adjusted Available Flow and Design Flow. InfoWater creates the Fireflow Design Report only if the Minimum Design Pressure option under the Design Flow Calculation section of the FireFlow tab in the Run Manager Dialog box is checked.

ID - Hydrant (Junction node) identifier.

Total Demand - The total nodal demand at the fire flow simulation timestep. This value corresponds to the sum of the baseline demand (Static Demand) and the fireflow demand (Fire-Flow Demand) for the selected timestep (12,00 gpm). Refer to #**2** in the Hydrant Graph below.

Critical Node 1 ID - Junction node within the **Critical Node Searching Range** with the lowest pressure when the current junction is loaded with the total demand.

Critical Node 1 Pressure - The calculated pressure at the Critical Node 1.

Adjusted Fire-Flow - The flow required at the hydrant to maintain the user-specified minimum design pressure at the Critical Node 1 (4,494.77 gpm to reach 20 psi at Junction 13). Refer to #**4** in the Hydrant Graph below.

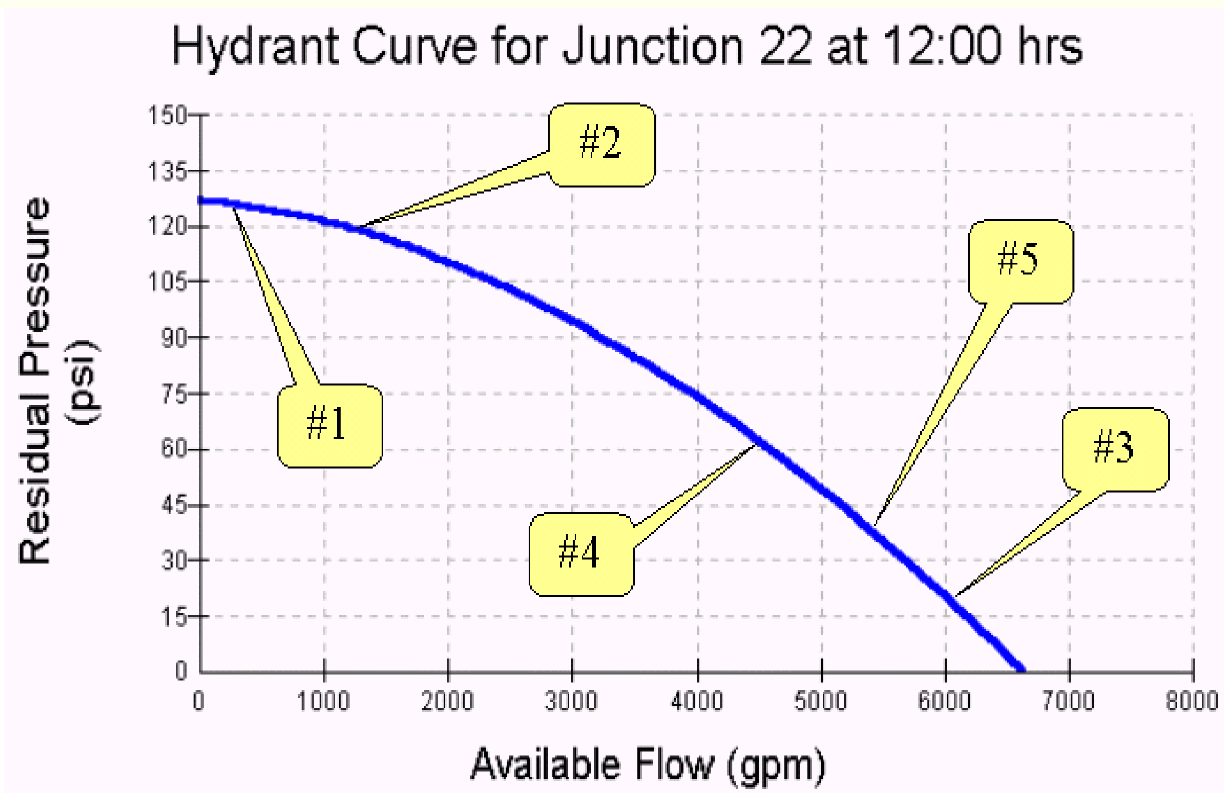
Available Flow @ Hydrant - The maximum flow that is available while maintaining the user-specified minimum residual pressure at the current node, assuming that only this hydrant (node) is flowing.

Critical Node 2 ID - Junction node within the **Critical Node Searching Range** with the lowest pressure when the current junction is loaded with the available flow.

Critical Node 2 Pressure - The calculated pressure at the Critical Node 2.

Adjusted Available Flow - The flow required at the hydrant to maintain the user-specified minimum design pressure at the Critical Node 2 (5,419.42 gpm to reach 20 psi at Junction 23). Refer to #**5** in the Hydrant Graph below.

Design Flow - The maximum flow available at the current hydrant (node) such that the pressure anywhere within the critical node searching range (Fire Nodes, Entire Network, or Domain) does not drop below the minimum design pressure specified. Based on the critical nodes 1and 2 identified (junction node with minimum pressure in the critical node searching range), InfoWater determines the design flow as the minimum of the two adjusted flows i.e., Adjusted Fire-Flow and the Adjusted Available Flow for each junction node needed to maintain the minimum design pressure in the entire critical node searching range. In final, point **#4** is ultimately the *Design Flow* as it is the least flow value of all considered residual pressures. This is the value that an engineer would provide to a local fire department by saying this is the maximum (theoretical) flow possible at the subject junction in order to maintain 20 psi at all times in the distribution system.



Other Related Topics - Available Report Types, Demand Cost, Energy Cost, Energy Summary, Fire Flow, Fire Flow Design, Junction

Report, Pipe Report, Pump Report, Range Reports, Reservoir Report, SCADA Alarm, SCADA Flow, SCADA Pressure, Tank Report, Valve Report

Functions & Operators

Logical expressions can be developed for use on the InfoWater Query Builder.

A Valid InfoWater query statement contains Operand1, Operand 2 and either a Logical Function or a Logical Operator connecting the two operands.

Logical Functions :

- **.AND. -** For the condition to be evaluated as true, the logical expression on both sides of the AND must be true.
- **.OR. -** For the condition to be evaluated as true, either the logical expression on one side or the other side of the OR must be true.
- **.NOT. Logical NOT -** returns false if the expression is true and returns true if the expression is false.
- **( ) Parentheses -** for grouping logical InfoWater expressions. InfoWater evaluates expressions in the inner-most parentheses first, then moving to the outer-most parentheses.
- **+ Addition**
- **- Subtraction**
- **‘ Quote -** For containing character strings or values in a character field. For example ‘CAST IRON’ or ‘10’ (assuming ‘10’ was stored in a character field).
- **SUBSTR( ) -** Returns a substring derived from a specified character field. For example SUBSTR(‘CAST IRON’,3,2) returns ‘ST’. SUBSTR(‘CAST IRON’,3) returns all characters from the third position to the end of the string; ‘ST IRON’.
- **UPPER( ) -** Converts all lowercase characters in a string to uppercase. For example UPPER(‘Main Street’) returns ‘MAIN STREET’.
- **CTOD( )** - Converts a character string to a date object. For example CTOD(‘2/15/98’) returns 02/15/98. CTOD(‘’) returns / / .

- **DTOC( )** - Converts a date expression to a character string. For example DTOC(02/15/98) returns ‘2/15/98’.
- **DTOS( )** - Converts a date expression to a character string in YYYYMMDD format. For example, DTOS(02/15/98) returns ‘19980215’.
- **DATE( )** - Returns your computers current system date. To evaluate the current date as a string, type the following: STR(DATE( )).
- **DAY( )** - Returns the numeric value of the day of the month in a date expression. For example DAY({2/15/98}) returns 15.
- **MONTH( )** - Returns the numeric value of the month in a date expression. For example MONTH({2/15/98}) returns 2.
- **YEAR( )** - Returns the year of a specified date expression as a four-digit number. For example YEAR({2/15/98}) returns 1998.
- **LEFT( )** - Returns a specified number of characters from the beginning of a character string. For example, LEFT(‘CAST IRON’,4) returns ‘CAST’.
- **VAL( )** - Returns a specified character string as a numeric value. For example, VAL(‘1200 Main Street’) returns 1200.
- **STR( )** - Returns the character string equivalent of a specified number. For example STR(-32) returns ‘-32’ and STR(100) returns ‘100’.

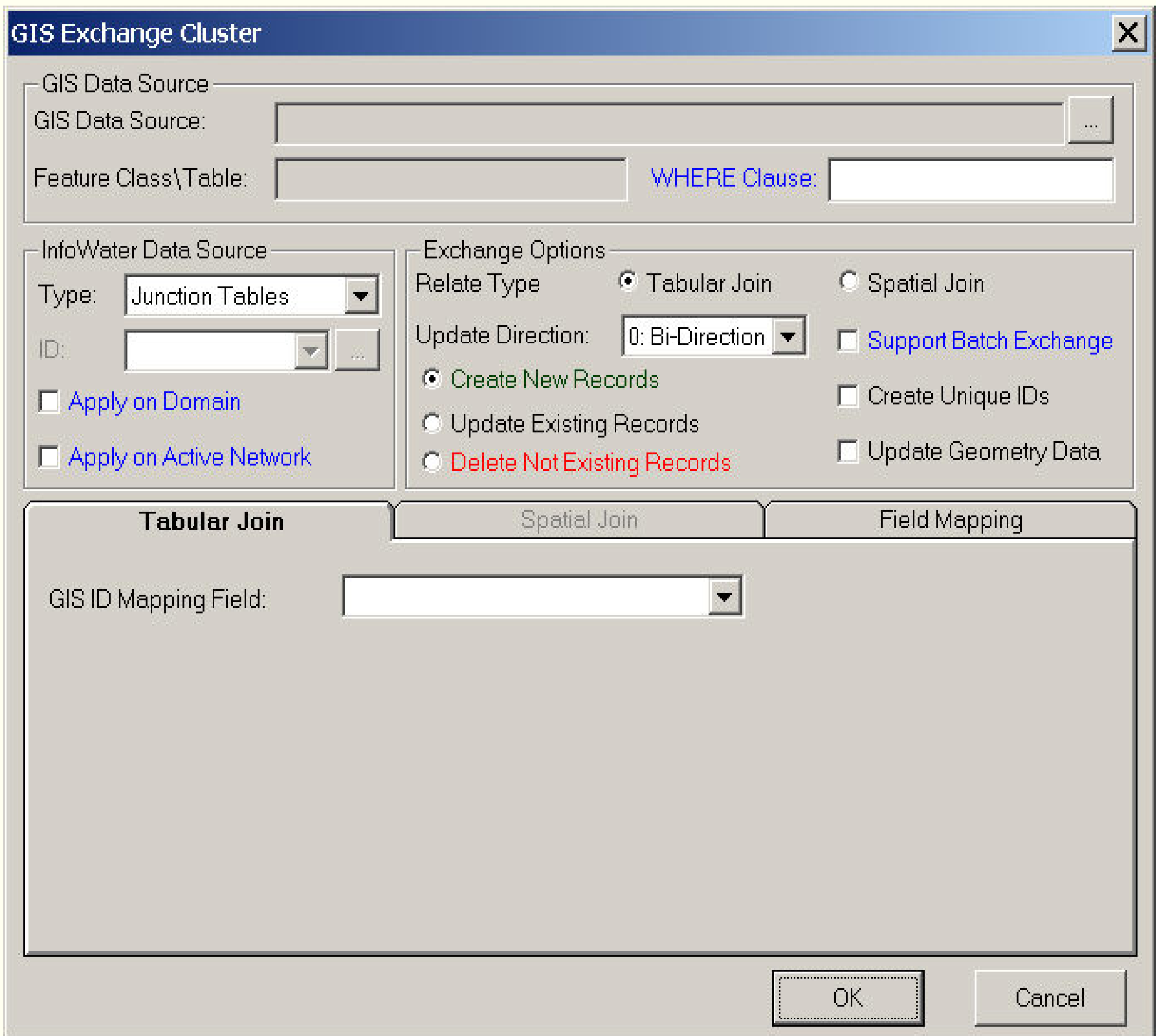
Logical operators :

- **= Equal** - Operand-1 is equal to Operand-2
- **<> Not Equal** - Operand-1 is not equal to Operand-2
- **=> Greater Than or Equal** - Operand-1 is greater than or equal to Operand-2
- **<= Less Than or Equal** - Operand-1 is less than or equal to Operand-2
- **> Greater Than -** Operand-1 is greater than Operand-2
- **< Less Than** - Operand-1 is less than Operand-2
- **<> Not Contain -** Operand-1 does not contain Operand-2.
- **$ Contains -** Operand-1 contains Operand-2.

GIS Exchange Cluster

A GIS Exchange Cluster defines a linkage toward the desired GIS data source and a set of data exchange parameters that allows the user to quickly load and save data when the cluster is applied. This helps to perform data exchange with various GIS data sources supported by ESRI including personal and enterprise Geodatabases. Click here to learn more about **Working with ESRI Geodatabases**.

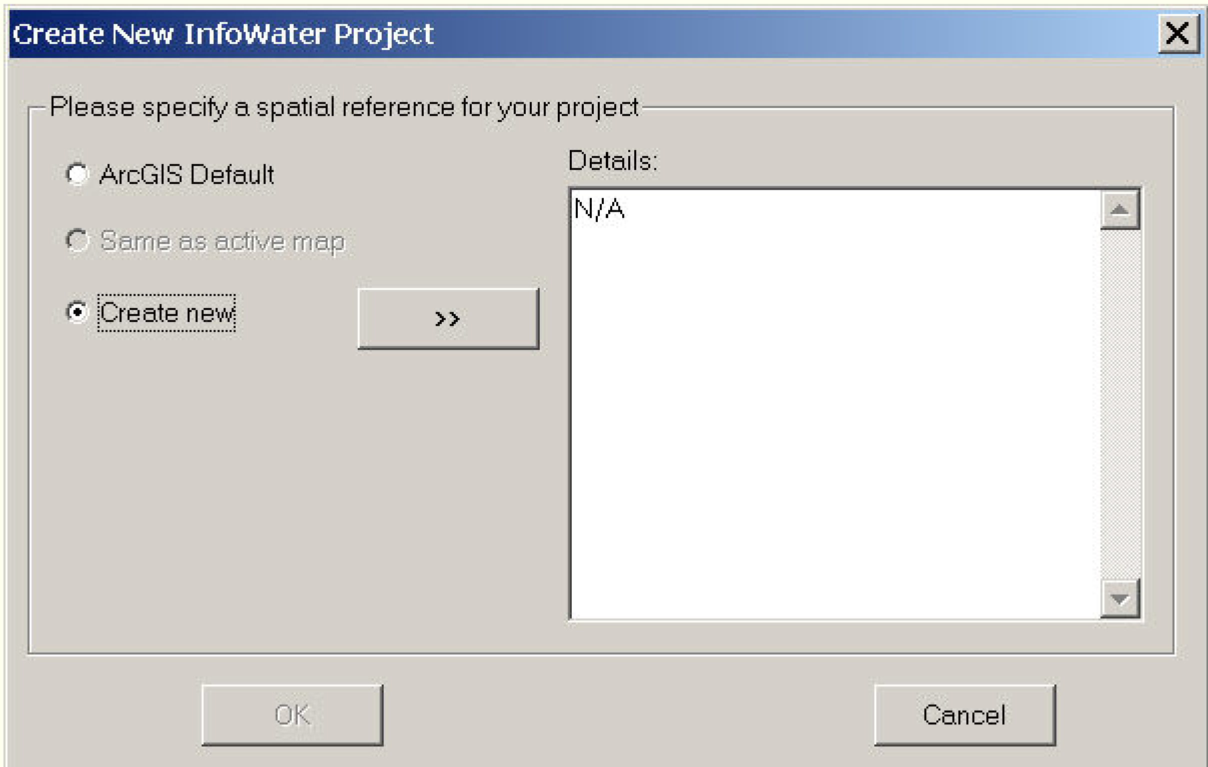
Click on any section below to learn more:



Other Related Topics - Working with ESRI Geodatabases, Geodatabase Theory, GIS Cluster -Spatial Join, GIS Exchange Cluster Tabular Join, GIS Field Mapping, GIS Gateway

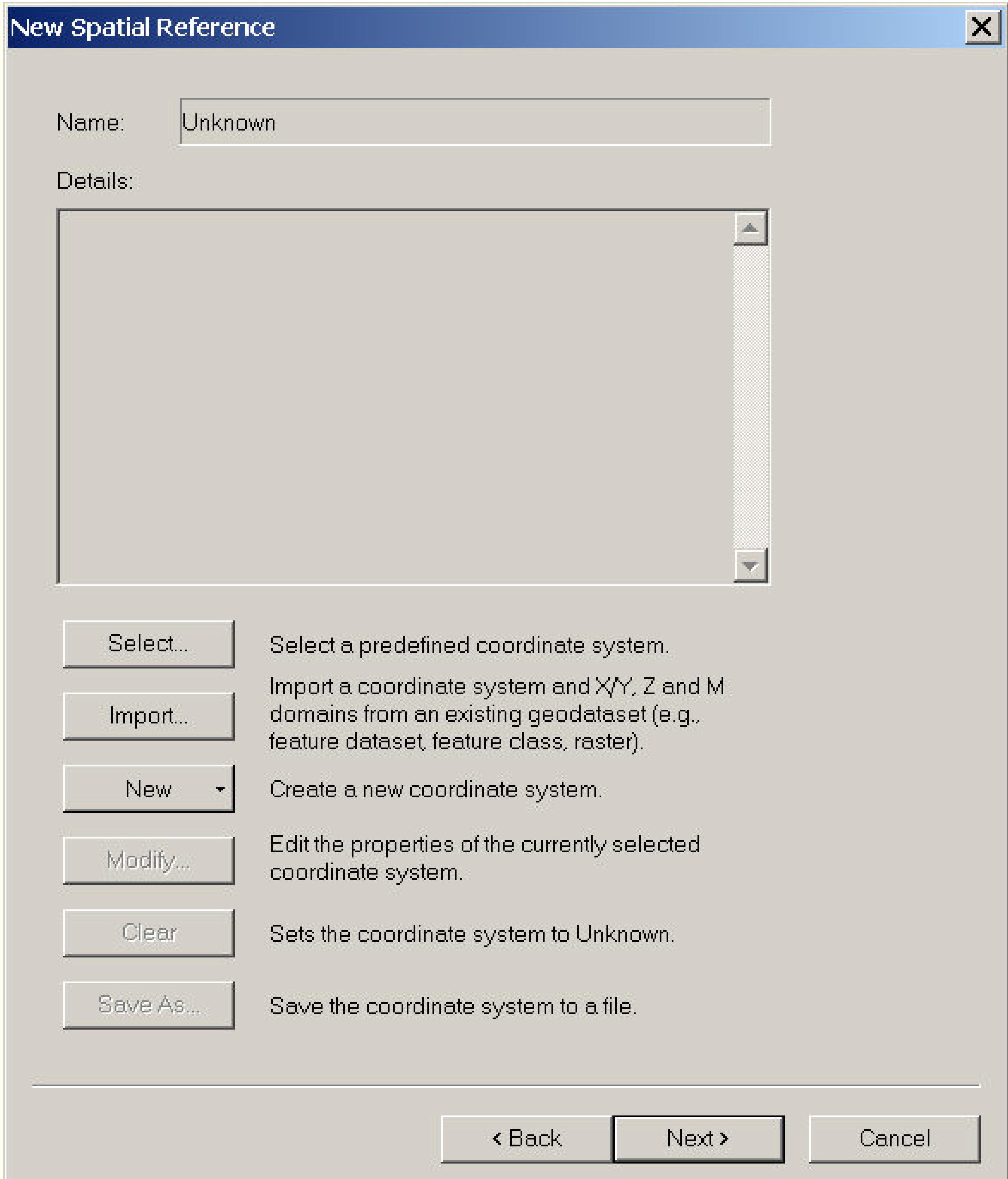
Setting Drawing Limits

After initializing your InfoWater project, be sure to set your drawing limits by choosing the **Create New** option.



In the next dialog box choose the Import button 

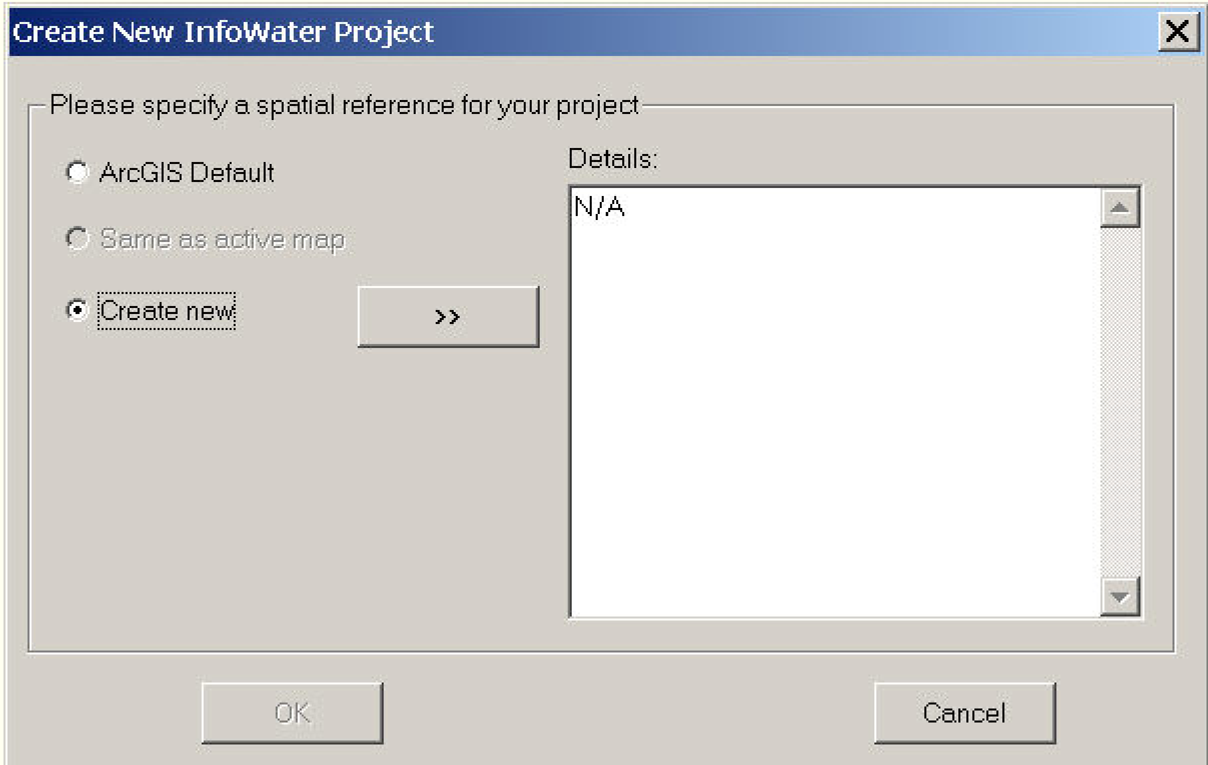
and browse to your H2OMAP project and choose the Nodes.shp file in the MAP sub-directory.



Click the Next button to choose the default settings for the remaining options and then click on the OK button to close the last dialog box.

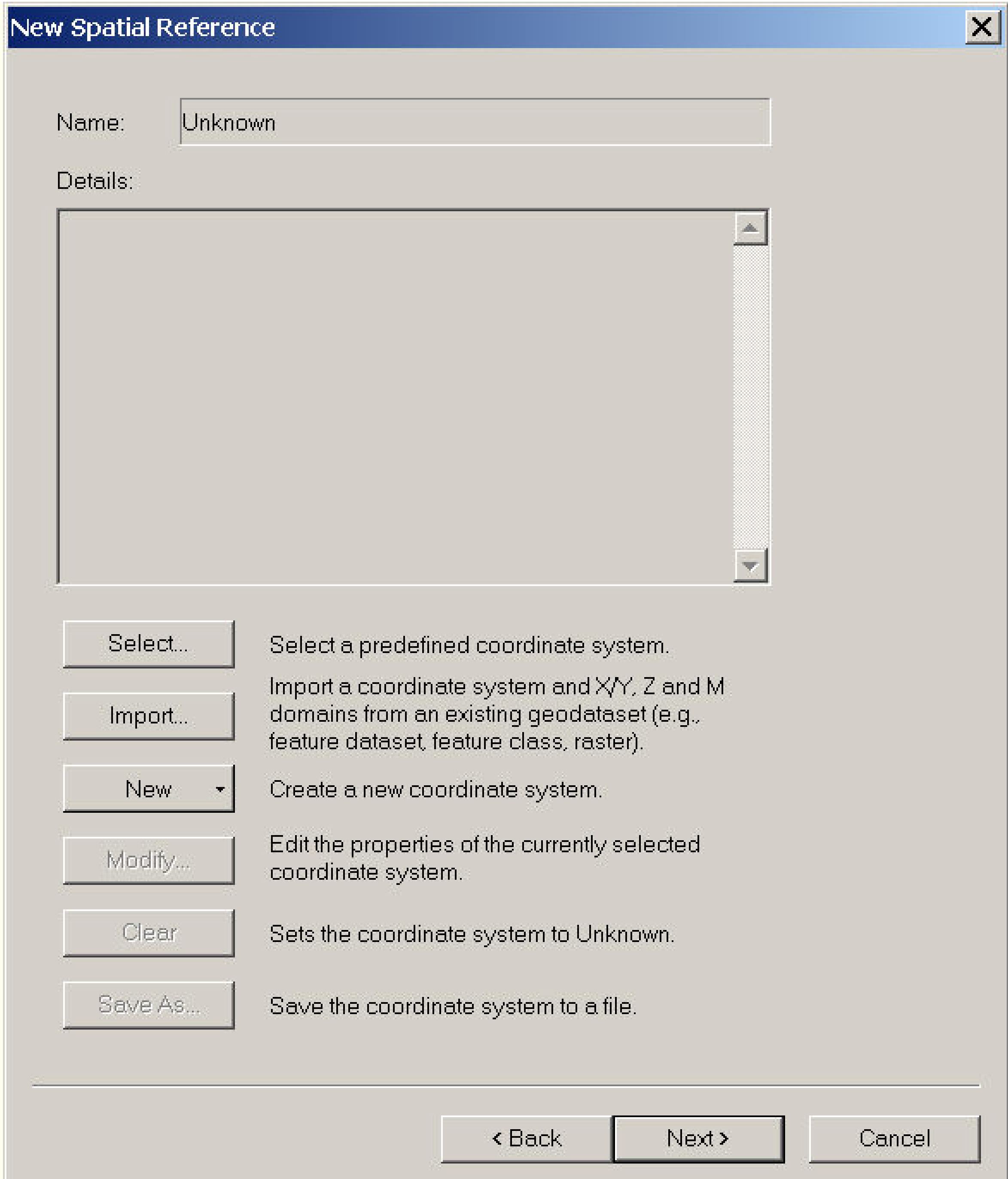
Setting Drawing Limits

After initializing your InfoWater project, be sure to set your drawing limits by choosing the **Create New** option.



In the next dialog box choose the Import button 

and browse to your H2ONET project and choose the AutoCAD drawing file (.DWG extension).



Click the Next button to choose the default settings for the remaining options and then click on the OK button to close the last dialog box.

Geodatabase Theory

Geodatabases contain geographic information and typically comprise of feature classes and/or tables.

A Feature class stores various geographic features represented as points, lines, or polygons, and their attributes. Tables on the other hand contain additional attributes for a feature class or geographic information such as addresses or x,y,z coordinates.

ArcView® includes a data model for representing geographic information called the geodatabase data model. The geodatabase can be used to define and work with different user - or application - specific models. By defining and implementing behavior on a generic geographic data model, the geodatabase provides a robust platform for virtually any GIS application.

Further ArcView® allows you to access two broad categories of geodatabases: personal geodatabases and multiuser geodatabases managed using ArcSDE™.

Personal Geodatabases

Personal geodatabases support many readers and a single editor. They are stored in a Microsoft Access database. You can create and work with personal geodatabases with ArcGIS™ without the need for any other software.

SDE Geodatabases

Multiuser geodatabases can be read and edited by multiple users; they require a DBMS, such as Oracle, SQL Server, Informix, or IBM DB2. Multiuser geodatabases can be used with any ArcGIS product (ArcView®, ArcInfo™, or ArcEditor™) but require ArcSDE for editing and schema management.

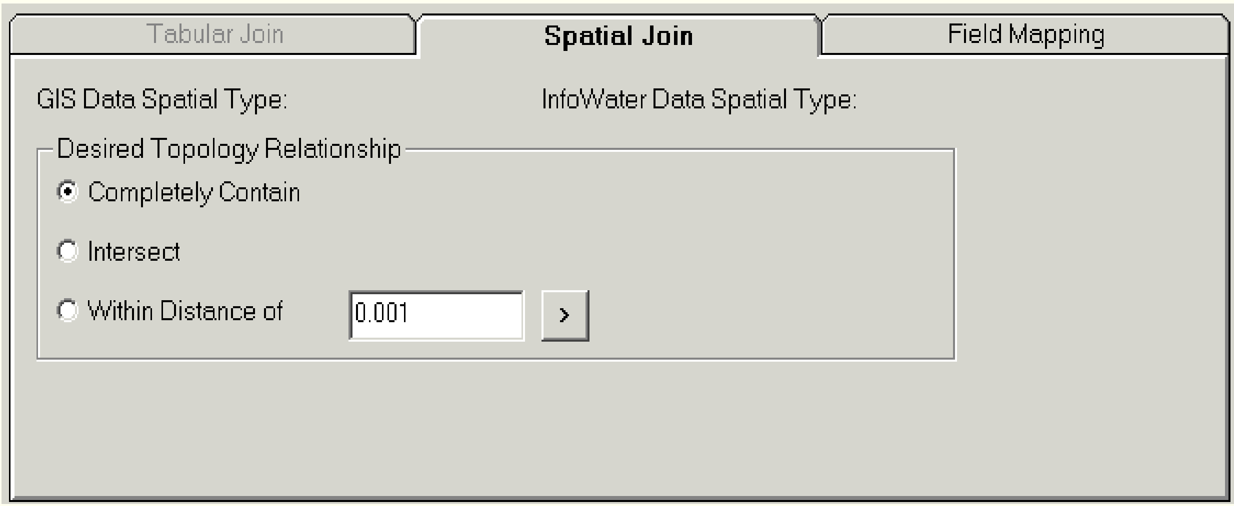
InfoWater allows you work with these Geodatabases. Refer to the section on Geodatabase Import/Export Methodology to learn more. To learn more about the ArcView® Geodatabases refer to the ArcGIS help file. Click here to learn more about **Working with ESRI Geodatabases**.

Other Related Topics - Working with ESRI Geodatabases, GIS Cluster - Spatial Join, GIS Exchange Cluster, GIS Exchange Cluster - Tabular Join, GIS Field Mapping, GIS Gateway

---

GIS Exchange Cluster - Spatial Join

A spatial join allows you to perform a join between two data layers based on their physical relationship to one another. This means that the data contained in one data layer can be assigned to another table by using 'smart topography' features of InfoWater.



Once the spatial join options have been specified, proceed to the Field Mapping tab to map the desired fields prior to exchanging the data. Click here to learn more about **Working with ESRI Geodatabases**.

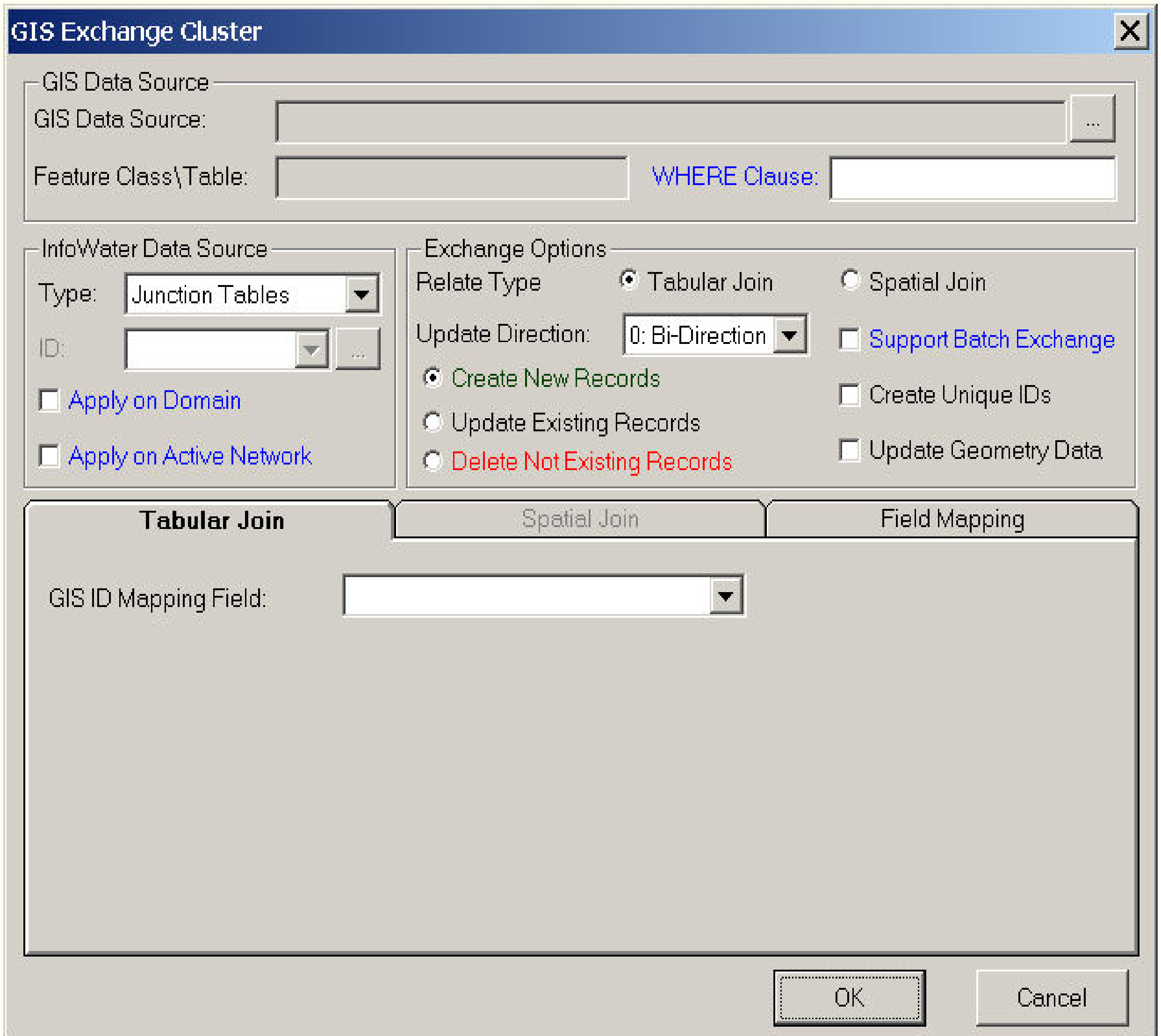
Other Related Topics - Working with ESRI Geodatabases, Geodatabase Theory, GIS Exchange Cluster, GIS Exchange Cluster Tabular Join, GIS Field Mapping, GIS Gateway

---

GIS Exchange Cluster - Tabular Join

A tabular join merely exchanges data fields between one GIS data source and another. While using the InfoWater GIS Gateway dialog box you may update your InfoWater data fields from an external geodatabase or alternatively update the external geodatabase with InfoWater data. Click here to learn more about **Working with ESRI Geodatabases**.

Click on any section below to learn more :



Once the tabular join options have been specified, proceed to the Field Mapping tab to map the desired fields prior to exchanging the data.

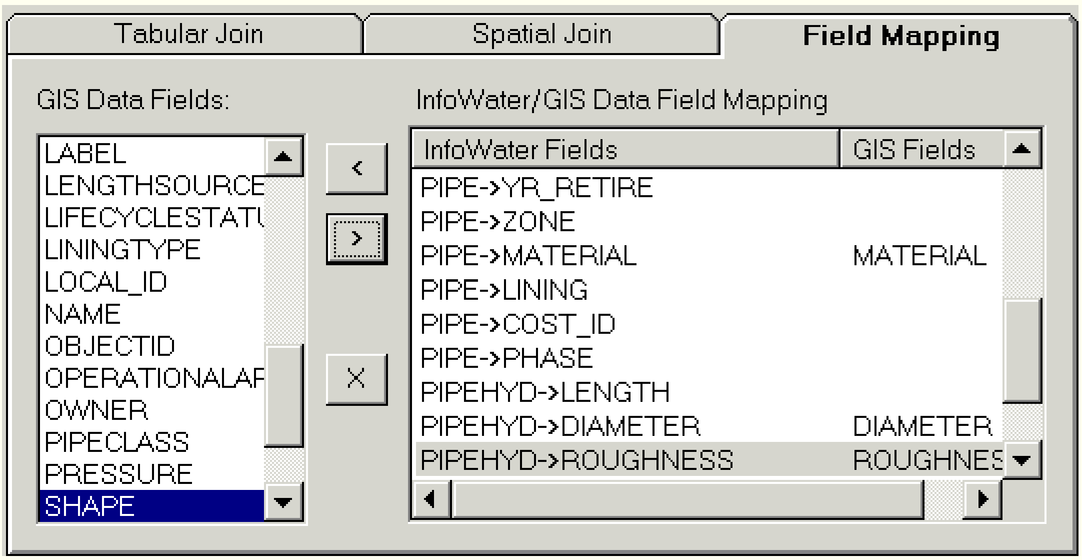
Other Related Topics - Working with ESRI Geodatabases, Geodatabase Theory, GIS Cluster -Spatial Join, GIS Exchange Cluster, GIS Field Mapping, GIS Gateway

---

GIS Field Mapping

The field mapping tab allows you to map corresponding fields to one another prior to performing the GIS Exchange. Merely select fields on the left hand side to be mapped to InfoWater fields on the right hand side. Once the field mapping has been completed, click on the **OK** button to finish the GIS Exchange. Click here to learn more about **Working with ESRI Geodatabases**.

Click on any section below to learn more.



Other Related Topics - Working with ESRI Geodatabases, Geodatabase Theory, GIS Cluster -Spatial Join, GIS Exchange Cluster, GIS Exchange Cluster Tabular Join, GIS Gateway

Control Sets

Initial pipe status (open, closed, setting) and any operational control rules defined at the time the control set was created. Controls only apply to Pumps, Pipes and Valves and are not applicable to nodes.

Curve Sets

All Curves in the InfoWater project.

Demand Set

Modeling data associated with junction nodes. Demand sets are used to save modeling data on junction node demands and demand pattern identifiers and retrieve those data as part of a scenario. The only attributes stored in a demand set are the baseline demands and the associative pattern. The Auto-Demand Reset preference governs how demands are assigned when activating demand sets.

Energy Sets

Energy data for pumps including pump efficiency, energy charging rate patterns, and demand charge patterns.

Fire FLow Sets

All the Fireflow demands for junction nodes.

Logic Sets

Logical controls used for rule based logic controls. Logical controls are created and maintained with the Rule-Based Control command (See InfoWater Control Center -> InfoWater button -> Edit -> Rule-Based Control). Use the Edit Logic Set to store logical controls in a logic set and to maintain those logic sets.

Pattern Sets

All patterns in the InfoWater project.

Pipe Sets

Modeling data associated with pipes. Pipe sets are used to save modeling data on pipes and retrieve those data as part of a scenario. Pipe sets contain modeling information including pipe diameters, lengths, roughness coefficients, minor loss coefficients, and presence or absence of check valves and flow totalizers.

Pump Sets

Modeling data associated with pumps. Pump sets are used to save modeling data on pumps and retrieve those data as part of a scenario. Pump sets contain modeling information including pump curve type and pump curve parameters.

Quality Sets

Water quality modeling parameters including quality source nodes, initial water quality at junction and storage nodes, and reaction rate coefficients for pipes and tanks. Water quality parameters are maintained with the Create, Edit, or Edit Group commands. Use

the Edit Quality Set command to create and manage quality sets.

Reservoir Sets

Modeling data associated with Reservoirs. Reservoir sets are used to save modeling data on reservoirs and retrieve those data as part of a scenario. Reservoir sets contain modeling information including type of storage node (fixed-grade reservoir or variable head reservoir) and related storage node characteristic data such as head and patterns defining grade in variable-head reservoirs.

SCADA Set

All the SCADA information for the InfoWater project.

Tank Set

Modeling data associated with Tanks. Tank sets are used to save modeling data on tanks and retrieve those data as part of a scenario. Tank sets contain modeling information including type of storage node (cylindrical tank and variable-area tank) and related storage node characteristic data such as bottom elevation, initial water level, tank dimensions including diameter, minimum and maximum levels, minimum volume and curves defining variable area tanks.

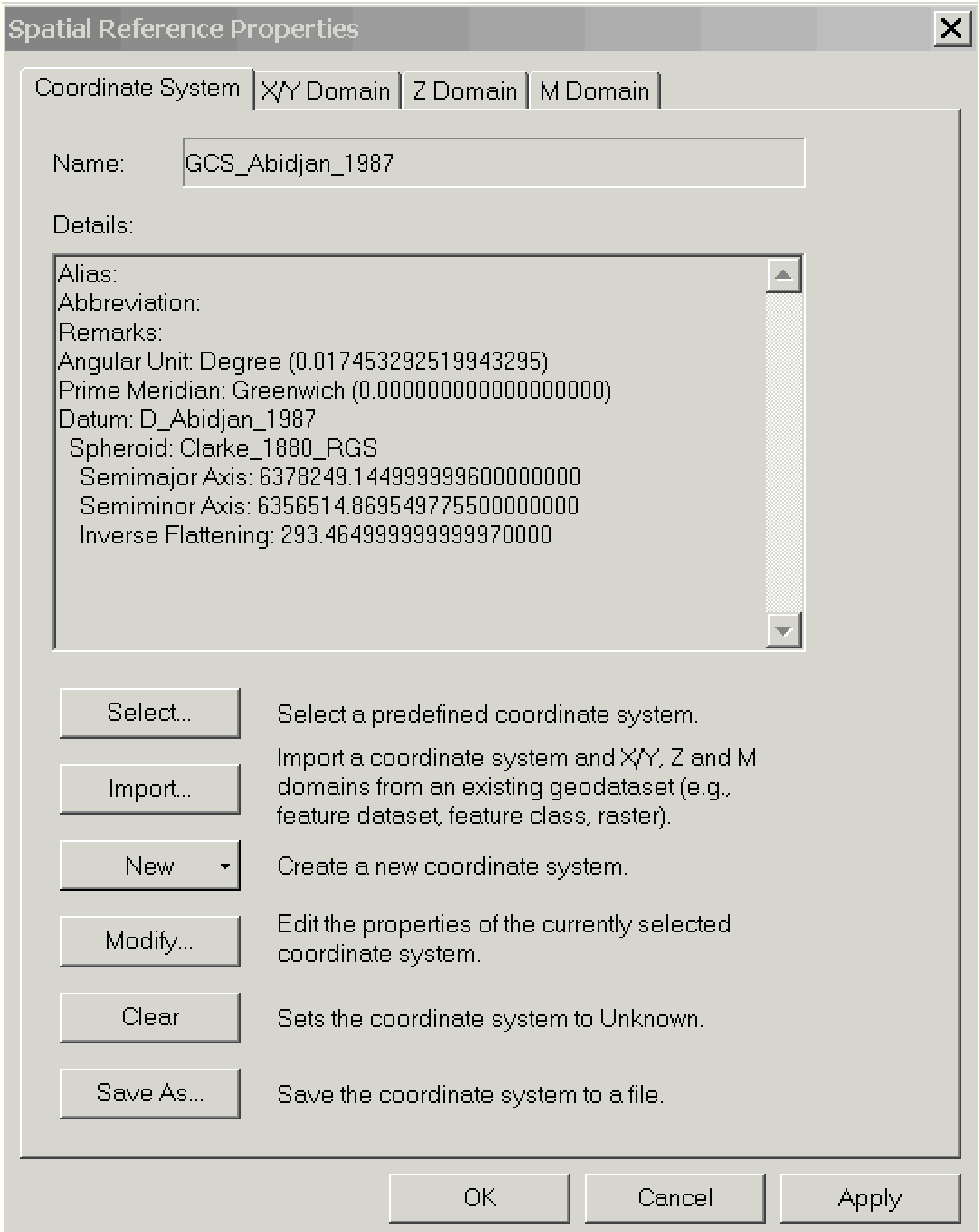
Valve Sets

Modeling data associated with control valves. Valve sets are used to save modeling data on control valves and retrieve those data as part of a scenario. Valve sets contain modeling information including valve type and related valve characteristic data such as pressure setting for PRVs, flow setting for FCVs, tank grades for float valves, etc.

New Spatial Reference Dialog box

Use this to either define a new custom coordinate system or use an existing available coordinate system from the ArcGIS library.

Click on any section below to learn more. Click on Next to launch the Spatial System Coordinate Range dialog box.



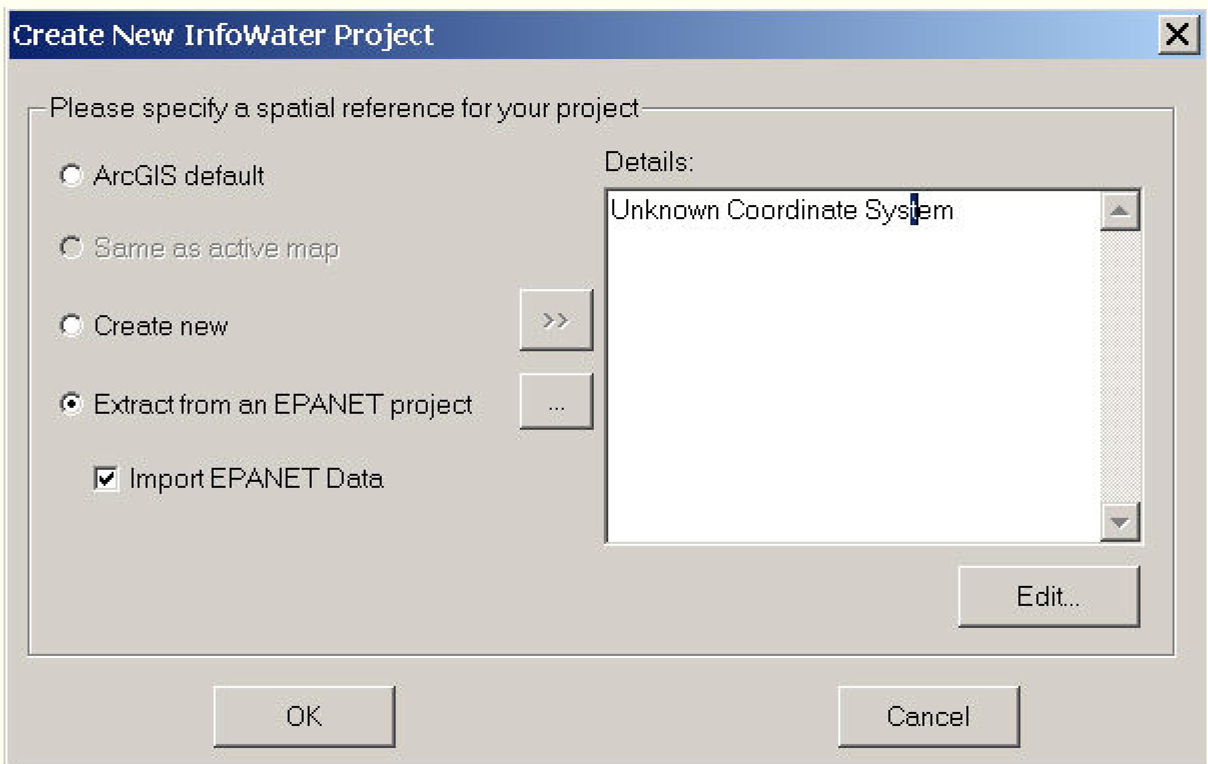
Other Related Topics - Change Project Spatial Reference, Create New InfoWater Project, Degree of Resolution, Spatial System Coordinate Range

---

Import EPANET data

The **Extract from an EPANET project** option can also be used to import EPANET v2.x data files into InfoWater. To import EPANET v2.x files perform the following:

- From Create New InfoWater Project dialog box select the option Extract from an EPANET project.
- Click on the More button to choose the location where you want to import your EPANET input file from.
- Then check the **Import EPANET data** box.
- Finally click **OK** to start the import process.



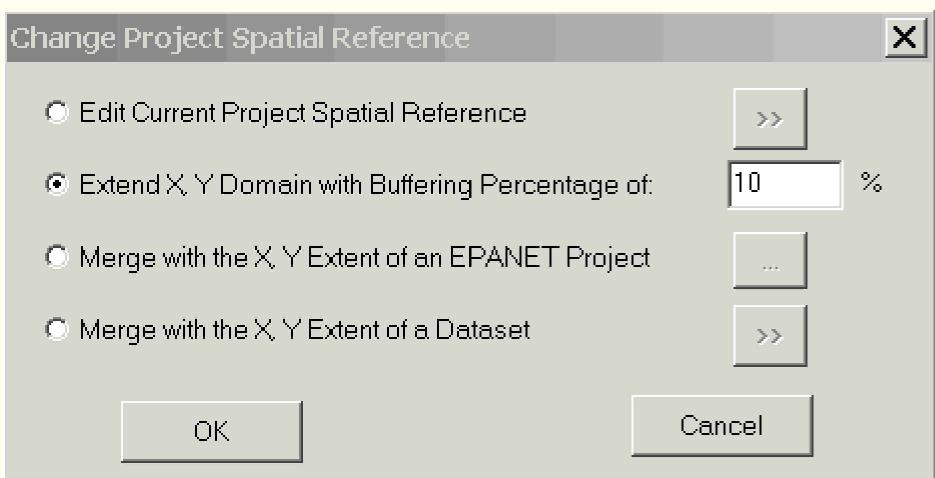
Change Project Spatial Reference

The spatial reference of a dataset in geodatabase can not be further modified once created in ArcGIS and new features can not be inserted if it's coordinates are beyond X, Y domain of the spatial reference. InfoWater provides a very powerful tool to overcome this limitation of ArcGIS.

In InfoWater the spatial reference of a project can be modified by either editing current project's spatial reference, extending the X, Y domain with buffering percentage, merging with the X, Y extent of an EPANET project, or merging with the X, Y extent of a dataset.

To launch the Change Project Spatial Reference dialog box, choose **Change Project Spatial Reference** from the **Project** pull down menu under the **InfoWater Control Center -> InfoWater** button.

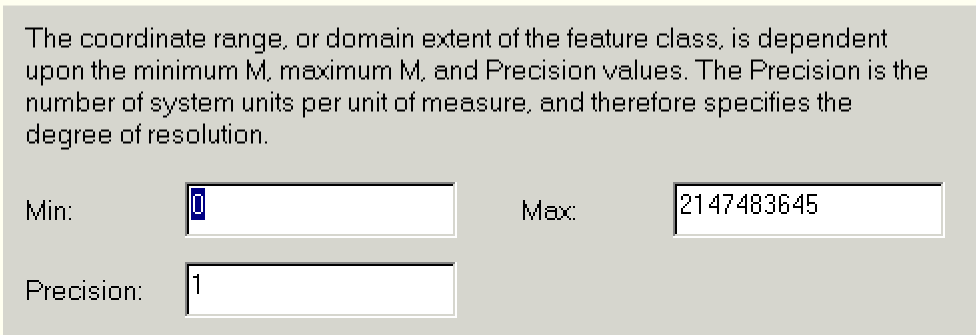
Click on any section below to learn more.



Other Related Topics - Create New InfoWater Project, Degree of Resolution, New Spatial Reference Dialog box, Spatial System Coordinate Range

Degree of Resolution

Specify the degree of resolution here. The resolution degree is specified by the precision value and the minimum and the maximum values as mentioned below.



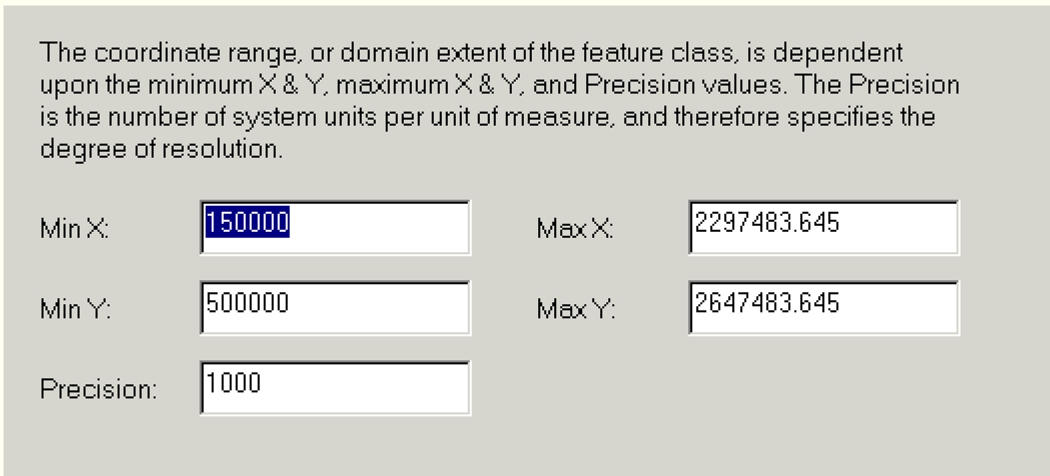
Finally click on Finish to create your new spatial reference file.

Other Related Topics - Change Project Spatial Reference, Create New InfoWater Project**,** New Spatial Reference Dialog box**,** Spatial System Coordinate Range

---

Spatial Coordinate Range

Click on **Next** on the New Spatial Reference Dialog box to launch this dialog box. Specify your coordinate range here. Click on Next to launch the Degree of Resolution dialog box.

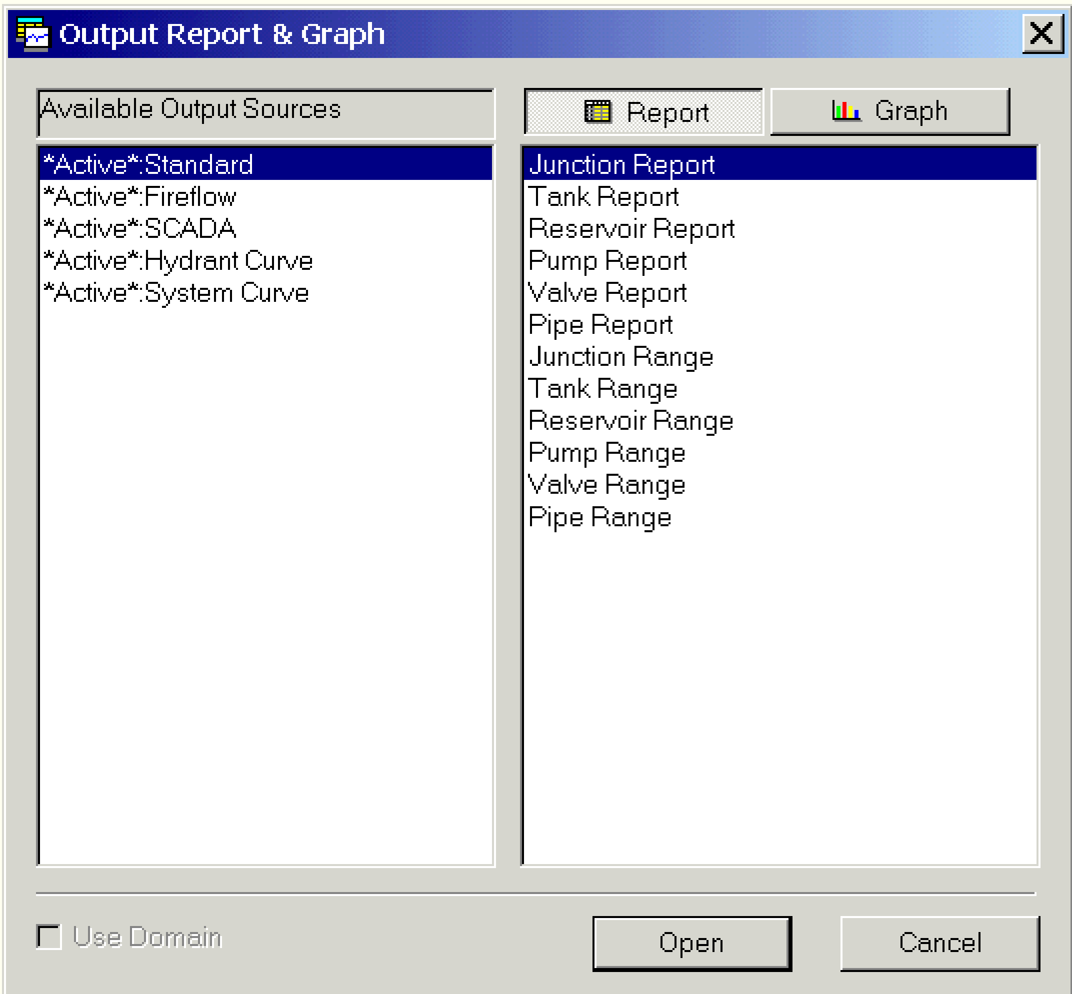


Other Related Topics - Change Project Spatial Reference, Create New InfoWater Project**,** Degree of Resolution**,** New Spatial Reference Dialog box

Output Report & Graph Dialog Box

This dialog box provides a means to select the output reports/graphs that you want to see in your **Output Report Manager**.

Click on any section for more information.



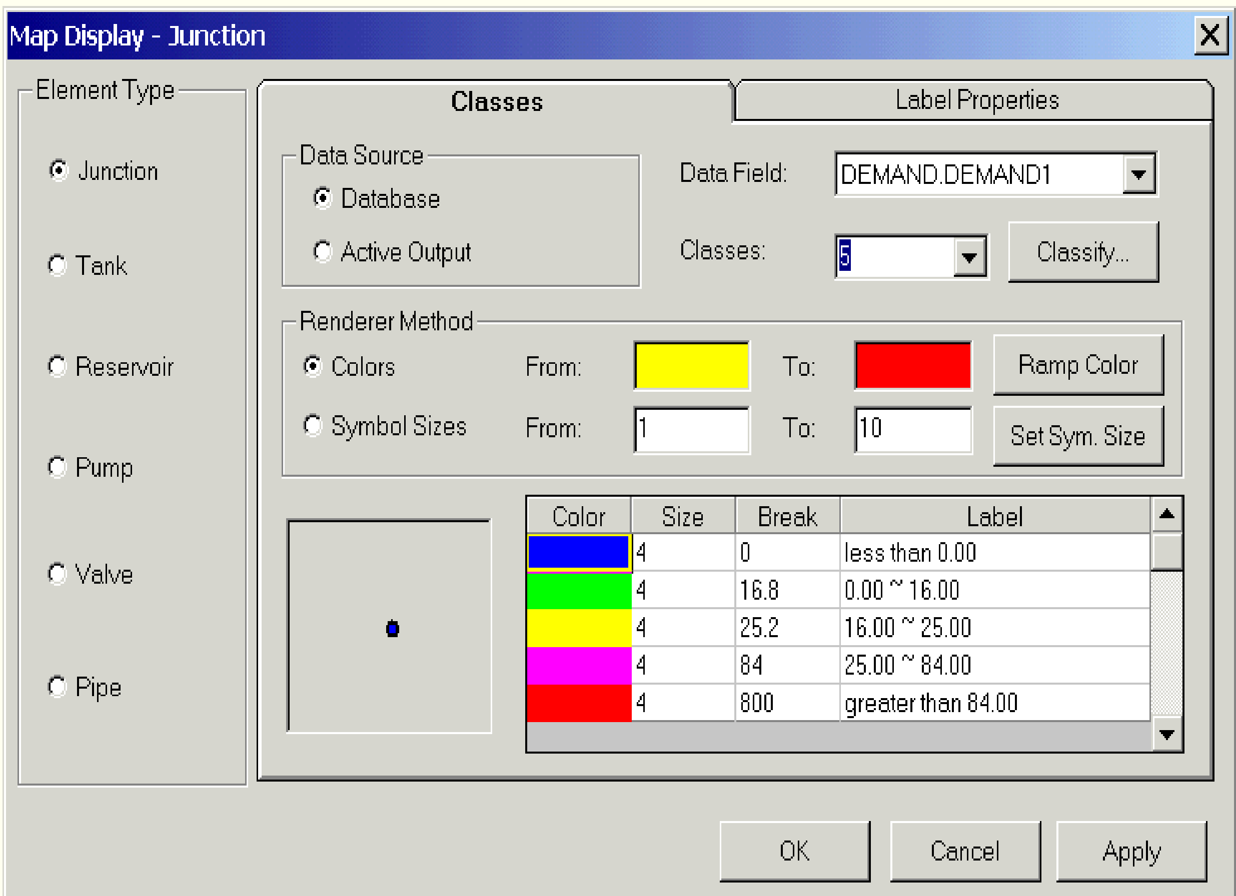
Other Related Topics - Available Report Types, Demand Cost, Energy Cost, Energy Summary, Fire Flow, Fire Flow Design, Fire Flow Report Explanation, Junction Report, Pipe Report, Pump Report, Range Reports, Reservoir Report, SCADA Alarm, SCADA

[Flow], [SCADA Pressure], [Tank Report], [Valve Report], [Output Report Manager], [Output Report Methodology]

Map Display

The Map Display button of the InfoWater Control Center is used by InfoWater to color code pipes or nodes based on information contained within the input and output databases. The Map Display feature only works on InfoWater data elements. Click here to learn how to color code and annotate your Map using the InfoWater Map Display feature.

Click on any portion of the dialog box below to learn more.



Note: Color Coding will **NOT** work for a domain. InfoWater will always color code the **ENTIRE** network.

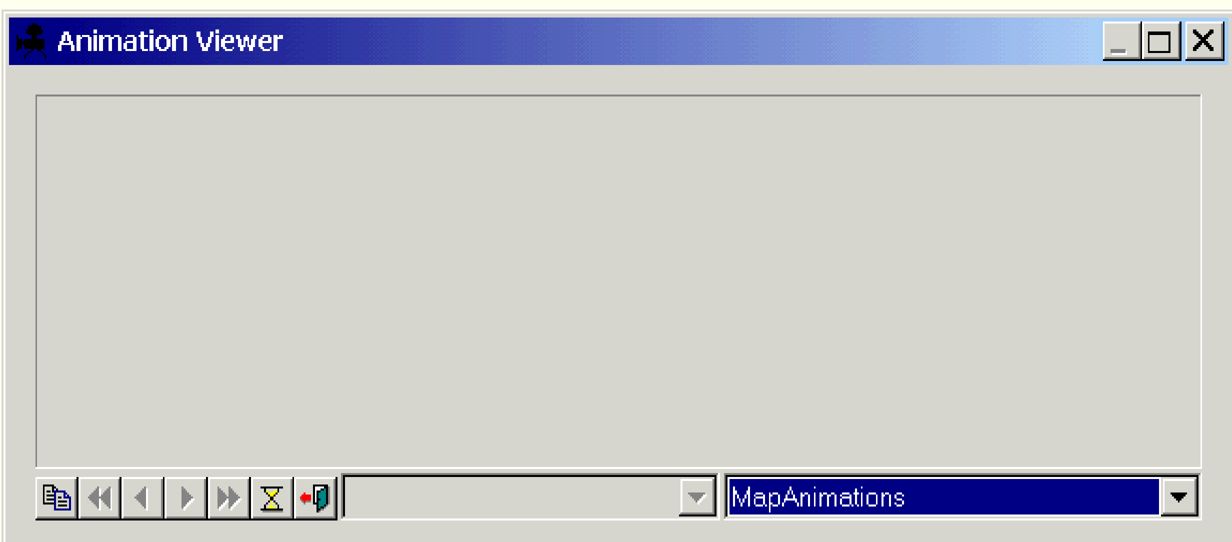
Other Related Topics - Map Display - Label Properties**,** Map Display Classification**,** Map Display Methodology

Animation Viewer

The Animation Viewer is used to retrieve and display map animations created and saved through the **Animation Editor** found under the **InfoWater Control Center -> InfoWater** button **-> Tools** menu and the output Status icon on your **InfoWater Output** toolbar. Map animations display simulation results for EPS timesteps in succession, thereby creating a "movie" effect that allows the user to see saved and view simulation results at any point in the future.

Once an animation is saved it can be viewed through the **Animation Viewer** dialog box. To access this box, from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select **Animation Viewer**. Using the drop-down list at the bottom of the Animation Viewer dialog box, the user can select any saved movie to play a stored animation. To learn more about creating Map Animations refer to the section on the Animation Methodology.

Click on any portion below to learn more.

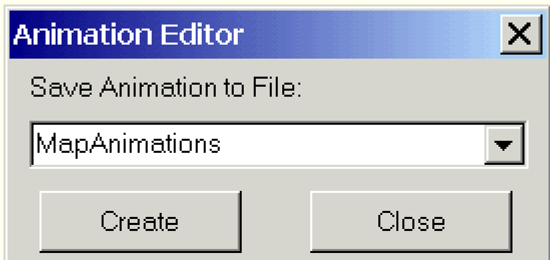


Other Related Topics - Animation Editor**,** Animation Viewer**,** Animation Viewer Methodology**,** Elevation Profile**,** Hydraulic

Calculator

Animation Editor

Use this to create New Animation Sets to display InfoWater Map renderings. The Files created here need to be selected when viewing the animations through your Animation Viewer. Specify the new animation file name and click on the **Create** button to create a new map animation file. To learn more about creating Map Animations refer to the section on the Animation Methodology.



Other Related Topics - Animation Viewer, Animation Viewer Methodology, Elevation Profile, Hydraulic Calculator

Elevation Profile

Use this command to generate ASCII text of the profile between two selected points while a known contour layer is active. Once the profile has been made, expand the Message Board to see the text generated from the profile. Highlight the text in InfoWater and then hold the CRTL-C keys down on your keyboard to copy the highlighted text to the Windows clipboard. Open a third party software package like Microsoft Excel and the then paste the text. This command is helpful for anyone wishing to determine elevation relief for existing pipes or new pipeline construction.

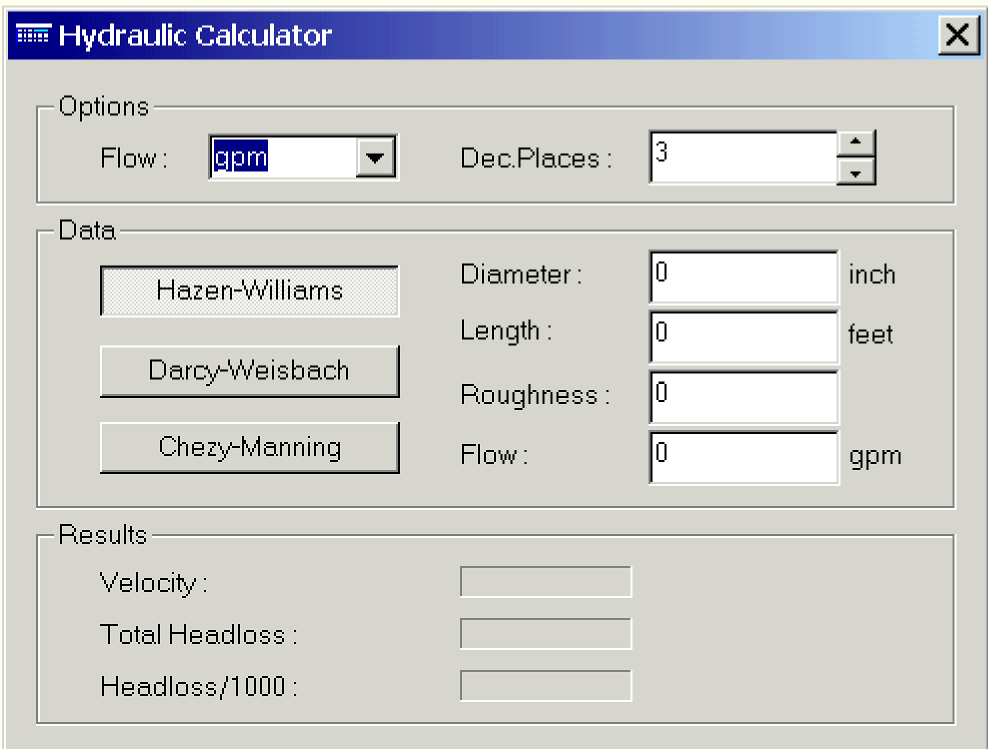
Other Related Topics - Animation Editor, Animation Viewer, Animation Viewer Methodology, Hydraulic Calculator

---

Hydraulic Calculator

The Hydraulic Calculator is used to determine the velocity and headloss for any given pipe in the distribution system. Merely specify the pipe variables and the results are automatically generated.

To view the Hydraulic Calculator, from the **InfoWater Control Center -> InfoWater** button **-> Tools** menu, select Hydraulic Calculator.

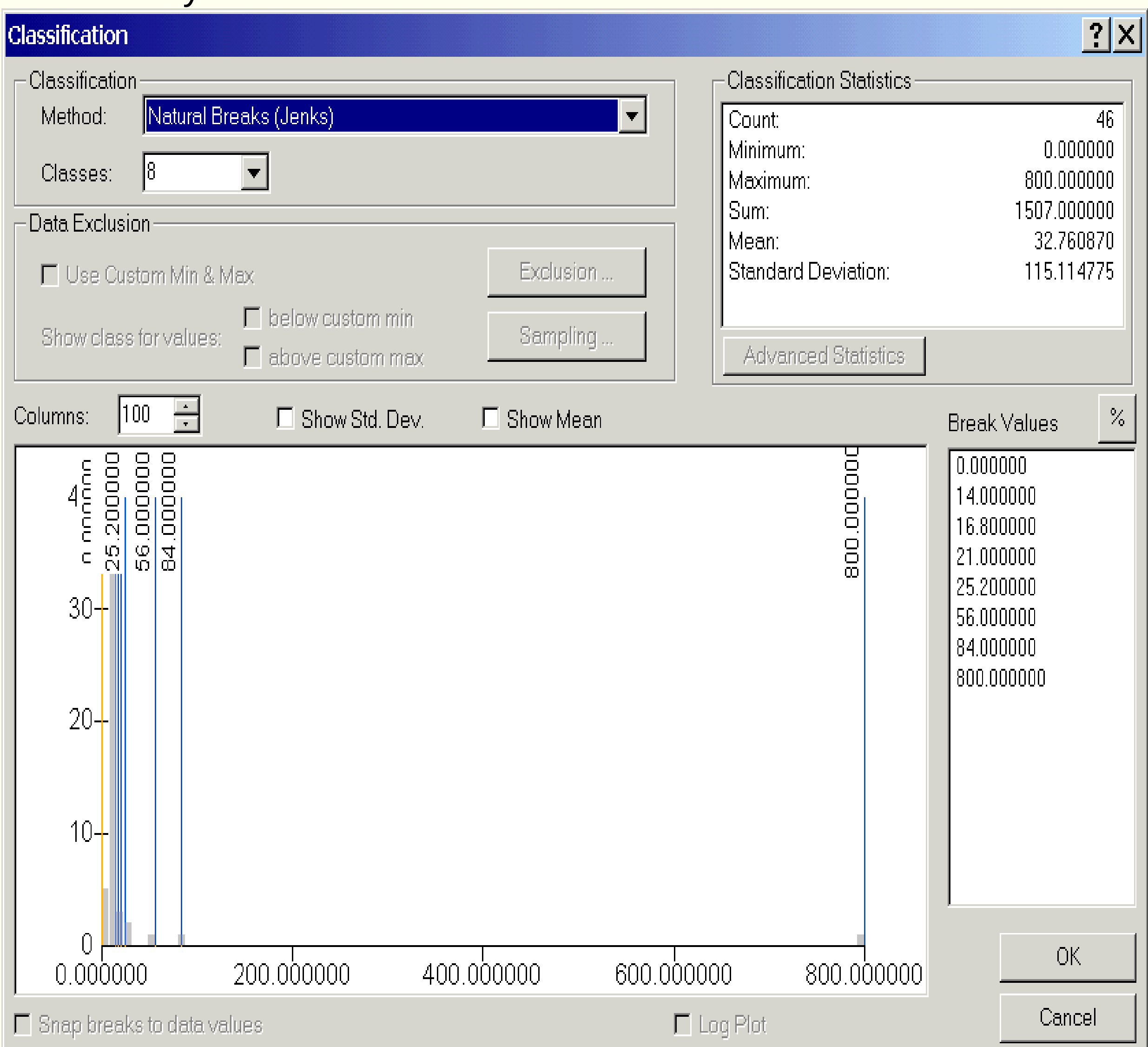


Other Related Topics - Animation Editor, Animation Viewer, Animation Viewer Methodology, Elevation Profile

Classification Manager

The Classification Manager can be launched by clicking on the **Classify** button on your **Map Display Manager**. The Classification Manager may be used to specify your classification method. Also classification statistics and the break values are displayed.

Click on any section below to learn more:

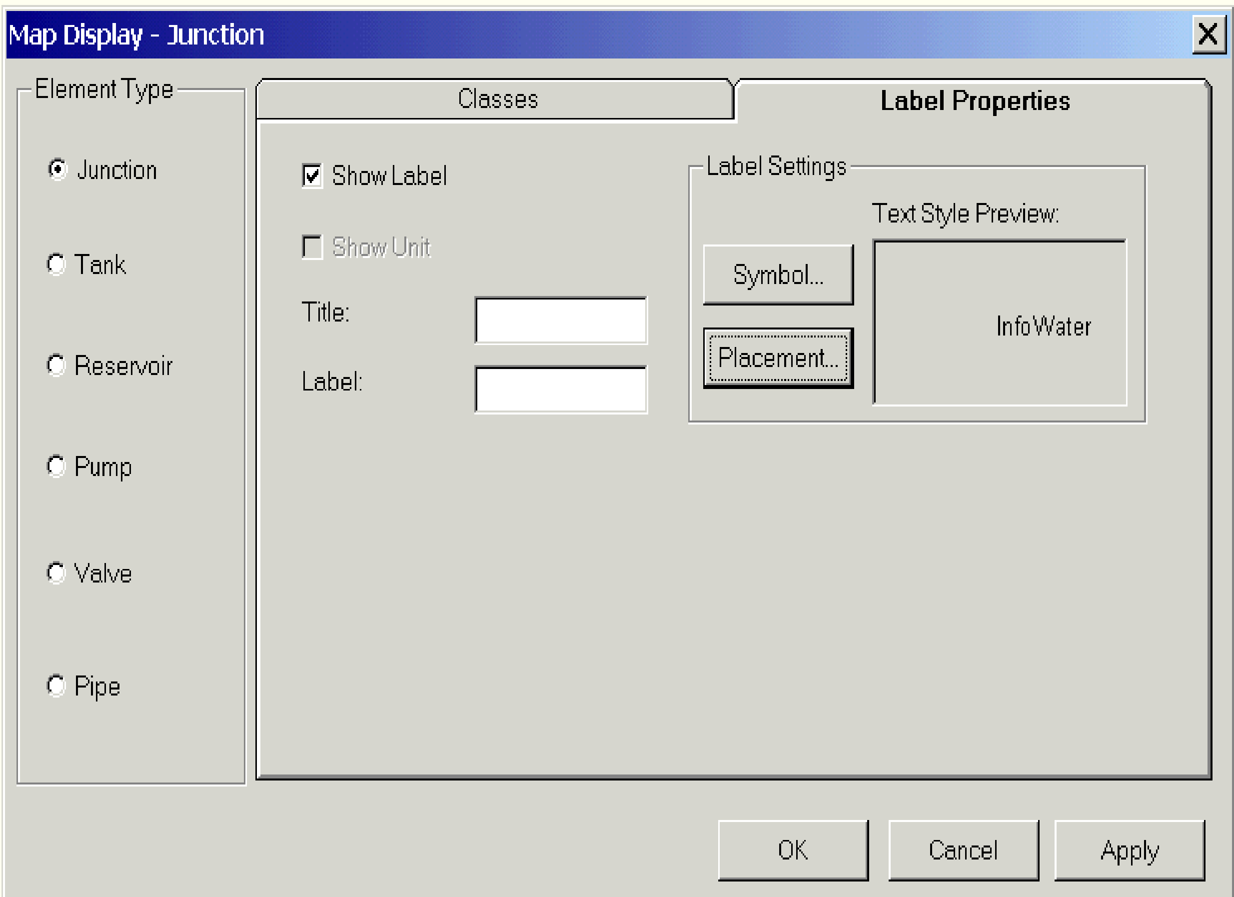


Other Related Topics - Map Display - Label Properties, Map Display Main Dialog Box, Map Display Methodology

Map Display - Label Properties

The InfoWater Map Display feature may be used to annotate your map. To learn more about color coding and/or changing symbol sizes click here.

Click on any section below to learn more:



Note: Color Coding will **NOT** work for a domain. InfoWater will always color code the **ENTIRE** network.

Other Related Topics - Map Display Classification, Map Display Main Dialog Box, Map Display Methodology