rNd PipeWeb

User's guide

Version 6.1

February 2025

Welcome to RND PipeWeb

RND PipeWeb is a powerful software application for hydraulic analysis of liquid flow in pipe networks. It has a comprehensive graphical user interface that will appeal to chemical engineers and a versatile simulation engine. Apart from steady state flow conditions, the application also supports analysis of transient hydraulic shock effects, also known as water hammer, as well as time-dependent (extended period) simulation of gradual changes, such as liquid levels in tanks and tracer transport.

RND PipeWeb is a full-fledged *Microsoft Windows* application, suitable for analysis of piping networks in such applications as the process industry and drinking water distribution. Among the many supportive features that make modeling with PipeWeb efficient and enjoyable are node and link template repositories, advanced result animation tools, and incorporated tables for ANSI pipe dimensions and minor loss coefficients.

In this document, the names of user interface controls are written in bold, where suitable. The names of object properties in PipeWeb are written with a capital start letter, whereas property values are generally displayed in italic. Internal document hyperlinks are indicated by an alternative text color.

Licensing and support

RND PipeWeb can be used for any purpose at no monetary cost for an unlimited period of time. Purchase of a PipeWeb pro license removes a constraint in the maximum project size and grants access to excellent user support service for one year. The software is available for download at pipeweb.rnd.nl.

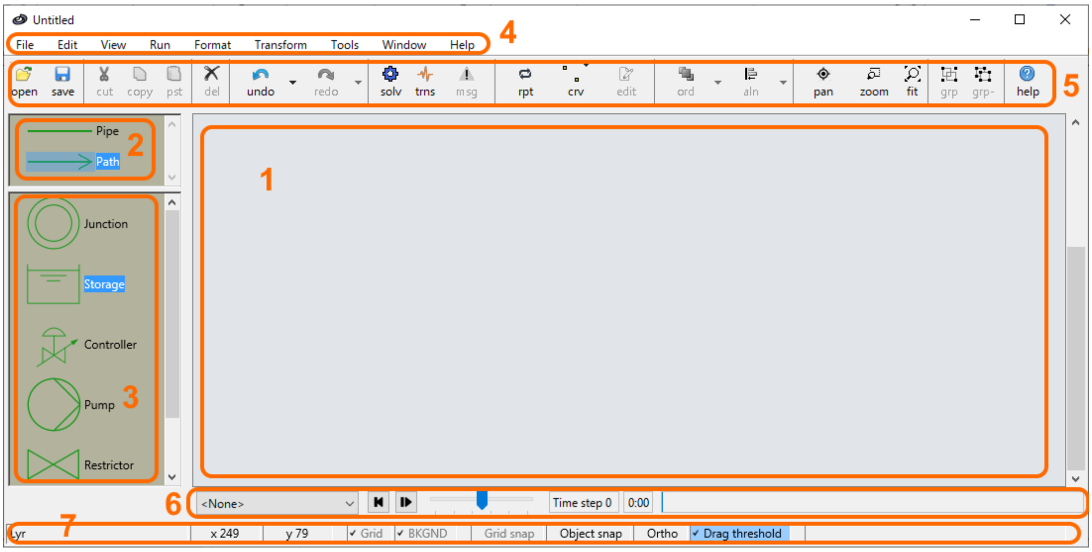
System requirements

Operating system: *Microsoft Windows*™ 10 or higher 64-bit.

Contents

1. The application workspace

RND PipeWeb simulation models are constructed and operated in the workspace window. The workspace window is the principal user interface of the application. It is opened when the application is started, and the program exits when the workspace window is closed.



This section starts with a discussion of the Application properties of the application and has separate topics for each of the elements of the workspace window:

1. Viewport — The model construction environment.
2. Link toolbox — Contains templates for link creation.
3. Node toolbox — Contains templates for node creation.
4. Menu bar — Categorized list of commands.
5. Design toolbar — Buttons for frequently used menu bar commands.
6. Contour plot toolbar — Controls hydraulic solver actions and display of color contours associated with hydraulic results
7. Status bar — Displays the current layer of the viewport and controls various editing options.

Additional characteristics of the behavior of the workspace are given in topics:

- Layers
- Blocks and fusions
- Curve types
- Viewport pages

1.1. Application properties

RND PipeWeb is a stand-alone application. Multiple instances of PipeWeb may be started in the operating system, but you can only operate a single project in a PipeWeb instance.

Projects are created, opened and saved by the usual commands **New project**, **New Civil project**, **Open project**, **Save project** and **Save project as**, which are available in the File menu of the Main window. Double-clicking a file extension associated with a RND PipeWeb file in Windows Explorer will open the file in a new instance of the application. Drag-dropping a file from Windows Explorer onto the viewport area of the Main window will open the file, after prompting to save the project being replaced. A PipeWeb instance is closed by the Exit command on the File menu or by clicking on the ‘X’-button on the title bar of the Main window.

RND PipeWeb project file types

There are three PipeWeb project file types:

- The PipeWeb project file (extension: *.Pipeweb_project)
- The PipeWeb node toolbox file (extension: *. PipewebNodeToolbox)
- The PipeWeb link toolbox file (extension: *. PipewebLinkToolbox)

The PipeWeb project file type supports storage of all information that can be contained in an RND PipeWeb simulation project, including picture components and all simulation data.

Project types

RND PipeWeb supports two project types. The standard project type is opened by the **New project** command in the File menu. Alternatively, the **New epanet project** command in the File menu creates a project of which the model configuration is consistent with the original EPANET application as introduced by the U.S. Environmental Protection Agency. The latter is intended for modelling of water distribution networks, whereas the standard project type was developed with chemical process engineering applications in mind. Both project types employ the open source EPANET solver for hydraulic analysis. The transient solver for shockwave simulation is available only for the RND PipeWeb standard project type.

1.2. Viewport

A PipeWeb model is constructed and operated in the viewport area. It supports the following operations, using the mouse, the Menu bar, the toolbar, or the contextual menu of the viewport:

- Add and delete nodes and links.
- Cut, copy & paste nodes and links.
- Group/Ungroup objects.
- Merge objects into Blocks and Fusions.
- Rearrange nodes.
- Resize and edit nodes.
- Edit link curves and change link connections
- Change the display order of nodes.
- Assign an object layer.
- Create link templates and node templates
- Operate simulations
- Create a page background

Node creation

Nodes are added to the model by drag-dropping a template from the Node toolbox onto the **viewport**, using the mouse. Node objects added to the viewport are placed in the Active layer. They are represented by an image and a name tag.

Link creation

Link objects are depicted on the viewport by a curve and an optional name tag. Link creation is started by selecting a link template from the Link Toolbox. The finger mouse pointer indicates that the viewport is ready to receive instructions for construction of a new link object. If the mouse hovers over a connectable region of a node, the possibility for making a link connection is indicated by a change of the mouse pointer. The connection location on the node is indicated by a diamond or triangle marker. The connection is established by a left mouse button click on the connection location. The location at which a link connects to a node is called a knot.

After the link start point has been created, each click on the viewport extends the curve path by addition of a control point, indicated by a small circle. Link construction is completed by creating a termination point using the same routine as for the start point. New link objects created on the viewport are placed in the Active layer. The curve type of a new link object is determined by the selected **Curve type** item in the Format menu. The orientation of the last segment of a link curve of type orthogonal can be switched between horizontal and vertical by holding down the space bar while the mouse hovers over a (non-auto arranged) destination connection point.

A link end point connected to a newly created junction node can be made by clicking anywhere on the viewport while holding the **Ctrl**-key pressed.

A node connected to the upstream side of a link is named the **origin node** of the link, while the link is referred to as a **downstream** link of the node. The node connected to the downstream side of the link is the **destination node** of the link, while the link is referred to as an **upstream** link of the node. Nodes can have an unlimited number of connected links.

Object selection
Selected nodes and links are indicated in the viewport by a semi-transparent mask over the object in the selection color of the operating system. Selection of a single object is simply done by clicking on it in the **viewport**. The selection result depends on whether or not the **Ctrl key** and **Shift key** are pressed during the selection operation.

Shift key state	***Ctrl key state***	***Selection result***
released	pressed	remove from selection
pressed	released	add to the selection
released	released	start new selection

To select/unselect several objects at the same time: Click with the left mouse button on an empty location on the viewport and move the mouse while keeping the mouse button pressed. A rectangle is displayed to indicate the operative region. The selection operation is executed when the mouse button is released. The result of this rectangular selection depends on the horizontal sweep direction. If the mouse was moved from left to right, only objects that lie completely inside the rectangle are affected. If the mouse is moved from right to left objects touching the rectangle are selected as well. The effect of pressing the Ctrl and **Shift** keys for to this selection method is the same as for selection of single objects by mouse clicks.

Object rearrangement and duplication
The position of nodes on the viewport can be changed by dragging their image across the viewport. If the control-button on the keyboard is pressed while starting the drag operation, the selected nodes and selected links connected to them are duplicated and subjected to the drag operation. As opposed to the copy/paste menu commands, duplication by mouse drag operation does not store the objects in the clipboard of the operating system.

Object editing
The **Edit objects** command of the Edit menu toggles the edit mode for selected objects in the viewport. In the edit mode, bounding box sizing handles and curve control points are displayed which can be used to change the objects in the viewport with mouse operations. The following object elements are available for modification:

- Height and width of the node icon
- Contour and connectivity of the link curve
- Tag anchor location of the link
- Size and location of the symbol text area
- Contour of the symbol curve.

Specific element types may be locked for all edited objects using the Edit options command of the Edit menu. Resizing handles are displayed as small rectangles on the edges of the bounding box of the element. Curve control points are displayed as small circles and the link tag location is indicated by an encircled cross sign. These markers can be selected in the same manner as objects are selected on the viewport by mouse operations. Objects are modified by dragging the selected markers using the mouse. The aspect ratio of resizing actions is preserved, unless **Ctrl key** is pressed. Curve control points can be duplicated by dragging a selected control point while keeping the **Ctrl key** pressed. Pressing the **Del key** removes the selected control points of the curve. The curve type of all edited node and link curves can be changed by selecting the desired **Curve type** item in the Format menu.

Links can be detached from connected nodes by dragging the link end point away from the node. Reversely, links can be attached to a node by dropping the end point on a suitable connection knot of a node. The knot is indicated by a marker in the shape of a diamond or a triangle. A vertical line inside a circle distinguishes the termination point of a curve from the start point. A polygon or spline curve with more than two control points is fully rescaled if one of the end points is dragged in the viewport. Alternatively, if the space bar is held down at the start of the drag operation, only the curve end point is displaced. For orthogonal link curves, the orientation of the first and last segment of a link can be switched between horizontal and vertical by holding down the space bar while the mouse hovers over a link end point other than an auto arranged node connection knot. Pressing the **Esc key** terminates the edit mode without storing any changes made to the selected objects.

Viewport manipulation

The viewport may be scaled and translated by dragging the mouse. The viewport is translated if the mouse is moved while keeping the **right button** pressed. The viewport is dynamically scaled if the **Ctrl key** is held down at the same time. The same behavior is obtained for the left button if the **Pan button** on the Toolbar is toggled on. In that case the right button scales the viewport without pressing the Ctrl key and the mouse cannot be used for object selection.

The viewport may also be scaled by the **Extend viewport** command on the View menu and by the **Zoom button** on the Toolbar. The Zoom button enables scaling to a region specified by dragging the mouse. If the mouse is **dragged** in the Viewport after pressing the zoom button, a rectangle is drawn of the same aspect ratio as the Viewport, centered around the mouse-down location. The size of the rectangle corresponds to the viewport scaling factor applied when the mouse button is released. The scaling direction depends on the direction of the mouse displacement. If the mouse was moved from to the right the viewport scale range is decreased to the rectangle area scale range. If the mouse button was moved to the left, the viewport is expanded to the range that results from shrinking the Viewport area into the rectangle area. For a small mouse displacement, the viewport scaling factor remains unchanged and the center of the viewport is translated to the mouse location.

Viewport contextual menu

The contextual menu of the viewport is opened by a right mouse button click. The availability commands of this pop-up menu depends on the selected objects in the viewport and on the applied model properties. All commands that may appear in the contextual menu of the viewport are listed below, with a brief description of their function.

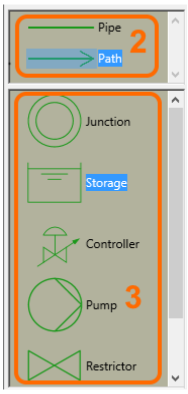
• **Cut/Copy/Paste**	Edit menu command clones
• **Editor options**	Clone of the corresponding options arrangement in the Edit menu.
• **Reset symbol text size**	Restores the default symbol text size of the edited nodes of which all text area sizing handles have been selected in object editing.
• **Align to**	Performs the action of the corresponding Edit menu item with the selected node indicated by the contextual mouse click location as reference location.
• **Reset chart scale range**	Restores the default scale range of the selected chart node indicated by the contextual mouse click location.
• **Open fusion window**	Performs the action of the corresponding Format menu Blocks command for the selected block or

fusion node indicated by the contextual mouse click location.

- **Create node template** Performs the action of the corresponding Transform menu item for the selected node indicated by the contextual mouse click location.
- **Create link template** Performs the action of the corresponding Transform menu item for the selected link indicated by the contextual mouse click location..
- **Move arranged link up** Changes the auto-arranged connection order at the node for the selected link and link connection side indicated by the contextual mouse click location.
- **Move arranged link down** Changes the auto-arranged connection order at the node for the selected link and link connection side indicated by the contextual mouse click location.
- **Insert page reference** Performs the action of the corresponding Transform menu item for the selected link indicated by the contextual mouse click location, regardless of the page location of the link origin and destination nodes.
- **Remove page reference** Performs the action of the corresponding Transform menu item for the selected page reference node indicated by the contextual mouse click location.
- **Remove empty page** Performs the action of the corresponding Transform menu item for an empty page indicated by the contextual mouse click location.
-

1.3. Toolboxes

Toolboxes are used to create objects in the viewport. There is a toolbox for link templates (Item 2 in the figure) and a toolbox for node templates (Item 3 in the figure). The toolboxes may be hidden by unchecking the **Toolbox** item on the View menu.



The toolboxes contain one or several default templates. Node and Link objects in the viewport can be stored as new template in the toolbox by the **Create template** command from the Transform menu or the contextual menu of a selected object. A new template inherits all properties from the source object. Template properties cannot be changed.

The default templates of the toolboxes include nodes and links representing EPANET Physical Components. In addition, the default node template *Text* can be used to add annotations to the worksheet.

Link toolbox

The link toolbox contains link templates. Links created on the viewport inherit all properties from the selected template in the link toolbox.

Node toolbox

The Node toolbox (Item 3) contains node templates. A new node object is created from a template by dragging a template from the toolbox and dropping it on the viewport.

1.3.1. Toolbox properties

The template images in the toolbox can be rearranged by drag-drop mouse operation.
The toolbox properties can be modified by commands of the contextual menu of the toolbox. Besides the regular **Cut**, **Copy**, **Paste** and **Delete** items, the Toolboxes have the following contextual menu item:

- Add category — Creates a new category for template categorization.
- Incrm. col. count — Adds a column for template organization in columns.
- Decrm. col. count — Removes the right-most column.
- Select children — Selects all viewport objects created by the selected template.
- Properties — Opens a window which allows modification of the toolbox properties.
- Config/Write — Saves the toolbox properties and content to disk.
- Config/Read — Reads the toolbox properties and content from disk.

- Config/Import Adds toolbox content from disk to the present configuration.
- Config/Restore Restores the default set of Toolbox templates.

1.4. Menu bar & Toolbar

The Menu bar is located right underneath the title bar of the Main window. It has a series of commands grouped into drop-down menus. The most frequently used Menu bar commands are also available as a button on the Toolbar, which is located under the menu bar.

The Menu bar commands are listed below, together with the corresponding Toolbar button icon, if existing, and a brief description of their function.

File menu

	New project	Starts a new project and discards the current.
	New epanet project	Starts a new project with epanet model configuration and discards the current project.
	Open project	Opens a project file.
	Save project	Saves the current project.
	Save project as	Saves the current project after prompting for a file name.
	Page setup	Opens the Page setup selection menu.
	Print	Prints the project to a printer. Requires a completed procedure for the above-mentioned page setup menu item.
	Exit	Closes the application without saving.

Edit menu

	Undo	Undo the last mutation to the project. Clicking on the drop-down arrow opens a mutations list to select an item from.
	Redo	Redo the last undone mutation to the project. Clicking on the drop-down arrow opens a list of undone mutations to select an item from.
	Repeat	Toggles the Repeat mode for object creation
	Cut	Removes selected objects from the viewport and stores them in the Windows clipboard.
	Copy	Copy the selected objects the viewport to the Windows clipboard. Also available as pop-up command on the viewport.
	Paste	Paste objects from the Windows clipboard to the viewport. Also available as pop-up command on the viewport.
	Delete	Delete objects selected in the viewport.
	Edit objects	Toggles the viewport object editing mode
	Editor options	Toggles editing of specific node elements, including symbol curves and text area size, node size and link tag anchor location
	Find	Opens the Search utility to select nodes by string comparison.
	Select	Selects specific object types in the viewport.
	Unselect	Unselects specific object types in the viewport.

View menu

	Extend viewport scale	Rescales the viewport to fit the entire model.
	Scrollbars	Determines when the viewport scrollbars are displayed. Selecting the Auto option displays the scrollbars only if the model does not fit into the viewport.
	Toolbox	Hides the Toolbox when unchecked.
	Toolbars	Hides a specific toolbar when unchecked.

Run menu

	Solve hydraulics	Resets the simulation run, and solves the hydraulic equation set for the model. Also available as button on the Hydraulics toolbar.
	Run transient analysis	Performs a transient simulation. Requires a successfully completed hydraulic solution. The transient simulation solver process can also be started, monitored and interrupted on the Hydraulics toolbar.
	Clear data	Discards all hydraulic and transient simulation data.

Format Menu

	Align	Allows alignment of selected node objects.
	Order	Allows adjustment of the node z-order. Newly created nodes are placed on top of existing nodes.
	Group objects	Joins the selected objects in the viewport into a group, forcing subsequent joint selection of all group members.
	Ungroup objects	Breaks up all selected groups in the viewport.
	Curve type	Controls the curve type for new links and for curve editing. The selected type is reflected by the menu icon. The icon with no line indicates the link template default or the value <*various*>.
	Blocks	Manages blocks and fusions.

Transform menu

	Invert links direction	Inverts the direction of the links selected on the worksheet
	Make link chain unidirectional	Realigns the direction of all of all selected links, provided the links selected form a continuous trajectory.
	Swap link name	Swaps the name properties of exactly two selected links
	Reset symbol text size	Adjusts the text area of symbol nodes to fill the symbol shape.
	Create link templates	Creates templates in the toolbox for all link objects selected in the viewport.
	Create node templates	Creates templates in the toolbox for all node objects selected in the viewport.
	Insert page reference	Inserts a page reference symbol pair on the selected links for which origin and destination nodes are on different pages . Requires a completed Page setup procedure.
	Remove page reference	Removes the selected page reference symbol pairs. Requires a completed Page setup procedure.
	Remove empty pages	Removes all pages that do not contain any node or link objects. Requires a completed Page setup procedure.

Tools menu

	Configuration	Opens the Configuration window, as will double-clicking the viewport.
	Pictures	Opens the Picture management console.
	Layers	Activates the Layer control panel.
	Page background editor	Opens the background editor window. Requires a completed Page setup procedure.
	Preferences	Opens the Preferences utility to change preference

		settings of the application and the project.

Window menu

	Background	Changes the background color of the viewport.
	Arrange windows	Opens the Window arrangement utility.
	Windows list	Presents a list of all open window titles. Clicking on an item in the list activates the window.

Help menu

	Getting started	Opens the Getting started section of the help document.
	PipeWeb help	Opens the help document.
	PipeWeb website	Opens the PipeWeb website in the default internet browser.
	Check for updates	Connects to the internet and checks if a more recent PipeWeb release is available for download.
	About	Displays the application's introductory splash window. Click the splash window to close it.

Toolbar only commands

	Pan	Toggles left mouse button viewport manipulation.
	Zoom	Toggles the zoom function for viewport manipulation.

1.5. Hydraulics toolbar

The Hydraulics toolbar is located beneath the viewport in the workspace window. The toolbar is used to start the hydraulic and transient solvers and to control results review by contour plots.

1.5.1. Solver control

The set of three buttons on the left represent solver control commands, which are also available on the Run menu. The left button starts the hydraulic solver. The middle button starts or cancels the transient solver procedure. The third button toggles message display on and off. The icon color of the latter reflects the content of the message list: *Orange*: List has critical messages which obstruct the solver procedure; *Yellow*: List has non-critical warnings; *Green*: List has info only, no warnings; *None* (disabled): List undetermined or no messages. The warnings list displays messages of either the hydraulic or the transient solver, whichever was last operated. The list is shown on top of the viewport and may be closed also by pressing the **Escape** key.

1.5.2. Contour plot properties

Contour plots mark a dedicated region of the default node icon on the viewport by a color associated with a range of hydraulic result values for a selected node parameter. A color legend displayed on the viewport indicates the applied parameter value ranges. A set of user-controlled options for the display style of the contour map in the viewport are discussed in section Reporting.

The toolbar allows selection of the node parameter and a scheduling time point for the contour plot in the viewport. It also controls contour plot animation for schedule properties.

The drop-down menu at the left, next to the solver command buttons, is used to select a node parameter for contour mapping. The time point for contour mapping can be adjusted using the mouse in the progress bar at the right. The forward button starts or pauses the animation, with the slider object in the middle controlling the animation speed. The animation stops at the last schedule time, but can be put into a continuous loop using the contextual menu of the progress bar. The rewind button returns the animation to the initial time point. Finally, two text labels on the toolbar indicate the time point selected for contour display.

1.6. Status bar

The Status bar is located at the bottom edge of the window and is divided into panels, as indicated in the figure.

Layer pane

The Layer pane (Item 1) displays the name of the Active layer. Clicking the layer panel activates the Layer control panel.

Mouse location panel

The mouse location panel (item 2) displays the mouse x/y-coordinates on the viewport

Viewport options panel.

The viewport options panel (item 3) contains two toggle buttons. The **Grid** button toggles the display of the Page grid. The **BKGND** button toggles the display of the viewport Page background.

Object snap panel

The object snap panel (item 4) contains a set of four toggle buttons, which aid precise positioning of objects on the viewport. The **Grid snap** button enables snapping to the viewport page grid. The **Object snap** button enables snapping of node edges end link end points. The **Ortho button** results into preservation of vertical or horizontal coordinates of displaced viewport objects. The **Drag threshold** button implements a threshold distance for mouse drag operations, which can be used to prevent unintentional displacement of objects in the viewport by small mouse movements. The threshold also applies to viewport translation by mouse dragging.

1.7. Layers

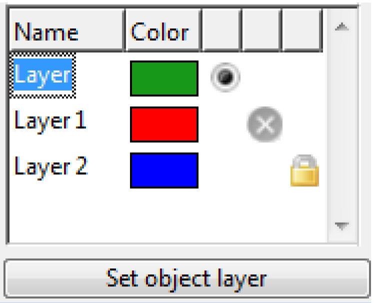
The node and link objects on the viewport are organized in layers. The layers have a specified color, and may be **locked** and **hidden**. The main purpose of layers is to cluster objects that are related in some meaningful way, for example by a shared function in the model. The organization of a project into layers offers many advantages to the project designer. The ability to hide layers and to apply different colors to layers makes it easier to visually distinguish objects in the viewport. Objects in locked layers cannot be selected in the viewport.

Active layer

New objects created on the viewport are placed in the layer that has been assigned **active layer**. The active or current layer can be identified in the Status bar and by the color of the default templates in the Toolboxes.

Layer control panel

The layers defined within project are managed by the Layer control panel, which is activated by a mouse click on the left-most panel of the Status bar or by the **Layers** command of the Tools menu. The Layer control panel is a pop-up control that presents a list of all layers defined for the project. It enables the creation and deletion of layers, modification of layer properties, and assignment of the current layer. It is also used to place objects on the viewport in a different layer.



The column labeled ***Name*** displays the layer name. The layer name can be edited in-cell, by double clicking the name field or by pressing a key on the keyboard. The assigned name must be unique and has the same format requirements as the Name property of nodes, links and components. The column labeled **Color** determines the default display color for objects contained in the layer. A click on the Color column activates a color selection dialog. The radio button in the third column identifies the **Active layer** of the viewport. The two other columns present additional layer properties indicated by an icon. The cross-out icon signifies that the layer is **hidden**. The padlock icon signifies that the layer is **locked**. A mouse-click on either of the two leftmost columns toggles the corresponding layer property.

The **Set object layer** button appears on the Status bar when the Layer list is activated. By pressing this button, all selected node and link objects on the viewport are placed in the selected layer in the Layer list. New layers are created by the **Add layer** command of the contextual menu of the layer list. The **Del key** deletes the selected layers in the Layer list.

While the layer panel is visible, the window's Menu bar & Toolbar are disabled, as are all other windows of the application. The Layer tool is deactivated by clicking on the **viewport**, or pressing the **Escape key** on the keyboard.

1.8. Blocks and fusions

Blocks and fusion are named object sets that act as a single object. You can use them to create composed templates and compress specific parts of a model. They can help you save time and make your model easier to view. Block and Fusion nodes can be stored as node templates in the Toolbox.

Blocks

A set of node and link objects selected in the viewport can be joined into a block node. A block node represents the block members collectively by a single node in the viewport. The constituents of the block node cannot be individually selected, rearranged, edited or modified. Links can still be connected to the individual nodes contained in a block.

The **Create block** command of the Blocks Format menu merges all selected objects on the viewport, including block nodes, into a new block node. The **Explode block** command reverses this transformation for all selected block nodes on the viewport.

Fusions

The **Fuse blocks** command of the Blocks Format menu transforms the selected block nodes in the viewport into fused block nodes. Fused block nodes are useful to represent a section of a project by a single node object. The default image of a fusion node is slightly different from the default node image.

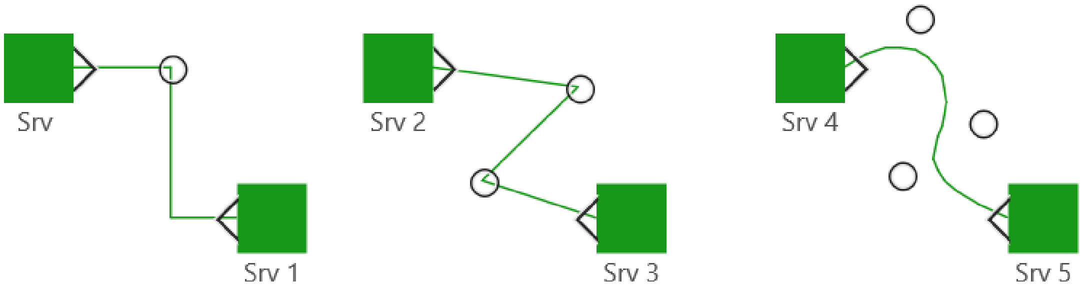
A link that is connected to a fused node, but not fused itself, is shown as connected to the fusion node in the viewport. Links can be connected to a predetermined destination or origin node contained in a fusion. The **Unfuse blocks** command transforms all selected fused block nodes back to a regular block node.

Fusion window

A fused block node may be displayed in a separate Fusion window. The **Open fusion window** command of the Blocks Format menu opens new fusion windows for all selected block and fusion nodes in the viewport. The fusion window is identical to the Workspace window, but does not have the File menu and some items in the Help menu. The fusion window permits modification of the block node by the same methods used in the viewport of the workspace window and supports all simulation Simulation display features. Changes made to the Layer control panel and the simulation control toolbar of the workspace window and fusion windows apply to all open windows. The **Close block window** command closes the block windows for all selected block nodes in the viewport.

1.9. Link curve types

The shape of a link curves in the viewport is determined by the curve format type and a series of control points. The application supports the curve types *Orthogonal, Polygon* and *Spline*.



Orthogonal **Polygon** **Spline**

The curve types exist for display purposes only: they have no logical distinctions in a model.

1.9.1. Link curve type selection

The curve type of a link is determined by the **Curve type** item in the Format Menu. This control is available both for new links and for object editing.

1.10. Viewport pages

Completion of the **Page setup** dialog of the File menu divides the viewport into pages. The specified page properties may be reviewed by the Page layout item of the Preferences console

Page grid

The Grid size item of the Preferences console allows definition of a page grid as reference for the arrangement of objects on a page. The page grid is only shown on viewport pages that have at least one node or link object. The grid can be hidden by the **Grid** button on the Viewport options panel.

Drawings

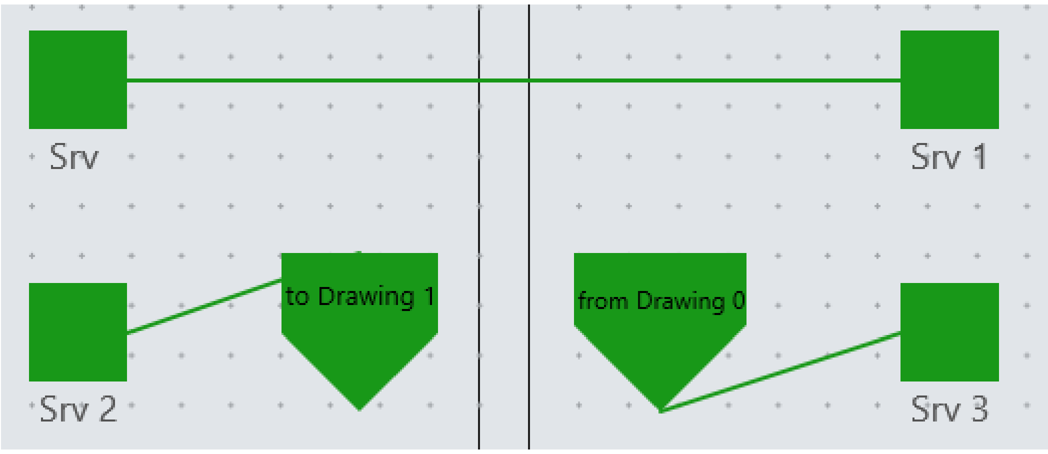
Each page on the viewport represents a drawing. The properties of a drawing are specified on the Specs tab of the Model console of the configuration window.

Page background

The **Page background editor** item on the Tools menu opens the background editor. The page background is drafted in the viewport of the background editor tool similar to the simulation model in the Workspace window, but cannot include link objects. The page background is displayed on every page of the workspace viewport. The properties of individual drawings can be expressed by text markup for node objects in the background. The page background can be hidden by the **BKGND** button on the Viewport options panel. Note that objects placed on the page background are not part of the simulation model.

Page reference nodes

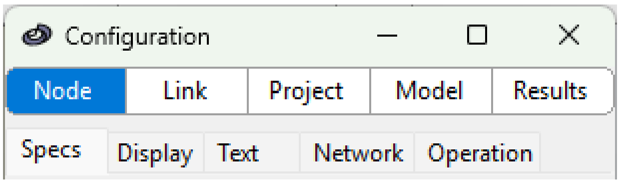
A page reference node pair presents a method to systematically arrange page crossing links. A link is transformed into a page referencing link by the **Insert page reference** item of the Transform menu. This transformation spits the link into two parts on the viewport, each connected to a newly created page reference node. The page transition is depicted by placing each page reference node on the matching source or destination page of the original link, which will consequently be represented on both pages by either of the two split parts.



The page reference nodes have no logical function in the simulation and the two link parts both represent the transformed link in the simulation. Simulation display is performed on both link elements. The page location of the page reference nodes can be expressed by text markup. The page reference construction is removed from a link by the **Remove page reference** item of the Transform menu.

2. Object configuration

The Configuration window provides access to the properties of objects on the viewport and many other elements of the project. The window is opened by the **Configuration** menu on the Tools menu or by double-clicking the viewport.



The category bar at the top of window selects the main category for which properties are displayed. Selecting a category displays the corresponding console with all relevant control items.

The properties configured by the console categories are:

Node	Node object properties
Link	Link object properties
Project	Properties of drawing objects or of the project itself
Model	General properties and properties of objects that are not represented on the viewport
Results	Simulation results and related information

This chapter discusses property configuration on the **Node**, **Link** and **Project** consoles. Properties of the model are discussed in chapter Model configuration and the review of results is covered in chapter Results.

Viewport context
The configuration window can be opened from the Workspace window and from a Fusion window. The window from which the configuration window is opened represents the viewport context. The Node, Link and Project consoles only display and configure properties of objects that have the selected state in the viewport. For the Project console, Drawings are considered to be selected if one or more objects on the drawing have been selected. If no drawings are selected, the Project console references the current project.

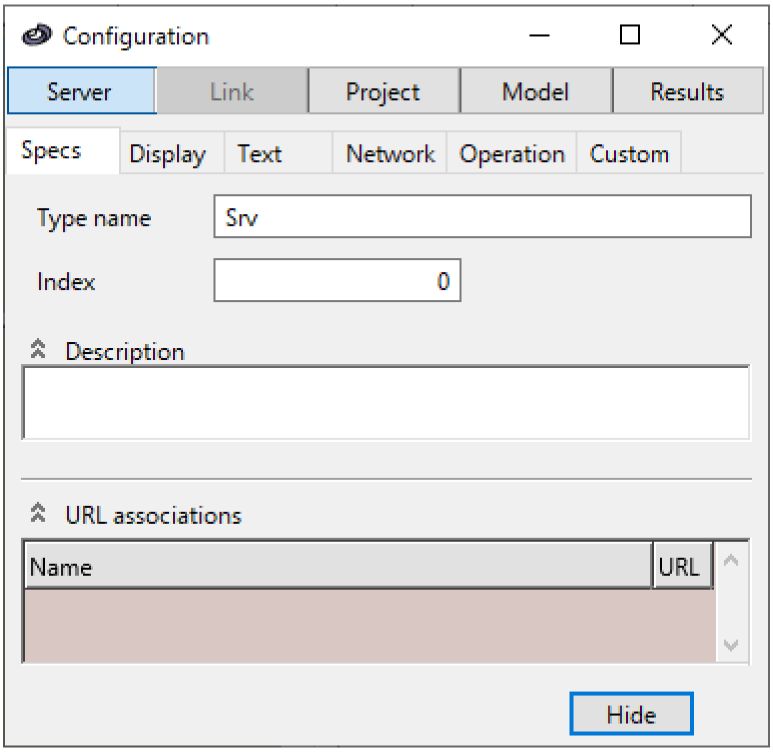
If the selected objects do not have a uniform value for a specific property, check boxes are grayed out, and pop menus are left empty, whereas in text fields and combo boxes the keyword <*various*> is displayed. In general, a control item on the console is disabled or hidden if it configures a property that does not affect any of the selected objects.

The Topics tab strip beneath the category bar generally selects a group of properties that are related in some way. The composition of this tab strip can vary between the different consoles. The next sections in this chapter are organized by the items categorized on the Topics tab strip:

- Specs
- Display
- History
- Text
- Network
- Operation

2.1. Specs

The **Specs tab** on the Node, Link and Project consoles presents general properties of selected nodes, links and drawings. If no drawings are selected, the Project console presents the properties of the project.



Type name

The Type name property may contain a maximum of 255 characters in the ranges A-Z, a-z, 0-9 and the underscore _ character. The property must start with an alphabetic character and is case-insensitive. The name of an object consists of type name followed by the non-zero value of the below-mentioned Index property. When the Type name property is altered, the Index property may change for any of the selected objects to render the name property unique in the project.

Index

The index property of an object in an integer number >0. The name of an object consists of the above-mentioned type name followed by the non-zero index value. The name of each object is unique in the project. When the index property is changed, indexing of the selected objects starts with the specified value, where the index value of non-selected objects with the same type name are skipped. The default tag property of an object is the type name followed by a space and the non-zero index.

Description

Descriptive text for the item.

URL associations

Displays a list of attached URLs to the selected viewport object, drawing or the project.

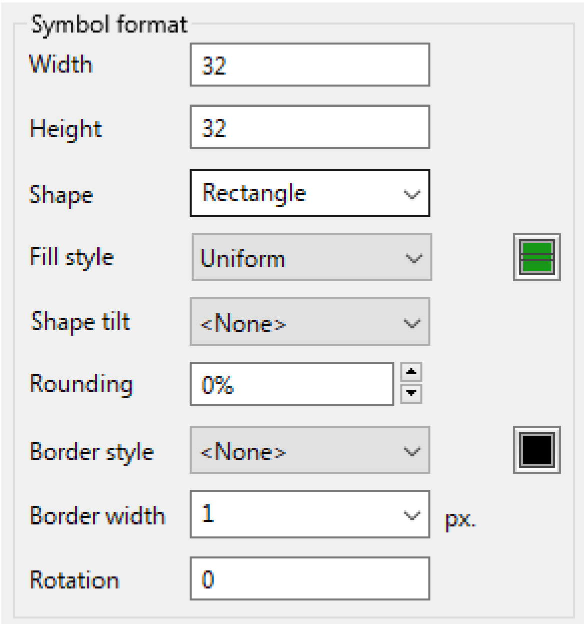
The **Add item** command of the contextual menu of the list adds a new untitled blank URL association to the list. The **Edit name** contextual menu command activates the edit mode of selected cell in the name column, as does pressing the space bar. The **Edit URL** contextual menu command activates the edit mode of selected cell in the URL column. The **Browse file** command allows opens a file selection dialog for assignment of a file URL to the selected item in the list. The **Show in folder** command opens a file URL in the system file browse. A **double-click** on the list opens the URL of the selected item with the appropriate system application.

2.2. Display

The **Display tab** on the Node and Link consoles presents the display properties of nodes and link objects.

2.2.1. Symbol format

A node object is represented on the viewport by a constructed symbol, consisting of a base shape and several other elements. The base shape can be an ellipse, a polygon or a chart. The shape can be solid or transparent, can be rounded and rotated and may have a border. The symbol can also have a curve, a picture and text displayed on top of the base shape. All symbol properties are controlled of the Display tab, except for the text properties, which are controlled on the Text tab.



Width

Determines the width of the node in the viewport. The value is specified in the value selected for the Project base unit. The command **Make symbols equilateral** of the right-click activated contextual menu of the text field adjusts the Width property of symbol nodes to obtain an aspect ratio that renders polygons equilateral and ellipses circular.

Height

Determines the height of the node in the viewport. The value is specified in the value selected for the Project base unit. The command **Make symbols equilateral** of the right-click activated contextual menu of the text field adjusts the Height property of symbol nodes to obtain an aspect ratio that renders polygons equilateral and ellipses circular.

Shape

Determines the base shape of the symbol. Selection of the *Chart* value furnishes the symbol with an additional element to display numerical data in a chart diagram. Pressing the adjacent toggle button **Config** enables configuration of the chart properties as described in the Chart Configuration section.

Fill style
Determines the fill style of the base shape. A click on the adjacent button opens a color section dialog that allows selection of the fill color. For the two Gradient values a second color button appears which represents the secondary gradient color.

Shape tilt
Tilts the symbol shape by 90-degree angle increments.

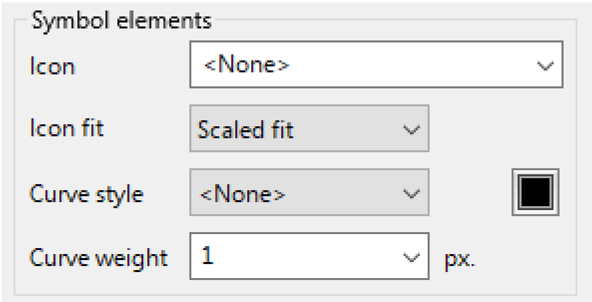
Rounding
Determines the corner rounding of the border within a range of 0 - 100%

Border style
Determines the border line style. A click on the adjacent button opens a color section dialog that allows selection of the border color.

Border Width
Determines the border width specified in mm.

Rotation
Determines the clockwise rotation angle of the symbol in degrees.

2.2.2. Symbol elements



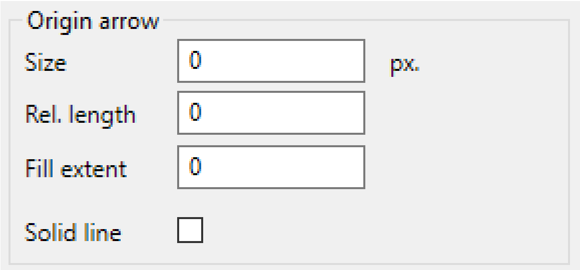
Icon
Determines the picture object to be displayed as icon in the node image. Type the name of the desired picture object or select it from the combo box.

Icon fit
Determines the arrangement of the above-mentioned icon in the symbol. Possible values are, *Center* (centered without size change), Scaled fit (scaled to fit the symbol with preservation of aspect ratio), *Filled fit* (stretched to fill the symbol), *Tile* (Repetitive tiling from top-left corner, cropped at left and bottom edges), *Shrink* (*center* and if larger than the symbol: *Filled fit*)

Curve style
Determines the curve line style. A click on the adjacent button opens a color section dialog that allows selection of the curve color.

Curve weight
Determines the curve width specified in the value selected for the Project base unit. The value *Hairline* represents the smallest value for the resolution of the selected printer.

2.2.1. Link arrow properties



Arrow size
Determines the distance of the arrow hand tip to the edge of the link curve, specified in mm or pixels.

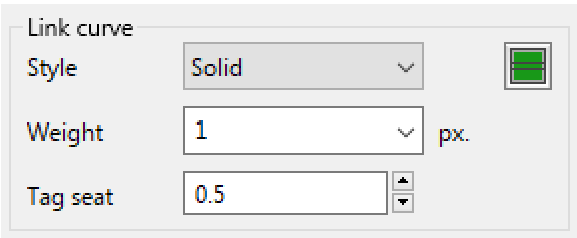
Rel. length
Determines the length of the arrow as fraction of the above-mentioned size property. The default value of 1.732 for the standard link templates is consistent with an equilateral triangular arrow head.

Fill extent
Determines the length of the solid arrow area along the curve as fraction of the above-mentioned relative length property

Solid line
Forces a solid line for the arrow outline for dotted and dashed curve styles

Serial
Determines whether or not arrow display is repeated for every curve segment of a segmented curve.

2.2.2. Link curve



Style
Determines the curve line style. A click on the adjacent button opens a color section dialog that allows selection of the curve color.

Weight
Determines the curve width specified in mm. The value *Hairline* represents the smallest value for the resolution of the selected printer.

Tag seat
Determines the tag location at the link curve in the range 0 to 1, where 1 denotes the destination location.

2.3. History

The **History tab** on the Project console presents revision information of the project and of the drawings in the project.

Revisions

Displays a list of revision numbers of the selected drawings in the project and of the project itself. A drawing is selected if any of the viewport objects in the drawing are selected.

The revision number is displayed in bold if a revision increment is pending because of modifications made to the drawing or the model since the last revision increment. Drawings obtain a pending revision increment if the drawing display on the worksheet changes or any of the drawing properties are altered. The project obtains a pending revision increment for any change made to the simulation model.

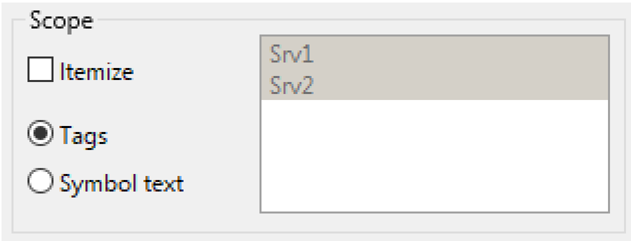
The revision numbering format and updating method are controlled by the revision control item in the project preferences. For manual revision control, the lowest level revision number may be incremented manually by the **Increment** command of the contextual menu of the list. This command is effective only if a revision increment is pending. The **Increment level** submenus of the contextual menu allow increments of higher-level revision numbers. This action resets all lower revision numbers and removes pending increments. In case of a drawing revision change, the model revision is adjusted to match the largest revision number of the drawings

2.4. Text

The **Text tab** on the Node and Link consoles presents the caption properties of nodes and link objects in the viewport.

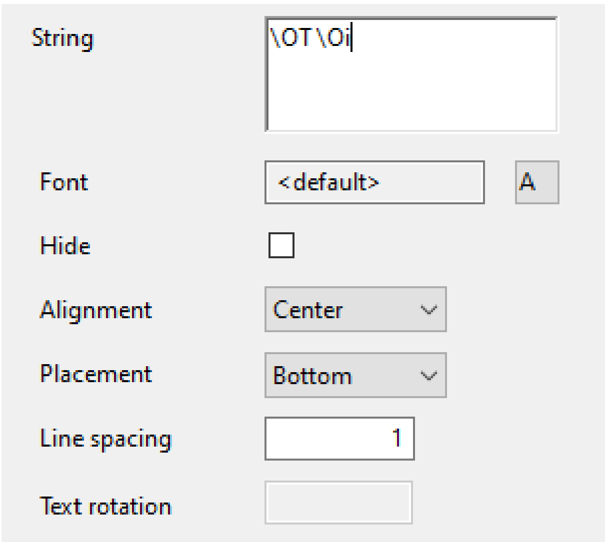
2.4.1. Scope

The scope frame determines which properties are referenced by the other controls on the tab. Selecting the **Symbol text** option (not available for links) changes the scope from node and link tags to the text displayed in symbol nodes.



Checking the **Itemize** box allows exclusion of individual objects by deselection in the list on the right, which presents a list of all node of link objects selected in the Viewport context of the configuration window.

The other controls on the Text tab manage the text string to be displayed and the display style.



String

Determines the text displayed. Multiple lines are allowed. Text markup can be used to insert object properties in placeholders. Symbol text also recognizes clock escape sequences. Any combination of escape sequences can be used. All characters that do not represent an escape sequence are displayed as literal text.

Font

A click on the Font text field opens a **Font selection dialog**, which allows defining the text font properties. Note that for symbol text the font size is specified as percentage of the extent to fit the text region in the symbol. Clicking the adjacent color selection button opens a dialog for selection of the text color. A click on the color button while pressing the **Ctrl-key** assigns the layer color to the text color. A click on the color button while pressing the **Shift-key** assigns the viewport foreground color to the text color.

Hide

Hides the text in the viewport

Alignment

Determines the horizontal text alignment. Possible values are *Left*, *Center* and *Right.* For tags only effective if the tag has multiple lines.

Placement

Determines the vertical text alignment. Possible values are *Top*, *Middle* and *Bottom.*

Line spacing

Determines the line spacing of the text display in the viewport, as fraction of the normal line spacing.

Text rotation

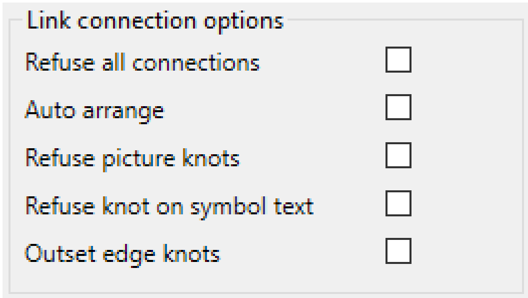
Determines the clockwise rotation angle of the link tag in degrees. An empty value of the text filed denotes alignment of the tag with the curve at the Tag seat location. Not available for nodes.

2.5. Network

The **Network tab** on the Node and Link consoles configures the connection properties of nodes and link objects in the viewport.

2.5.1. Link connection options

The link connection options frame presents several node options that configure the manner by which new links are connected to the node.



Refuse all connections

If checked, links cannot be connected to the node.

Auto arrange

Determines the method by which link connections are organized. If checked, new link connections are snapped to the node edges in a uniformly spaced arrangement. If unchecked new links can be connected to any location on the node. Changing the value of the Auto arrange property does not affect existing link connections

Refuse picture knots

If checked, link connections cannot be snapped to picture knots of the symbol icon.

Refuse knot on symbol text

If checked, link connections cannot be snapped to the outline of symbol text.

Outset edge knots

If checked, links connections are shifted away from the node edge by half the curve width, such that the connected link curves do not overlap with the node image.

2.5.2. Auto arrangement options

Determines the arrangement of link connections under application of the above-mentioned Auto arrange option. The control has three properties, organized in columns for the four sides of the node image as indicated in the first column. The **Max. knots** column determines the maximum number of permitted connections for each side. The value can be set by clicking the corresponding cell and typing the desired number. An empty field denotes that there is no limit to the number of connections. The check boxes in the **Static** column denote that the link connection spacing is predetermined in accordance with the specified Maximum knots value, independent of the number of connections. If checked, links can only be connected to a free knot position on the edge. The **Spacing** column determines the spread of the connections on the edge. A value between 0 and 1 can be specified, which denotes the relative length of connected section of the edge.

2.5.3. Connections

The connections frame contains a list of all connections of the selected node or link objects.

Connections

Upstream link	#	Server	#	Downstream link
Lnk2	1	Srv	1	Lnk
		Srv	2	Lnk1

The list has 5 columns. The center column shows the name of the selected object. The two outer columns display the names of the connected origin and destination nodes for links or the names of the connected links on both sides for nodes. The two columns on either side of the center column labeled with a hashtag display the sequence position of the connection. The names of selected objects are displayed in the system's selection highlight color in the two outer columns. For links, the destination and origin node can be changed by selecting another node name from the drop-down list that appears upon a mouse click on the popup-arrow at the edge of the list.

2.6. Operation

2.6.1. Hydraulic object properties

Operation type
Selects the operation type of the node or link. The available options include the most common unit operations, such as a piping *Junction*, a liquid *Reservoir,* a *Pump* and various valve types. The link value *Direct link* signifies a non-dimensional frictionless link between a tank or reservoir and a valve or pump.

Elevation
The elevation in feet (meters) above some common reference of the node. This is a required property without default value. Elevation is used only to compute pressure at the node. It does not affect any other computed quantity.

Length
Pipe length of the link

Feed flow rate
The flow rate of fluid entering the system at the junction (negative values indicate fluid removed from the system). The text area on the right-hand side specifies the name of the time pattern used to characterize time variation for this property. The adjacent button opens the Time pattern utility, which can be used to create, modify, and select time patterns. Only available for the standard project type. Not applied if the junction is a transient Mitigator.

Demand
The average base demand for fluid by the main category of consumer at the junction. A negative value is used to indicate an external source of flow into the junction. The text area on the right-hand side specifies the name of the time pattern used to characterize time variation in demand for the main category of consumer at the junction. The pattern provides multipliers that are applied to the base demand to determine actual demand in a given time period. The adjacent button opens the Time pattern utility, which can be used to create, modify, and select time patterns. Equivalent of the above-mentioned Flow rate property for the epanet project type.

Demand Categories
When the **ctrl**-enter key is pressed in the text field of the above-mentioned Flow rate property, the Demand category editor is displayed. This tool allows base demands and time patterns to be assigned to self-defined categories of consumers at the junction. Only available for the epanet project type.

Emitter rating
Discharge capacity of an emitter (sprinkler or nozzle) placed at a junction, expressed as a given flow rate at a reference pressure value. Set to 0 if no emitter is present. See Emitters for more details. Only available for the epanet project type.

Initial Level
The height in feet (meters) of the liquid surface above the bottom elevation of the tank at the start of the simulation. A time pattern can be assigned to the level property if the Elevation and Diameter property are zero and Scheduling is applied. In that case, the text area on the right-hand side specifies the name of the level time pattern. The adjacent button opens the Time pattern utility, which can be used to create, modify, and select time patterns.

Liquid level min. / max.
The minimum and maximum height in feet (meters) of the liquid surface above the bottom elevation of a tank. The tank level will not be allowed to drop below the minimum level and not to rise above the maximum level.

Top inlet headspace height
The height difference of the inlet nozzle level and the maximum liquid level in the tank. This property is applied to all inlet links connected to the tank. If the field is left empty (default), inlet links are connected to the bottom of the tank, at zero liquid level, as are all discharge links of the tank.

Volume curve or diameter
The (equivalent) diameter of a tank in feet (meters). The text field also accepts the name of a curve used to describe the relation between tank volume and liquid level. This property is useful for characterizing irregular-shaped tanks. The adjacent button opens the curve utility, which can be used to create, modify, and select volume curves.

Pump Curve or power
Numeric value of the power supplied by the pump, which assumes that the pump supplies the same amount of energy no matter what the flow is. The text field also accepts the name of a pump curve used to describe the relationship between the head delivered by the pump and the flow through the pump. The adjacent button opens the curve utility, which can be used to create, modify, and select pump curves.

Speed factor
The relative speed setting of the pump. For example, a speed setting of 1.2 implies that the rotational speed of the pump is 20% higher than the normal setting and a multiplier of zero implies that the pump will be shut off during the corresponding time period. The text area on the right-hand side specifies the name of the time pattern used to characterize time variation in pump speed. The adjacent button opens the Time pattern utility, which can be used to create, modify, and select time patterns. Leave blank if not applicable.

Initial Status
State of the pump or link (*Active* or *Closed*), or of the control valve or obstruction (*Active*, *Open*, *Closed*) at the start of the simulation period. For the values *Open* and *Closed* the setpoint of a control valve is ignored. The actual status can be altered during a simulation run using control statements. If a valve's initial status was *Open* or *Closed*, then it can be made active again using a control that assigns a value to the setpoint or loss coefficient. The link status value *Closed* is unavailable for links connected to a Check valve node or check valve links of the epanet project type.

Efficiency
The pump's wire-to-liquid efficiency (as fraction). The text field also accepts the name of an efficiency curve used to describe the relationship between pump efficiency and the flow rate. The adjacent button opens the curve utility, which can be used to create, modify, and select efficiency curves. The efficiency information is used only to compute energy consumption. Leave blank if the global Default pump efficiency is to be used.

Internal diameter
The internal diameter of the pipe, valve or obstruction. For pipes, clicking the adjacent button opens the Pipe size selector tool, which allows specification of the diameter value from a table of ANSI standard pipe sizes. A selected standard pipe size is indicated in the text field by a code containing schedule and nominal pipe size, rather than the actual diameter value. An empty text field leaves the diameter

undefined. For valve and obstruction nodes, the applied undefined diameter value is the smallest pipe diameter of all connected pipes. For an undefined diameter of a pipe link, PipeWeb seeks an appropriate value upstream of the link, which is indicated by a message if successful.

Object Type
The obstructor type: *Bidirectional resistor* or *Check valve* (non-return valve)

Valve Type
The valve type for pressure or flow rate control

Setpoint
The desired active setting of the control valve.

Minor loss coefficient
Unitless minor loss coefficient for the pipe. Clicking the adjacent button opens the pipe Minor loss evaluation tool, which allows calculation of the coefficient value for a range of valve types and other commonly used flow obstacles.

Wall roughness
The wall roughness coefficient of the pipe. It is unitless for Hazen-Williams or Chezy-Manning roughness and has units of millifeet (mm) for Darcy-Weisbach roughness.

Open state loss coefficient
Unitless minor loss coefficient that applies when the valve is completely opened. The contextual menu items **Calculate from Cv\Kv-value**, convert a number specified in the text field to the loss coefficient value, provided the internal diameter property of the valve can be resolved.

Head loss curve or coefficient
Unitless minor loss coefficient that applies when an obstruction is in the *Active* status. The text field also accepts the name of a head loss curve used to describe the relationship between head loss and the flow rate. The adjacent button opens the curve utility, which can be used to create, modify, and select head loss curves. The contextual menu items **Calculate from Cv\Kv-value**, convert a number specified in the text field to the loss coefficient value, provided the internal diameter property of the valve has been specified.

2.6.2. Tracing properties

Initial concentration
Species concentration at the node at the start of the simulation.

Source type
Selects the tracer source type for a storage or junction node where fluid enters the system. The options are:

Feed concentration	Fixes the concentration of any external inflow entering the network at a node.
Mass flow rate	Adds a fixed mass flow to that entering the node from other points in the network.
Concentration supplement	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).
Concentration increase	Adds a fixed concentration to that resulting from the mixing of all inflow to the node from other points in the network.

Source Quantity
Baseline or average concentration (or mass flow rate, depending on selection for the above-mentioned Source type property) of source. The text area on the right-hand side specifies the name of the time pattern used to characterize time variation in source quantity at the node. The pattern provides multipliers that are applied to the base value to determine actual quantity in a given time period. The adjacent button opens the Time pattern utility, which can be used to create, modify, and select time patterns.

Mixing Model
The type of liquid quality mixing that occurs within the tank. The choices include

Ideal	Fully mixed (MIXED),
Two-compartments	(2COMP)
Plug flow	First-in-first-out (FIFO)
Stagnant	Last-in-first-out (LIFO)

The text area on the right-hand side specifies the fraction of the tank's total volume that comprises the inlet-outlet compartment of the two-compartment (2COMP) mixing model. Not used for any other type of mixing models. See the Mixing Models topic for more information.

Bulk rate constant
The bulk reaction coefficient for chemical reactions in a tank or pipe. Use a positive value for growth reactions and a negative value for decay. Time units are 1/days. Leave blank if the Global bulk reaction constant is to be applied. See Liquid quality reactions for more information.

Wall rate constant
The wall reaction coefficient for the pipe. Use a positive value for growth and a negative value for decay. Time units are 1/day. Leave blank if the Global wall reaction constant is to be applied. See Liquid quality reactions for more information.

2.6.3. Transient properties

Action type
Selects the transient operation type options for non-Junction nodes (Pump, valve, obstruction). The available options are:

None	Normal operation, no specific transient actions.
Modulate	Changes the pump speed factor or head loss coefficient in accordance with a pre-determined profile.
Close \ Trip	Invokes a pump trip or completely closes a valve or obstruction at a controlled rate.

For the value *None* control valves keep their controlling function and pumps continue to run during the transient operation.

Start time
Start time of the transient operation

Transition duration
Duration time of the transient operation

Transition rate exponent
Specifies the constant m in the rate function $(1 - \tau^m)$ by which the modulated property varies in time, where τ is the dimensionless transient operation time (0 to 1). The transition is linear for $m = 1$, accelerating for $m > 1$, and decelerating for $m < 1$.

Final speed factor
Final value of the speed factor of a pump, at the end of the modulation operation.

Final loss coefficient

Final value of loss coefficient of a control valve or obstruction, at the end of the modulation operation.

Design rotational speed

Rotational speed at design conditions of the pump

Angular momentum

Angular momentum at steady state. Pressing the adjacent button displays a popup window that assists in estimation of the Angular momentum value.

Valve characteristic

Name of the characteristic curve for the obstruction or control valve. The valve characteristic curve defines the value of the object's flow coefficient Cv/Kv as a function of the open percentage status or stem travel of the instrument. The transient closing operation causes a decrease of the open percentage status from the steady-state value to zero at a rate determined by the other transition parameters applied. Press the adjacent button to define or select a characteristic curve for the valve. If no characteristic curve is specified, a linear decrease of the flow coefficient with the stem travel is applied. An error message is displayed if the steady-state flow coefficient value is outside of the characteristic curve range.

Elbow locations

Specifies a pattern for the elbow locations pattern for unbalanced pipe force evaluation on pipe sections divided by elbows. The text area specifies the name of the time pattern used to characterize the relative distance of elbows along the length of the pipe. The adjacent button opens a Pattern utility, which can be used to create, modify, and select elbow location patterns. If the text filed is left empty no elbows are assumed to be present and the pipe force on the entire pipe is evaluated.

Mitigator node properties

Mitigator type

Selects the mitigator type for junction nodes. The available options are:

None No transient mitigation actions.

Gas accumulator The node represents a vessel with a compressed gas chamber separated from the liquid content by a moving interface, such as a
piston, bladder or membrane.

Relief valve The node represents a relief valve that opens and closes at predetermined pressure setpoints.

Rupture disk The node represents a disk that that breaks at a predetermined pressure.

Standpipe The node represents a vertical pipe open to the ambient air at the top.

The applied value of the Feed flow rate property of a junction node depends on the selected value for this mitigator type property. For all transient mitigators (with selected value <> *None)* the property is ignored, and no feed flow is applied in the hydraulic solver. For transient junctions (value = *None*) the property is constant in the transient solver independent of the node pressure.

Gas accumulator properties

Tank diameter
Diameter of the cylindrical reservoir.

Tank height
Total height of both chambers of the cylindrical reservoir.

Polytropic index
A constant in the gas compression and expansion process of the accumulator. The default value 1.4 represent adiabatic conditions.

Precharge pressure
The initial pressure of the gas chamber in the accumulator before start-up of the hydraulic system.

Relief valve properties

Opening pressure
Pressure at which the surge relief valve opens.

Reseating pressure
Pressure at which the surge relief valve shuts. Cannot be greater than the above-mentioned Opening pressure. The difference between the Opening pressure and Reseating pressure is often referred to as blowdown. If unspecified, the relief valve pops open at the Opening pressure property, and never closes again after opening.

Valve flow coefficient Cv
The value of the flow coefficient Cv of the surge relief valve in fully open state. Only available if the *US Customary* option is selected for the Base unit system property.

Valve flow factor Kv
The value of the flow factor Kv of the surge relief valve in fully open state. Only available if the *Metric* option is selected for the Base unit system property.

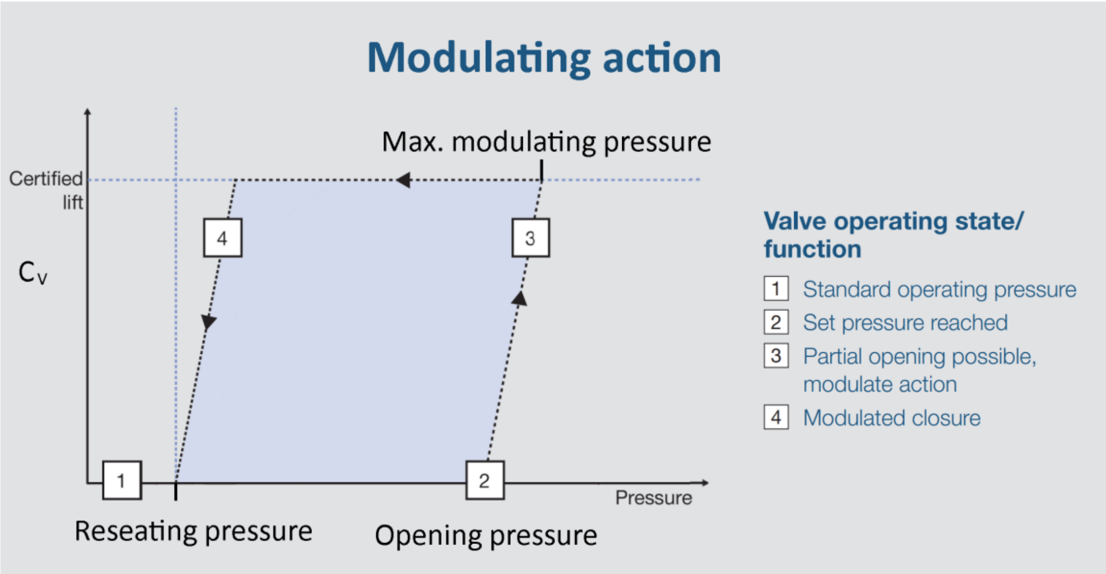
Back pressure
Specifies the (piezometric) backpressure of a surge relief valve or rupture disk mitigator, as constituent of the pressure differential driving force for the mitigating flow. It is assumed that reversed flow occurs if the node pressure of an open valve or rupture disk is lower than the back pressure. If unspecified for a relief valve, the valve will discharge into a connected downstream link. In that case the node must have a single upstream and a single downstream connected link to be functional. Otherwise the discharged liquid is removed from the system.

Relief valve mechanism properties

Valve type
Selects the operation method applied to the relief valve. The specific pressure control parameter that governs valve operation depends on the below-mentioned Balanced property of the valve. The Valve type value *Pop action* results into immediate opening of the valve when the pressure control parameter exceeds its' Opening pressure property. The valve remains completely opened until it closes when the control parameter becomes smaller than the Reseating pressure property of the valve. For the value *Modulating action,* the Kv- or Cv-value of the valve changes linearly with the control parameter until the control parameter reaches the Max. modulating pressure. Further details on relief operation mechanism specifics may be obtained from the white paper: *The Function, Advantages And Limitations Of Pilot-Operated Safety Valves*.

Balanced
The operation action of a balanced relief valve, such as a bellows-sealed spring valve, is independent of the downstream pressure. In that case the governing control parameter for valve operation is the upstream pressure at the valve. If the valve is not balanced, the control parameter is the difference between upstream pressure and back pressure.

Max. modulating pressure
The value of the pressure control parameter above which the valve is completely open. The modulating operation and the significance of the Maximum modulating pressure and other relief valve properties are depicted in the figure below.



This property is not available for the *Pop action* value for the above-mentioned Valve type property.

Min. transition time
The minimum duration of the opening and closing actions of the relief valve. Limits the change rate of the C_v- or K_v-property of the valve to 100% / Max transition time. Not available for the *Pop action pilot* value for the above-mentioned Valve type property.

Rupture disk properties

Burst pressure
Pressure at which the rupture disk breaks.

Discharge coefficient Cd
Value of the discharge coefficient of the aperture of the rupture disk

Aperture diameter
Diameter of the aperture of the rupture disk.

Stand pipe properties

Pipe diameter
Diameter of the standpipe.

Pipe length

Length of the standpipe. Determines the maximum liquid level in the pipe, resulting in overflow of the standpipe if liquid continues to enter the pipe.

Check valve properties

Closing velocity

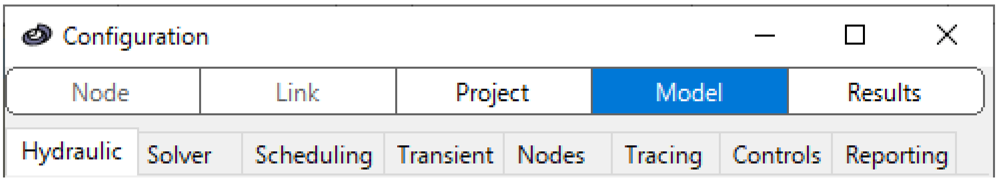
Flow velocity at which the check valve closes. Enter a negative number for reversed flow. Note that a check valve is implemented within the transient operation of any pump node, with the default value *0* for the *Closing velocity* property as well as for the below-mentioned **Required opening head** property.

Required opening head

Head differential across the valve required to re-open a closed check valve.

3. Model configuration

The configuration window provides access to general properties of the model and many other elements of the project. The window is opened by the **Configuration** menu on the Tools menu or by double-clicking the viewport.



The category bar at the top of window selects the main category for which properties are displayed. Selecting a category displays the corresponding console with all relevant control items.

The **Model** console contains general properties of the model that are not associated with objects represented on the viewport.

This chapter discusses the model property configuration, organized by the available tabs on the topics tab strip beneath the category bar:

- Hydraulic
- Solver
- Scheduling
- Transient
- Nodes
- Tracing / Quality
- Controls
- Reporting

3.1. Hydraulic

Head loss equation
Formula used to compute head loss as a function of flow rate in a pipe. Choices are:
- Hazen-Williams
- Darcy-Weisbach
- Chezy-Manning

Because each formula measures pipe roughness differently, switching formulas might require that all pipe roughness coefficients be updated.

Base unit system
Preselect the default units in which properties and results are expressed.

Unit flow rate
Selects the units for expression of flow rate

Unit pressure
Selects the units for expression of pressure (gauge).

Unit pressure differential
Selects the units for expression of pressure differentials

Density
Liquid density

Viscosity
Kinematic viscosity of the liquid

Default pump efficiency
Global default pump efficiency value

3.2. Solver

Maximum Trials
Maximum number of trials used to solve the nonlinear equations that govern network hydraulics at a given point in time. Suggested value is 40.

Accuracy
Convergence criterion used to signal that a solution has been found to the nonlinear equations that govern network hydraulics. Trials end when the sum of all flow changes divided by the sum of all link flows is less than this number. Suggested value is 0.001.

Maximum Head Error
Augments the ACCURACY option. Specifies the maximum head loss error any network link can have for hydraulic convergence to occur. The default value of 0 indicates that no head error limit applies. Units of this parameter are feet (US) or meters (SI).

Maximum Flow Change
Augments the ACCURACY option. Specifies the largest change in flow that any network element (link, emitter, or pressure- dependent demand) can have for hydraulic convergence to occur. The default value of 0 indicates that no flow change limit applies. It is specified based on the current project flow unit setting.

CHECKFREQ
This sets the number of solution trials that pass during hydraulic balancing before the status of pumps, check valves, flow control valves and pipes connected to tanks are once again updated. The default value is 2, meaning that status checks are made every other trial. A value equal to the maximum number of trials would mean that status checks are made only after a system has converged. (Whenever a status change occurs the trials must continue since the current solution may not be balanced.) The frequency of status checks on pressure reducing and pressure sustaining valves (PRVs and PSVs) is determined by the DAMPLIMIT option (see below).

MAXCHECK
This is the number of solution trials after which periodic status checks on pumps, check valves, flow control valves and pipes connected to tanks are discontinued. Instead, a status check is made only after convergence is achieved. The default value is 10, meaning that after 10 trials, instead of checking status every CHECKFREQ trials, status is checked only at convergence.

DAMPLIMIT
This is the accuracy value at which solution damping and status checks on PRVs and PSVs should begin. Damping limits all flow changes to 60% of what they would otherwise be as future trials unfold. The default is 0 which indicates that no damping should be used and that status checks on control valves are made at every iteration. Damping might be needed on networks that have trouble converging, in which case a limit of 0.01 is suggested.

Continue if unbalanced
Action to take if a hydraulic solution is not found within the maximum number of trials. Choices are STOP to stop the simulation at this point or CONTINUE to use another 10 trials, with no link status changes allowed, in an attempt to achieve convergence.

Below are some typical values that might be used for the status checking parameters:

TYPICAL VALUES FOR STATUS CHECKING PARAMETERS			
CHECKFREQ	***MAXCHECK***	***DAMPLIMIT***	***REMARKS***
2	10	0	Frequent status checking; tends to produce solutions in the least number of iterations.
10	100	0.01	Less frequent status checking; might be needed for networks that have difficult in converging.
Max. Trials	Max. Trials	Convergence Accuracy	Status checks made only after convergence is achieved; might produce convergence when other settings fail.

3.3. Scheduling

Simulation duration time
Total length of a simulation in hours. Use 0 to run a single period (snapshot) hydraulic analysis.

Hydraulic Time Step
Time interval between re-computation of system hydraulics. Normal default is 1 hour.

Tracing Time Step
Time interval between routing of liquid tracing/quality constituent. Normal default is 5 minutes (0:05 hours).

Pattern Time Step
Time interval used with all time patterns. Normal default is 1 hour.

Pattern Start Time
Hours into all time patterns at which the simulation begins (e.g., a value of 2 means that the simulation begins with all time patterns starting at their second hour). Normal default is 0.

Reporting Time Step
Time interval between times at which computed results are reported. Normal default is 1 hour.

Report Start Time
Hours into simulation at which computed results begin to be reported. Normal default is 0.

Clock start time
Clock time (e.g., 7:30 am, 10:00 pm) at which simulation begins. Default is 12:00 am (midnight).

3.4. Transient

Transient simulation time
Specifies the transient simulation time span. Leave empty to disable transient analysis.

Liquid bulk modulus
Specifies the compressibility of the liquid. The default value of 2.15 represents the value for water at room temperature.

Piping Young's modulus
Specifies the elasticity of the liquid. The default value of 200 is representative for carbon steel at room temperature.

Pipe friction model
Selects the friction model applied to transient pipe flow. The available options are:

Adopt from steady state	Uses the friction factor value determined for each pipe by the steady state hydraulics solver as a constant value.
Darcy-Weisbach	Evaluates pipe friction losses for each pipe as a function of transient velocity in accordance with the Darcy-Weisbach friction model.
Unsteady Vardy-Brown	Evaluates pipe friction losses for each pipe as a function of transient velocity using a well-established convolution-based friction model. This option is the most demanding for CPU resources.

Cavitation model
Liquid may evaporate at some point in a piping system if the pressure becomes smaller than the saturated vapor pressure of the liquid. Initially, the vapor created will form small bubbles or cavities in the liquid. RND PipeWeb allows accounting for the effect of such evaporation phenomena by using the Discrete vapor cavity model (DVCM) to calculate the volume fraction of vapor in a pipe. The DVCM is generally considered to be accurate up to vapor volume fractions of 10%.

Saturated vapor pressure
Specifies the saturated vapor pressure of the liquid as reference for the above-mentioned cavitation model.

Pipe segmentation aim
Specifies the desired mean of the number of discrete segments per pipe for all pipes in the transient solver. The accuracy of the solution increases with the pipe segment count at the cost of higher CPU load.

Discretization characteristics
The Discretization characteristics frame provides an overview of the discretization properties of the model. A number of key parameter values are displayed together in a concise list. The table is updated automatically after each simulation run. Pressing the update button refreshes the table to the values applicable to the next simulation run for the present model configuration and value of the above-mentioned property: Pipe segmentation aim. The discretization properties displayed in the list are:

Time step	Simulation time step in seconds
Time step count	Total number of time steps in the simulation.
Min. pipe segment count	Minimum number of segments in a pipe.

Max. pipe segment count	Maximum number of segments in a pipe. Note that values above 25 do normally not offer significant additional accuracy.
Min. oscillation count	Minimum number of wave oscillations for all pipes in the system expressed as: (*simulation time)* / (4 * *pipe wave travel time*).
Max. wave speed adjustment	The solution method applied requires some adjustment of the pipe wave speeds in order to satisfy the condition of equal wave travel time for all pipe segments in the model. The reported value indicates the maximum % adjustment applied to all pipes in the model. A value of 0% is the best attainable result.
CPU load	An indicator to the computational effort required for the simulation, expressed as: (*total pipe segments*) * (*total time steps*). Excludes all calculations performed on nodes.

3.5. Nodes

Junction flow rate multiplier

Global multiplier applied to all demands to make total system consumption vary up or down by a fixed amount (e.g., 2.0 doubles all demands, 0.5 halves them, and 1.0 leaves them as is).

Spray emitter exponent

Power to which pressure is raised when computing the flow through an emitter device. The textbook value for nozzles and sprinklers is 0.5. This may not apply to pipe leakage. Consult the discussion of Emitters for more details. Only available for the epanet project type.

Low pressure flow curtailing

Selects between demand or pressure driven analysis – DDA or PDA, respectively. DDA assumes demands are fixed at a given point in time, while PDA assumes demands depend on pressure at low pressure levels. The PDA option can be used to find a solution when negative pressures are present in a DDA. See Pressure Driven Demand for more information.

Minimum curtail pressure

In a PDA, as selected by the above-mentioned option, the pressure below which demand is assumed to be zero.

Required curtail pressure

In a PDA, the pressure required to deliver the full demand.

Curtail transition exponent

PDA assumes a pressure demand relation raised to an exponent. Standard value is 0.5.

Transient compliant network

Select *Yes* to disable junction properties that are not compatible with the transient solver.

3.6. Tracing / Quality

Tracing model

Type of liquid tracing/quality parameter to be modeled. Only available if scheduling is applied for the hydraulic solver. The options are:

- None (no tracing),
- Reacting species (compute chemical concentrations)
- Residence time (compute liquid age)
- Source tracking (trace flow from a specific node)

Unit concentration

Concentration units used to express concentration. Choices are mg/L or ug/L. Units for Age and Trace analyses are fixed at hours and percent, respectively

Tracing tolerance

Smallest change in tracing/quality that will cause a new parcel of liquid to be created in a pipe. A typical setting might be 0.01 (for species in units mg/L). This tolerance parameter determines when the tracing parameter of one parcel of fluid is essentially the same as another parcel.

Source node name

Name of the node whose flow is being traced. Applies only to source tracing

Pipe wall reaction order

Power to which concentration is raised when computing a bulk flow reaction rate. Choices are FIRST (1) for first-order reactions or ZERO (0) for constant rate reactions. If no global or pipe-specific wall reaction coefficients are assigned then this option is ignored.

Pipe bulk reaction order

Power to which concentration is raised when computing a bulk flow reaction rate. Use 1 for first-order reactions, 2 for second-order reactions, etc. Use any negative number for Michaelis-Menton kinetics. If no global or pipe-specific bulk reaction coefficients are assigned then this option is ignored.

Tank bulk reaction order

Power to which concentration is raised when computing a bulk flow reaction rate. Use 1 for first-order reactions, 2 for second-order reactions, etc. Use any negative number for Michaelis-Menton kinetics. If no global or pipe-specific bulk reaction coefficients are assigned then this option is ignored.

Global wall rate constant

Wall reaction rate coefficient K_w assigned to all pipes. Can be overridden by editing this property for specific pipes. Use a positive number for growth, a negative number for decay, or 0 if no wall reaction occurs. Units are ft/day (US) or m/day (SI) for first-order reactions and mass/sq ft/day (US) or mass/sq m/day (SI) for zero- order reactions.

Global bulk rate constant

Default bulk reaction rate coefficient K_b assigned to all pipes. This global coefficient can be overridden by editing this property for specific pipes. Use a positive number for growth, a negative number for decay, or 0 if no bulk reaction occurs. Units are concentration raised to the (1 - n) power divided by days, where n is the bulk reaction order.

Diffusivity

Molecular diffusivity of the chemical being modeled. Only used when modeling mass transfer for pipe wall reactions. Set to zero to ignore mass transfer effects.

Concentration limit

Maximum concentration that a substance can grow to or minimum value it can decay to. Bulk reaction rates will be proportional to the difference between the current concentration and this value. See discussion of Bulk Reactions in section Liquid Quality Simulation Model for more details. Set to zero if not applicable.

3.7. Controls

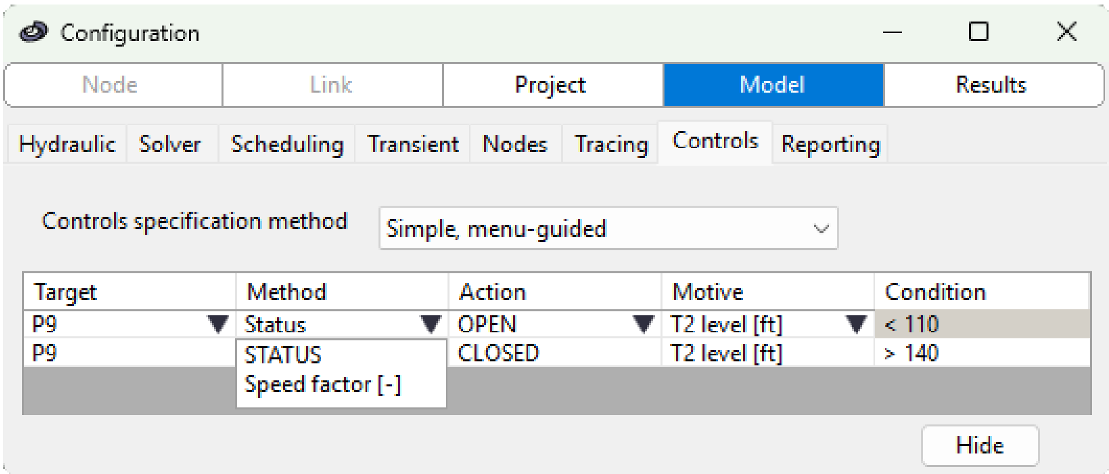
Control of the hydraulic system is configured in the controls tab. The section Control system provides further details on how to use define in a model.

Controls specification method
Selects the method for definition of controls in the hydraulic model. The options are:

- **Menu-guided** Simple controls depending on a single condition in the network
- **Scripted** Rule-based controls depending on multiple simultaneously occurring conditions.

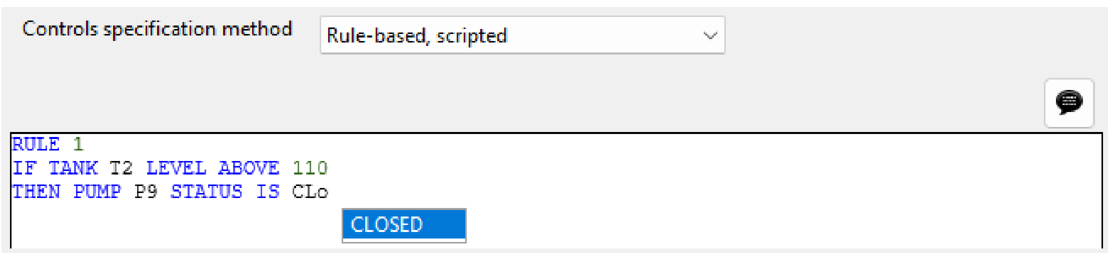
3.7.1. Menu guided controls

Simple controls can be added or removed using the contextual menu which appears upon a right mouse button click on the control items list. The content of the last column can be edited after clicking on the corresponding cell. The other columns have a drop-down menu which permits selection of the applied value.



3.7.2. Rule based controls

Rule-based controls can be specified by the rule-based control composer, which is a basic text editor, with some additional features that assist coding. Note that a rule definition must be start with the keyword RULE followed by a sequence number.



Automatic code completion

This feature automatically fills out method names and object names while editing scripts. A list of selectable phrases appears in a popup list on the text area when the **Tab key** or **Ctrl+Space key** is pressed. The list contains method or object names which represent an appropriate continuation of the text preceding the caret position on the text area. The desired item in the list can be selected with the navigation keys on the keyboard. Pressing the **Tab key, spacebar** or the **Enter key** will complete the text at the caret position with the selected item in the pop-up list. Pressing the **Esc key** closes the list without making any changes to the script code.

Comment button

The comment button on the top right of the frame converts selected code lines into comments by inserting a hyphen character at the beginning of the line. If all selected lines start with a hyphen character the hyphens are removed in order to convert the comments back into script code.

Find and replace

A set of text search controls appears above the text area upon pressing the **Ctrl-F shortcut key** combination. A click on the **Find button,** which is located at the right of the text field, will highlight all matching occurrences in the script. Additional **Ctrl F** keystrokes will loop through the search matches by selecting the search matches. Holding down the **Shift key** as well reverses the loop direction. The items in the **Search options** contextual menu offer several options for the search method applied. The **Close button** to the left of the search text box clears the search and hides the text search controls.

The command **Replace** in the contextual menu of the Search text field or pressing the **Ctrl R** shortcut key combination exposes an additional text field and push button underneath the Search text field. The push button performs a replacement of the selected search text match by the text specified in the Replace text field. The command **Replace all** in the contextual menu of the Replace text field replaces all search matches by the replacement text.

3.8. Reporting

Time display format

Selects the format for time display. The options include:

HHH	number of completed hours since simulation start
ddd	number of completed days since simulation start
HH:MM:SS	24-hour clock time display
.#	fractional hour or day time

Contour map scale min. / max.

Specifies the extent of the color contour map display scale range clipping.

Contour map log scale

Check to activate the log scale option for color contour map display.

Contour map clip to range

Check to hide color contours with values outside the above-mentioned scale min. / max. extent.

Contour legend annotation

Check to activate indication of the color contour map legend scale range values.

Contour legend location

Selects the location of the contour map color legend.

Contour legend size

Selects the size of the contour map color legend.

Prohibit chart data compression

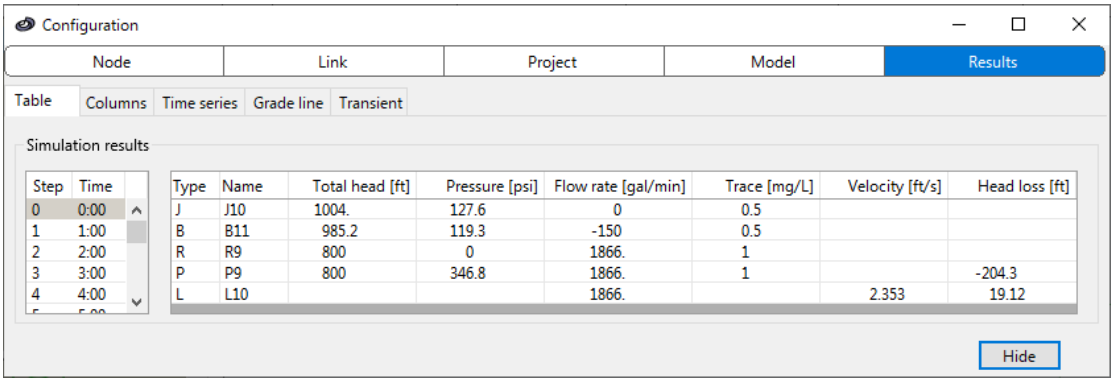
Check to prevent chart data compression for smoother chart curves on the results tab for transient simulations with high wave oscillation counts or scheduled simulations with high duration times.

4. Results

The configuration window provides access to simulation results and many other elements of the project. The window is opened by the **Configuration** menu on the Tools menu or by double-clicking the viewport.

The category bar at the top of window selects the main category for which properties are displayed. Selecting a category displays the corresponding console with all relevant control items.

The **Results** console provides access to results of both hydraulic simulation and transient analysis.



The tabs on the **Presentation tab strip** located below the category bar at the top of window select the configuration of the data presentation as follows:

- Table — Displays hydraulic simulation node or link results at a specified point in time in a table.
- Columns — Displays hydraulic simulation node or link results at a specified point in time in a column diagram.
- Time series — Displays hydraulic simulation node or link results as a function of time in a chart.
- Grade line — Displays hydraulic head results along a flow trajectory at a specified point in time in a chart.
- Transient data — Displays transient analysis results either as a function of time or along a flow trajectory in a chart.

4.1. Results display control

Both the results table and the various chart elements may be configured using the contextual menu, which appears upon a right mouse click on the chart or table. The available commands are:

Column display

Selects the below-mentioned Hydraulic parameters for display in a column in the results Table.

Place chart on the viewport

Places the chart as configured on the Viewport. Charts placed in the viewport are updated when the results are updated by a new solver run. In addition, they can be resized and styled using Chart Configuration.

Reset Y-scale range

Rematches the Y-scale range to the span of the y-values data set. This command can be helpful after changing the schedule time selection or the Transient parameter control bar value for a chart. For these actions the scale range is not automatically reduced if the Y-data range is contracting, but this adjustment can still be imposed by this command.

Show curves for range min/max values

Forces the display of separate curves for minimum and maximum values in time for Transient data charts. Only effective along a continuous piping trajectory and if a single transient data parameter is selected. The min/max value curves are plotted in the same formatting as the transient parameter curve, but with a smaller line thickness.

Legend adjacent

Toggles between legend placement inside the chart plot area or adjacent to the chart at the top right.

Hydraulic parameters

The hydraulic solver results can be displayed for the following parameters:

- **Total head** Total head at a node [m or ft]
- **Pressure** Stagnation pressure (gauge) at a node [Selected pressure unit]
- **Flow rate** Flow rate [Selected flow rate unit]. (epanet project type: < 0 if entering the system; Standard project type: < 0 if leaving system).
- **Curtailing** Percent curtailing applied at a junction node [%]
- **Tank level** Liquid level in a tank node [m or ft]
- **Trace** Depends on the selection for Tracing model. Concentration of tracer compound [mg/L or µg/L] or fluid residence time [hrs], or % fluid from source.
- **Elevation** Elevation of a node with respect to a fixed model datum [m or ft]
- **Pipe length** Length of a pipe link [m or ft]
- **Velocity** Flow velocity in a pipe, valve or obstruction [m/sec or ft/sec]
- **Pressure loss** Pressure differential across a pump, pipe, valve or obstruction [Selected pressure differential unit]
- **Dissipation** Energy dissipation in a pump, pipe, valve or obstruction [kW]

- **Loss coeff.** Resistance coefficient 2·g·ΔH / (v^2) of a pipe, valve or obstruction [-]
- **Pipe diam.** Pipe diameter [m or ft]
- **Reynolds** Pipe Reynolds number [-]
- **Spec. dissip.** Pipe energy dissipation per unit length [kW/km or kW/mile]

Schedule times list

If scheduling is applied for the hydraulic solver, the schedule time point for hydraulic data display can be selected in the Scheduling times list. This list is only available for Columns and Grade line charts and on the Table tab, if more than one link and node objects are selected in the viewport. It is located at the left side of the **Simulation results** frame.

4.1. Results display elements

Depending on the selected tab on the **Presentation tab strip** located below the category bar at the top of window the data displayed on the chart area is one of the items given below. The chart properties may be adjusted using contextual menu commands as discussed in the Results display control section. If scheduling is applied for the hydraulic solver, data displayed for some chart types also depends on the selected row in the Scheduling times list.

Table

The Table tab presents a table of hydraulic simulation results of all objects selected for various hydraulic parameters on the datasheet at a selected point in time. The Parameter columns displayed can be determined by the contextual menu which appears upon a right-click on the table with the mouse.

Columns

The Columns tab presents a column chart of hydraulic simulation results of all objects selected on the viewport for one or more hydraulic parameters. The displayed parameters can be selected from the set of checkboxes above the chart. The time point for data display can be selected in the times list located to the left of the columns chart. A Column results chart can be placed on the Viewport using the right-button contextual menu of the chart.

Time series

The Time series tab presents a chart of hydraulic simulation results of all objects selected on the viewport for one or more hydraulic parameters as a function of time. The displayed parameters can be selected from the set of checkboxes above the chart. A Time series results chart can be placed on the Viewport using the right-button contextual menu of the chart.

Grade line

The Grade line tab presents a line chart of hydraulic simulation results of a chain of links selected on the viewport forming a continuous flow trajectory, for one or more hydraulic head parameters. The set of checkboxes above the chart allows selection of the plotted parameters. The hydraulic head parameter values [m or ft] that can be displayed are:

- **Total head** — Total head: Sum of elevation and static pressure head and velocity head
- **Elevation** — Elevation head
- **Static head** — Static pressure head
- **Velocity head** — Velocity head: $\rho \cdot v^2/(2 \cdot g)$
- **Stagnation head** — Sum of static pressure head and velocity head
- **Hydraulic head** — Sum of elevation and static pressure head

The elevation parameter varies linearly with pipe length from the source node elevation to the target node elevation. The outflow of liquid into a tank or reservoir results into complete dissipation of the velocity head and of any elevation head of the tank inlet nozzle above the liquid level (see headspace height). The corresponding head loss is indicated by a vertical segment of the total head grade line at the node location.

The time points list located to the left of the columns chart selects the time for data display. A grade line chart can be placed on the Viewport using the right-button contextual menu of the chart. No data is displayed if the selected links chain does not form a continuous flow path.

Transient data

The Transient data tab presents a line chart of hydraulic simulation results of all objects selected on the viewport, for one or more transient parameters. The displayed parameters can be selected from the set of checkboxes above the chart. The transient analysis parameter values that can be displayed are:

- **Pressure** Static pressure at a pipe location [Selected pressure unit]
- **Velocity** Flow velocity at a pipe location [m/sec or ft/sec]
- **Vapor fraction** Vapor mass fraction at a pipe location [-]
- **Pipe force** Unbalanced force exerted on a pipe or pipe segments divided by elbows [kN]
- **Node flow rate** Node flow rate as a function of time [Selected flow rate unit]. For reservoir and junction nodes this parameter represents the flow leaving or entering the system either in steady state or as mitigator action. For other node types it represents the flow rate through the node.
- **Liquid level** Liquid level of a junction node of mitigator type *Gas accumulator* or *Standpipe* as a function of time [m or ft]
- **Modulation** Modulation action as a function of time [Dimensionless]. The parameter is either the modulated valve loss coefficient, or ratio of the actual flow coefficient Kv/Cv to the maximum value of a modulated closing valve or a modulating relief valve, or the actual speed factor of a tripping or modulated pump.

A set of controls, depicted below, located underneath the chart, govern the transient data display properties. A chart type switch button is located at the left. The large parameter control bar in the middle can be adjusted by mouse dragging or by the small step buttons on either side of the bar. The switch button toggles between time and location charts. A time chart plots a transient results variable as a function of time, with the location being determined by the parameter control bar. Alternatively, a location chart plots a transient variable as a function of pipe location, with time fixed by the parameter control bar. The latter option indicates the node locations in the parameter control bar and is available only if the selected links chain forms a continuous flow path. The active parameter plot option is indicated by the color of the parameter control bar and the chart x-axis title.

Holding down the left mouse button on one of the two arrow buttons for a short time activates an animation with continuously proceeding chart parameter value. The animation is stopped by clicking on any of the transient result controls.

A Transient data results chart can be placed on the Viewport using the right-button contextual menu of the chart. This menu also allows to enable display of min/max value curves together with a curve for a parameter along a continuous piping path

5. Hydraulic and transient solvers

This section focuses on the architecture of the hydraulic solver and the transient solver. The text of sections 5.1 through 5.5 has been adopted with minor changes from https://github.com/USEPA/epanet-manual.

5.1. EPANET Physical Components

EPANET models a liquid distribution system as a collection of links connected to nodes. The links represent pipes, pumps, and control valves. The nodes represent junctions, tanks, and reservoirs. Fig. 5.1 below illustrates how these objects can be connected to one another to form a network.

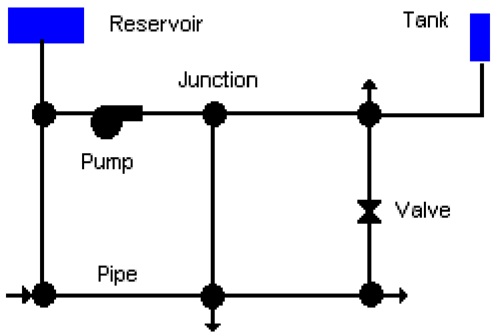


Fig. 5.1 Physical components in a liquid distribution system.

5.1.1. Junctions

Junctions are points in the network where links join together and where liquid enters or leaves the network. The basic input data required for junctions are:

- Elevation above some reference (usually mean sea level)
- Liquid demand (rate of withdrawal from the network)
- Initial liquid 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 (gauge)
- Liquid quality

Junctions can also:

- Have their demand vary with time
- Have multiple categories of demands assigned to them
- Have negative demands indicating that liquid is entering the network
- Have pressure driven demand
- Be liquid quality sources where constituents enter the network
- Contain emitters (or sprinklers) which make the outflow rate depend on the pressure

5.1.2. Reservoirs

Reservoirs are nodes that represent an infinite external source or sink of liquid to the network. They are used to model such things as lakes, rivers, groundwater aquifers, and tie-ins to other systems. Reservoirs can also serve as liquid quality source points.

The primary input properties for a reservoir are its hydraulic head (equal to the liquid surface elevation if the reservoir is not under pressure) and its initial quality for liquid quality analysis.

Because a reservoir is a boundary point to a network, its head and liquid quality cannot be affected by what happens within the network. Therefore it has no computed output properties. However its head can be made to vary with time by assigning a time pattern to it (see Time Patterns below).

5.1.3. Tanks

Tanks are nodes with storage capacity, where the volume of stored liquid can vary with time during a simulation. The primary input properties for tanks are:

- Bottom elevation (where liquid level is zero)
- Diameter (or shape if non-cylindrical)
- Initial, minimum and maximum liquid levels
- Initial liquid quality

The principal outputs computed over time are:

- Hydraulic head (liquid surface elevation)
- Liquid quality

Tanks are required to operate within their minimum and maximum levels. EPANET 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 liquid quality source points.

5.1.4. 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^γ
```

where q = flow rate, p = gauge pressure, C = discharge coefficient, and γ = pressure exponent. For nozzles and sprinkler heads γ 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 are 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) or compute a fire flow at the junction (the flow available at some minimum residual pressure). In the latter case one would use a very high value of the discharge coefficient (e.g., 100 times the maximum flow expected) and modify the junction's elevation to include the equivalent head of the pressure target. EPANET treats emitters as a property of a junction and not as a separate network component.

Note: The pressure-flow relation at a junction defined by an emitter should not be confused with the pressure-demand relation when performing a pressure driven analysis (PDA). See Pressure Driven Demand for more information.

5.1.5. Pipes

Pipes are links that convey liquid from one point in the network to another. EPANET assumes that all pipes are full at all times. Flow direction is from the end at higher hydraulic head (internal energy per weight of liquid) to that at lower head. The principal hydraulic input parameters for pipes are:

- Start and end nodes
- Diameter
- Length
- Roughness coefficient (for determining headloss)
- Status (open, closed, or contains a check valve)

The status parameter allows pipes to implicitly contain shutoff (gate) valves and check (non-return) valves (which allow flow in only one direction).
The liquid quality inputs for pipes consist of:

- Bulk reaction coefficient
- Wall reaction coefficient

These coefficients are explained more thoroughly in section Liquid Quality Simulation Model.

Computed outputs for pipes include:

- Flow rate
- Velocity
- Headloss
- Darcy-Weisbach friction factor
- Average reaction rate (over the pipe length)
- Average liquid quality (over the pipe length)

The hydraulic head lost by liquid flowing in a pipe due to friction with the pipe walls can be computed using one of three different formulas:

- Hazen-Williams formula
- Darcy-Weisbach formula
- Chezy-Manning formula

The Hazen-Williams formula is the most commonly used headloss formula in the US. It cannot be used for liquids other than liquid and was originally developed for turbulent flow only. The Darcy-Weisbach formula is the most theoretically correct. It 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:

$$h_L = A\, q^B$$

where h_L = headloss (Unit: Length), q = flow rate (Unit: Volume/Time), A = resistance coefficient, and B = flow exponent. Table 5.1 lists expressions for the resistance coefficient and values for the flow exponent for each of the formulas. Each formula uses a different pipe roughness coefficient that must be determined empirically. Table 5.2 lists general ranges of these coefficients for different types of new pipe materials. Be aware that a pipe’s roughness coefficient can change considerably with age.

With the Darcy-Weisbach formula EPANET 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).
- The Swamee and Jain approximation to the Colebrook-White equation is used for fully turbulent flow (Re > 4,000).
- A cubic interpolation from the Moody Diagram is used for transitional flow (2,000 < Re < 4,000).

Consult the text in Analysis Algorithms for the actual equations used.

TABLE 5.1 PIPE HEADLOSS FORMULAS FOR FULL FLOW (FOR HEADLOSS IN FEET AND FLOW RATE IN CFS)		
Formula	***Resistance Coefficient* (A)**	***Flow Exponent* (B)**
Hazen-Williams	$4.727\ C^{-1.852}\ d^{-4.871}\ L$	1.852
Darcy-Weisbach	$0.0252\ f(\varepsilon,d,q)\ d^{-5}\ L$	2
Chezy-Manning	$4.66\ n^{2}\ d^{-5.33}\ L$	2

Notes:

C = Hazen-Williams roughness coefficient
ε = Darcy-Weisbach roughness coefficient (ft)
f = friction factor (dependent on ε, d and q)
n = Manning roughness coefficient
d = pipe diameter (ft)
L = pipe length (ft)
q = flow rate (cfs)

Table 5.2 Roughness Coefficients for New Pipe			
Material	***Hazen-Williams* C *(unitless)***	***Darcy-Weisbach* ε (ft x 10^{-3})**	***Manning's* n *(unitless)***
Cast Iron	130 – 140	0.85	0.012 – 0.015
Concrete or Concrete Lined	120 – 140	1.0 – 10	0.012 – 0.017
Galvanized Iron	120	0.5	0.015 – 0.017
Plastic	140 – 150	0.005	0.011 – 0.015
Steel	140 – 150	0.15	0.015 – 0.017
Vitrified Clay	110		0.013 – 0.015

Pipes can be set open or closed at preset times or when specific conditions exist, such as when tank levels fall below or above certain set points, or when nodal pressures fall below or above certain values. See the discussion of Controls in Section 3.2.

Minor Losses
Minor head losses (also called local losses) are caused by the added turbulence that occurs at bends and fittings. The importance of including such losses depends on the layout of the network and the degree of accuracy required. They can be accounted

for by assigning the pipe a minor loss coefficient. The minor headloss becomes the product of this coefficient and the velocity head of the pipe, i.e.,

$$h_L = K \cdot v^2 / (2 \cdot g)$$

where K = minor loss coefficient, v = flow velocity (Unit: Length/Time), and g = acceleration of gravity (Unit: Length/Time^2)

Pumps are links that impart energy to a fluid thereby raising its hydraulic head. The principal input parameters for a pump are its start and end nodes and its pump curve (the combination of heads and flows that the pump can produce). In lieu of a pump curve, the pump could be represented as a constant energy device, one that supplies a constant amount of energy (horsepower or kilowatts) to the fluid for all combinations of flow and head.

The principal output parameters are flow and head gain. Flow through a pump is unidirectional and EPANET will not allow a pump to operate outside the range of its pump curve.

Variable speed pumps can also 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 (see the section on Pump Curves below).

As with pipes, pumps can be turned on and off at preset times or when certain conditions exist in the network. A pump’s operation can also be described by assigning it a time pattern of relative speed settings. Each pump can be assigned an efficiency curve. If these a global pump efficiency value will be used.

Flow through a pump is unidirectional. If system conditions require more head than the pump can produce, EPANET shuts the pump off. If more than the maximum flow is required, EPANET extrapolates the pump curve to the required flow, even if this produces a negative head. In both cases a warning message will be issued.

5.1.6. Valves

Valves are links that limit the pressure or flow at a specific point in the network. Their principal input parameters include:

- Start and end nodes
- Diameter
- Setting
- Status

The computed outputs for a valve are flow rate and headloss. The different types of control valves included in EPANET are:

- Pressure Reducing Valve (PRV)
- Pressure Sustaining Valve (PSV)
- Pressure Breaker Valve (PBV)
- Flow Control Valve (FCV)

The valves types included in EPANET that correspond with the PipeWeb object type "Restrictor" are:

- Throttle Control Valve (TCV)
- General Purpose Valve (GPV)

PRVs limit the pressure at a point in the pipe network. EPANET 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)

PSVs maintain a set pressure at a specific point in the pipe network. EPANET 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)

PBVs force a specified pressure loss to occur across the valve. Flow through the valve can be in either direction. PBV's are not true physical devices but can be used to model situations where a particular pressure drop is known to exist. Only available for the epanet project type.

FCVs limit the flow to a specified amount. The program produces a warning message if this flow 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).
Restrictors simulate a partially closed valve by adjusting the minor head loss coefficient of the valve. A relationship between the degree to which a valve is closed and the resulting head loss coefficient is usually available from the valve manufacturer. The user may supply a special flow - head loss relationship curve instead of a fixed minor loss coefficient. These curves can be used to model turbines, well drawdown or reduced-flow backflow prevention valves.

Shutoff valves and check (non-return) valves, which completely open or close pipes, are not considered as separate valve links but are instead included as a property of the pipe in which they are placed. As an exception, for the RND PipeWeb standard project type, check valves are specified as Restrictor option.

Each type of valve has a different type of setting parameter that describes its operating point (pressure for PRVs, PSVs, and PBVs; flow for FCVs;
Valves can have their 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 statements.
Because of the ways in which valves are modeled the following rules apply when adding valves to a network:

- A PRV, PSV or FCV cannot be directly connected to a reservoir or tank (use a length of pipe to separate the two)
- PRVs cannot share the same downstream node or be linked in series
- Two PSVs cannot share the same upstream node or be linked in series
- A PSV cannot be connected to the downstream node of a PRV

5.2. Non-Physical Components

In addition to physical components, EPANET employs three types of informational objects – curves, patterns, and controls - that describe the behavior and operational aspects of a distribution system.

5.2.1. Curves

Curves are objects that contain data pairs representing a relationship between two quantities. Two or more objects can share the same curve. An EPANET model can utilize the following types of curves:

- Pump Curve
- Efficiency Curve
- Volume Curve
- Head Loss Curve Pump Curve
- Valve characteristic curve (Not available for the epanet project type)

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 liquid 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.
EPANET will use a different shape of pump curve depending on the number of points supplied.

Single-Point Pump Curve – A single-point pump curve is defined by a single head-flow combination that represents a pump's desired operating point. EPANET 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. Fig. 5.2 shows an example of a single-point pump curve.

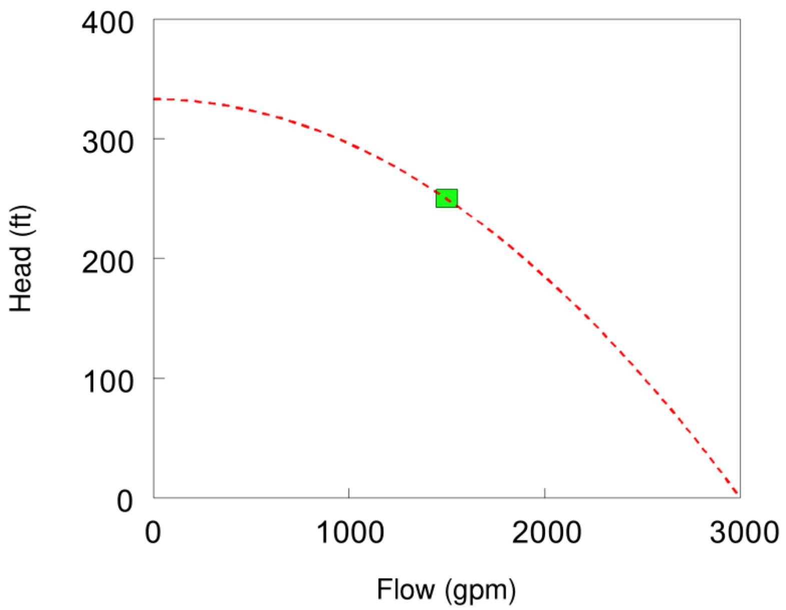


Fig. 5.2 Single-point pump curve.

Three-Point Pump 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). EPANET tries to fit a continuous function of the form

$$h_G = A - B \bullet q^C$$

through the three points to define the entire pump curve. In this function, h_G = head gain, q = flow rate, and A, B and C are constants. Fig. 5.3 shows an example of a three-point pump curve.

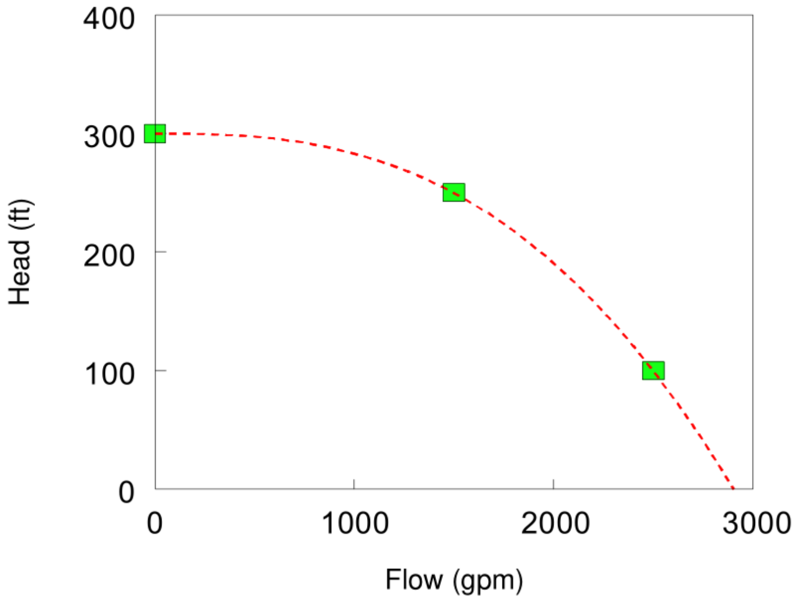


Fig. 5.3 Three-point pump curve.

Multi-Point Pump Curve – A multi-point pump curve is defined by providing either a pair of head-flow points or four or more such points. EPANET creates a complete curve by connecting the points with straight-line segments. Fig. 3.4 shows an example of a multi-point pump curve.

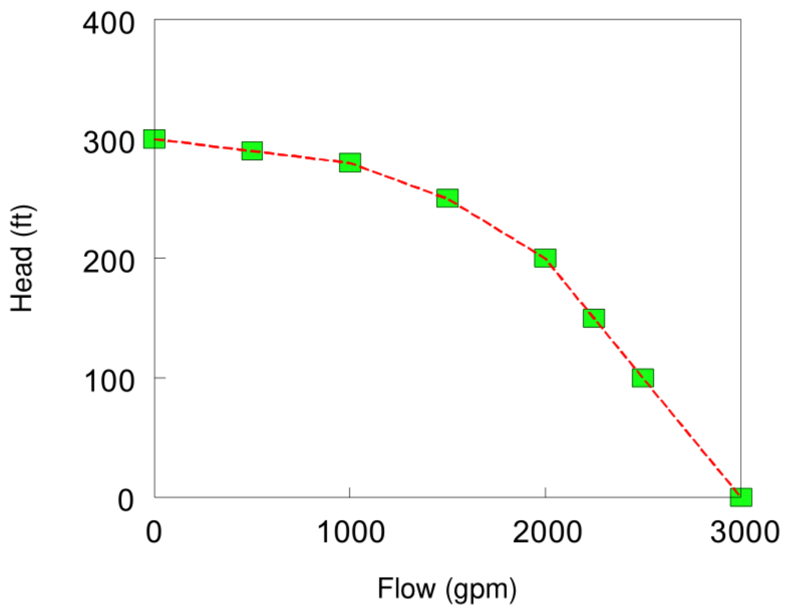


Fig. 5.4 Multi-point pump curve.

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

$$Q_1/Q_2 = N_1/N_2$$

and

$$H_1/H_2 = (N_1/N_2)^2$$

Fig. 11.5 shows an example of a variable-speed pump curve.

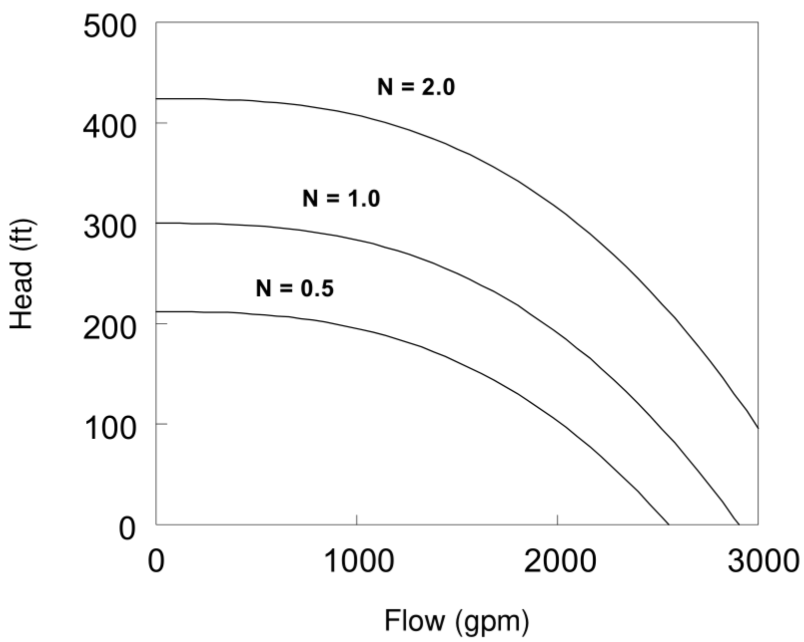


Fig. 5.5 Variable-speed pump curve.

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 in Fig. 3.6. Efficiency should represent wire-to-liquid 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. If not supplied for a specific pump then a fixed global pump efficiency will be used.

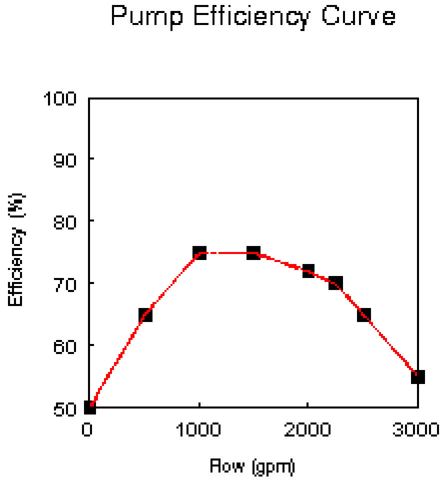


Fig. 5.6 Pump efficiency curve

Volume Curve

A Volume Curve determines how storage tank volume (Y in cubic feet or cubic meters) varies as a function of liquid level (X in feet or meters). It is used when it is necessary to accurately represent tanks whose cross-sectional area varies with height. The lower and upper liquid levels supplied for the curve must contain the lower and upper levels between which the tank operates. An example of a tank volume curve is given in Fig. 3.7.

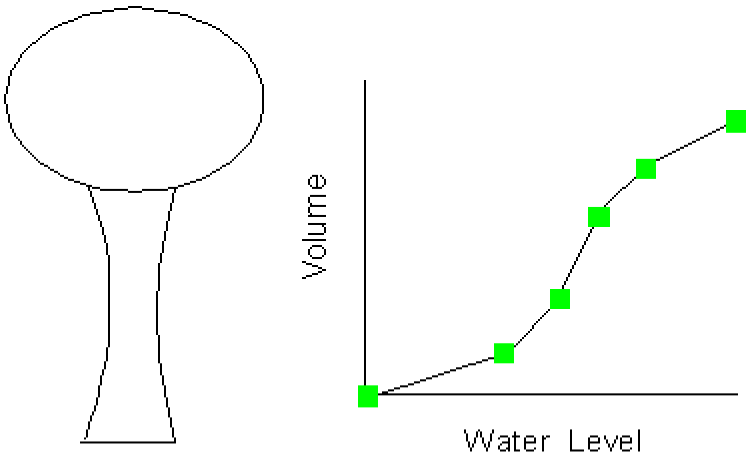


Fig. 5.7 Tank volume curve

Headloss Curve

A Headloss Curve is used to described 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.

5.2.2. Time Patterns

A Time Pattern is a collection of multipliers that can be applied to a quantity to allow it to vary over time. Nodal demands, reservoir heads, pump schedules, and liquid quality source inputs can all have time patterns associated with them. The time interval used in all patterns is a fixed value, set with the project's Time Options in the Scheduling tab of the Model properties console. Within this 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. Although all time patterns must utilize the same time interval, each can have a different number of periods. When the simulation time span exceeds the number of periods in a pattern, the pattern wraps around to its first period again.

As an example of how time 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 as in the table below:

Table 5.4 Example Time Pattern Multipliers

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:

Table 5.5 Actual Demands at Node

Hours	**0-4**	**4-8**	**8-12**	**12-16**	**16-20**	**20-24**	**24-28**
Demand	5	8	10	12	9	7	5

5.2.3. Control system

Controls are statements that determine how the network is operated over time. They specify the status of selected links as a function of time, tank liquid levels, and pressures at select points within the network. There are two categories of controls that can be used:

- Simple Controls
- Rule-Based Controls Simple Controls

Simple Controls
Simple controls change the status or setting of a link based on:

- The liquid level in a tank
- The pressure at a junction
- The time into the simulation
- The time of day

They are statements expressed in one of the following three formats:

LINK	*x*	*status*	*IF*	*NODE*	*y*	*ABOVE/BELOW*	*z*
LINK	x	status	AT	TIME	t		
LINK	x	status	AT	CLOCKTIME	c	AM/PM	

where: | x = a link ID label, | status = OPEN or CLOSED, a pump speed setting, or a control valve setting, | y = a node ID label, | z = a pressure for a junction or a liquid

level for a tank, | t = a time since the start of the simulation (decimal hours or hours:minutes), | c = a 24-hour clock time.
Some examples of simple controls are (Table 5.6):

Table 5.6 Examples of Simple Controls	
Control Statement	**Meaning**
LINK 12 CLOSED IF NODE 23 ABOVE 20	(Close Link 12 when the level in Tank 23 exceeds 20 ft.)
LINK 12 OPEN IF NODE 130 BELOW 30	(Open Link 12 if the pressure at Node 130 drops below 30 psi) drops below 30 psi)
LINK 12 1.5 AT TIME 16	(Set the relative speed of pump 12 to 1.5 at 16 hours into the simulation)
LINK 12 CLOSED AT CLOCKTIME 10 AM LINK 12 OPEN AT CLOCKTIME 8 PM	(Link 12 is repeatedly closed at 10 AM and opened at 8 PM throughout the simulation)

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 liquid above the tank bottom, not the elevation (total head) of the liquid surface.
Note: Using a pair of pressure controls to open and close a link 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 controls might provide more stability.

Rule-Based Controls
Rule-Based Controls allow link status and settings to be based on a combination of conditions that might exist in the network after an initial hydraulic state of the system is computed. Here are several examples of Rule-Based Controls:

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
```

A description of the formats used with Rule-Based controls can be found in section Rule-Based Controls syntax.

5.3. Hydraulic Simulation Model

EPANET's hydraulic simulation model computes junction heads and link flows for a fixed set of reservoir levels, tank levels, and liquid demands over a succession of points in time. From one time step to the next reservoir levels and junction demands are updated according to their prescribed time patterns while tank levels are updated using the current flow solution. The solution for heads and flows at a particular point in time involves solving simultaneously the conservation of flow equation for each junction and the headloss relationship across each link in the network. This process, known as "hydraulically balancing" the network, requires using an iterative technique to solve the nonlinear equations involved. EPANET employs the "Gradient Algorithm" for this purpose. Consult chapter Analysis Algorithms for details.

5.3.1. Pressure driven demand

EPANET may 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 for example when portions of the network can only receive fluid through links that have been closed off. As a resolution, a pressure driven analysis (PDA) can be performed to determine a hydraulic solution assuming a pressure-demand relationship at junctions. The hydraulic solution found will have curtailed or zero flow rates at low pressure and negative pressures will be largely eliminated. Only available for the epanet project type. Consult chapter Analysis Algorithms for details.

The hydraulic time step used for extended period simulation (EPS) can be set by the user. A typical value is 1 hour. Shorter time steps than normal will 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

5.4. Liquid Quality Simulation Model

5.4.1. Basic Transport

EPANET's liquid quality simulator uses a Lagrangian time-based approach to track the fate of discrete parcels of liquid as they move along pipes and mix together at junctions between fixed-length time steps. These liquid 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.

The method tracks the concentration and size of a series of non-overlapping segments of liquid that fills each link of the network. As time progresses, the size of the most upstream segment in a link increases as liquid enters the link while an equal loss in size of the most downstream segment occurs as liquid leaves the link. The size of the segments in between these remains unchanged.

For each liquid quality time step, the contents of each segment are subjected to reaction, a cumulative account is kept of the total mass and flow volume entering each node, and the positions of the segments are updated. New node concentrations are then calculated, which include the contributions from any external sources. Storage tank concentrations are updated depending on the type of mixing model that is used (see below). Finally, a new segment will be created at the end of each link that receives inflow from a node if the new node quality differs by a user-specified tolerance from that of the link's last segment.

Initially each pipe in the network consists of a single segment whose quality equals the initial quality assigned to the upstream node. Whenever there is a flow reversal in a pipe, the pipe's parcels are re-ordered from front to back.

5.4.2. Mixing in Storage Tanks

EPANET can use four different types of models to characterize mixing within storage tanks:

- Ideal (Complete Mixing)
- Two-Compartment Mixing
- First-in First-out (FIFO) Plug Flow
- Last-in First-out (LIFO) Plug Flow

Different models can be used with different tanks within a network.

The Complete Mixing model (Fig. 3.8) assumes that all liquid that enters a tank is instantaneously and completely mixed with the liquid already in the tank. It is the simplest form of mixing behavior to assume, requires no extra parameters to describe it, and seems to apply quite well to a large number of facilities that operate in fill-and-draw fashion.

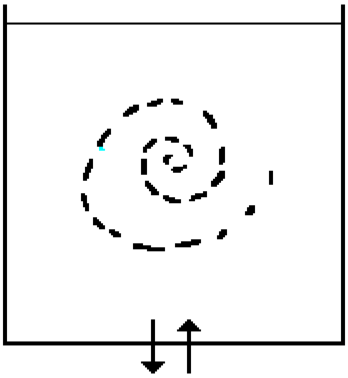


Fig. 3.8 Complete mixing.

The Two-Compartment Mixing model (Fig. 3.9) 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 liquid that enters the tank mixes with the liquid in the first compartment. If this compartment is full, then it sends its overflow to the second compartment where it completely mixes with the liquid already stored there. When liquid leaves the tank, it exits from the first compartment, which if full, receives an equivalent amount of liquid 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.

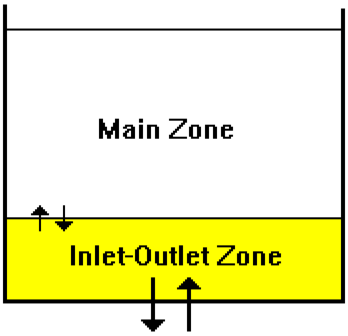


Fig. 5.9 Two-compartment mixing.

The FIFO Plug Flow model (Fig. 3.10) assumes that there is no mixing of liquid at all during its residence time in a tank. Liquid 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.

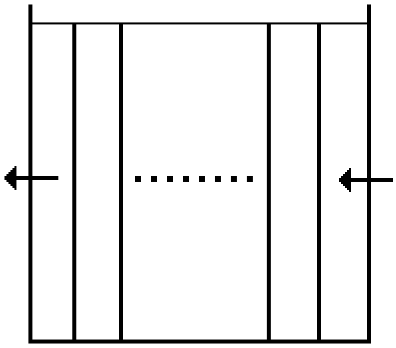


Fig. 5.10 Plug flow - FIFO.

The LIFO Plug Flow model (Fig. 3.11) also assumes that there is no mixing between parcels of liquid that enter a tank. However in contrast to FIFO Plug Flow, the liquid parcels stack up one on top of another, where liquid 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.

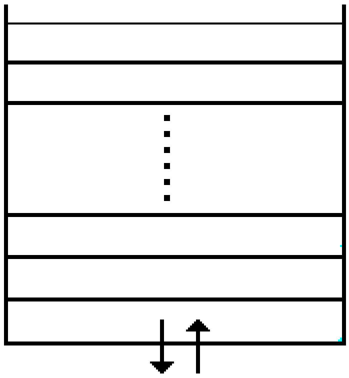


Fig. 5.11 Plug flow - LIFO.

5.4.3. Chemical Reactions

EPANET can track the growth or decay of a substance by reaction as it travels through a distribution system. 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 Fig. 3.12. 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. EPANET allows a modeler to treat these two reaction zones separately.

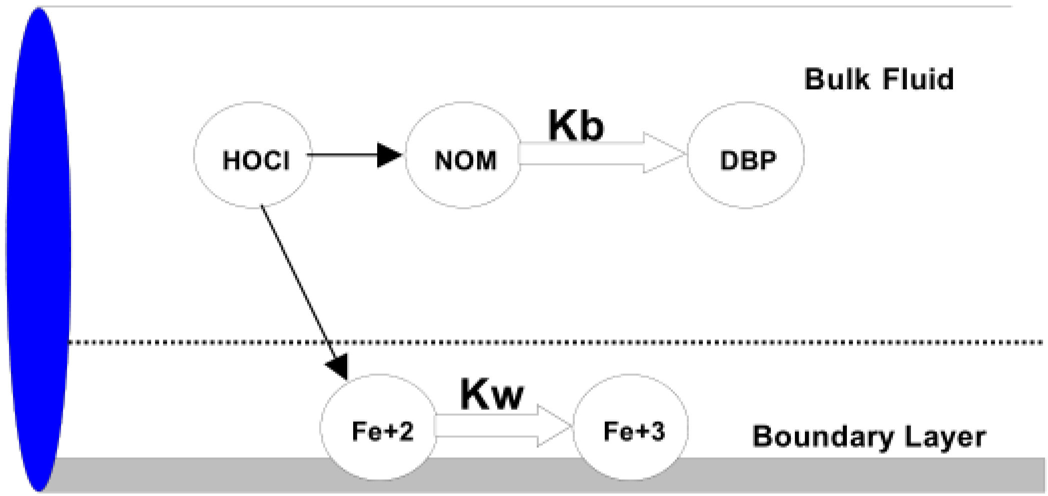


Fig. 5.12 Reaction zones within a pipe.

Bulk Reactions
EPANET 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

$$R = K_b \cdot C^n$$

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 power (1 - n) divided by time. It is positive for growth reactions and negative for decay reactions.

EPANET 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

$$R = K_b(C_L - C) \times C^{(n-1)}$$
$$for\ n > 0, K_b > 0$$

$$R = K_b(C - C_L) \times C^{(n-1)}$$
$$for\ n > 0, K_b < 0$$

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 are provided in Table 5.7 (see Water quality for more examples):

Table 5.7 Special Cases of Well-known Kinetic Models

Model	Parameters	Examples
First-Order Decay	$C_L = 0, K_b < 0, n = 1$	Chlorine
First-Order Saturation Growth	$C_L > 0, K_b > 0, n = 1$	Trihalomethanes
Zero-Order Kinetics	$C_L = 0, K_b <> 0, n = 0$	Liquid Age
No Reaction	$C_L = 0, K_b = 0$	Fluoride Tracer

The K_b for first-order reactions can be estimated by placing a sample of liquid in a series of non-reacting glass bottles and analyzing the contents of each bottle at different points in time. If the reaction is first-order, then plotting the natural log (C_t/ C_0) against time should result in a straight line, where C_t is concentration at time *t* and C_0 is concentration at time zero. The parameter K_b would then be estimated as the slope of this line.

Bulk reaction 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

Wall Reactions

The rate of liquid 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

$$R = (A/V) \cdot K_w \cdot C^n$$

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. EPANET limits the choice of wall reaction order to either 0 or 1, 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.

The variable K_w should be adjusted to account for any mass transfer limitations in moving reactants and products between the bulk flow and the wall. EPANET does this automatically, basing the adjustment on the molecular diffusivity of the substance being modeled and on the flow's Reynolds number. Refer to the discussion in Water quality for details. (Setting the molecular diffusivity to zero will cause mass transfer effects to be ignored.)

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.

5.4.4. Liquid Age and Source Tracing

In addition to chemical transport, EPANET can also model the changes in the age of liquid throughout a distribution system. Liquid age is the time spent by a parcel of liquid in the network. New liquid entering the network from reservoirs or source nodes enters with age of zero. Liquid age provides a simple, non-specific measure of the overall quality of delivered liquid. Internally, EPANET treats age as a reactive constituent whose growth follows zero-order kinetics with a rate constant equal to 1 (i.e., each second the liquid becomes a second older).

EPANET can also perform source tracing. Source tracing tracks over time what percent of liquid reaching any node in the network had its origin at a particular node. The source node can be any node in the network, including tanks or reservoirs. Internally, EPANET treats this 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 liquid from two or more different raw liquid supplies. It can show to what degree liquid from a given source blends with that from other sources, and how the spatial pattern of this blending changes over time

5.5. Rule-Based Controls syntax

Purpose:

Defines rule-based controls that modify links based on a combination of conditions.

Format:

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
conditon_n = a condition clause
action_n = an action clause
Priority = a priority value (e.g., a number from 1 to 5)

Condition Clause Format

A condition 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
LINK 44 STATUS IS OPEN
SYSTEM DEMAND >= 1500
SYSTEM CLOCKTIME = 7:30 AM
```

The Object keyword can be any of the following:

NODE	**LINK**	**SYSTEM**
JUNCTION	**PIPE**	
RESERVOIR	**PUMP**	
TANK	**VALVE**	

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 Link-Type objects:

- **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 = LINK, 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:

```
LINK 23 STATUS IS CLOSED
PUMP P100 SETTING IS 1.5
VALVE 123 SETTING IS 90
```

Remarks:

1. Only the **RULE**, **IF** and **THEN** portions of a rule are required; the other portions are optional.
2. 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 ...
   ```

3. The **PRIORITY** value is used to determine which rule applies when two or more rules require that conflicting actions be taken on a link. A rule without a priority value always has a lower priority than one with a value. For two rules with the same priority value, the rule that appears first is given the higher priority.

Example:

```
[RULES]
RULE 1
IF TANK 1 LEVEL ABOVE 19.1
THEN PUMP 335 STATUS IS CLOSED
AND PIPE 330 STATUS IS OPEN

RULE 2
IF SYSTEM CLOCKTIME >= 8 AM
AND SYSTEM CLOCKTIME < 6 PM
AND TANK 1 LEVEL BELOW 12
THEN PUMP 335 STATUS IS OPEN

RULE 3
IF SYSTEM CLOCKTIME >= 6 PM
OR SYSTEM CLOCKTIME < 8 AM
AND TANK 1 LEVEL BELOW 14
THEN PUMP 335 STATUS IS OPEN
```

`

5.6. Transient analysis

Sudden changes in flow velocity in a piping system for transport of liquids cause a pressure surge or shock wave, travelling at high speed through the system. This phenomenon is called hydraulic shock, or sometimes water hammer. The velocity changes involved are typically caused by a closing valve or a pump stop or start operation. Severe cases may result in loss of containment by pipe rupture or damage to equipment.

RND PipeWeb has a transient transport model which can be used to analyze the behavior of hydraulic shock phenomena and the effect of mitigation measures. The model employs the Method of Characteristics (MOC) to solve the governing system of partial differential equations for continuity and momentum balance. Various friction models, including an advanced convolution-based model are implemented, as well as a model for vapor formation by cavitation.

RND PipeWeb supports modeling of several surge mitigator equipment, including:

- Surge relieve valve
- Compressed gas accumulator vessel
- Rupture disk
- Standpipe

The implementation of the transient model and boundary conditions are largely based on the methods described in:

Fluid Transients in Systems. By E. B. WYLIE and V. L. STREETER. Prentice Hall, 1993, ISBN 0-13-322173-3.

Other useful resources include:

- https://www.youtube.com/watch?v=bChifSS9oF4&ab_channel=SimpleEngineeringSnippets
- https://www.youtube.com/watch?v=lrt7oY5DFRQ&ab_channel=MikeCrowley
- https://tsnet.readthedocs.io/en/latest/transient.html
-

6. Tools & utilities

The App has a variety of utilities that assist the user in project construction and review. The most commonly used tools are:

- Chart configuration tool
- Object alignment utility
- Search utility
- Window arrangement utility
- Picture management
- Preferences

6.1. Chart configuration

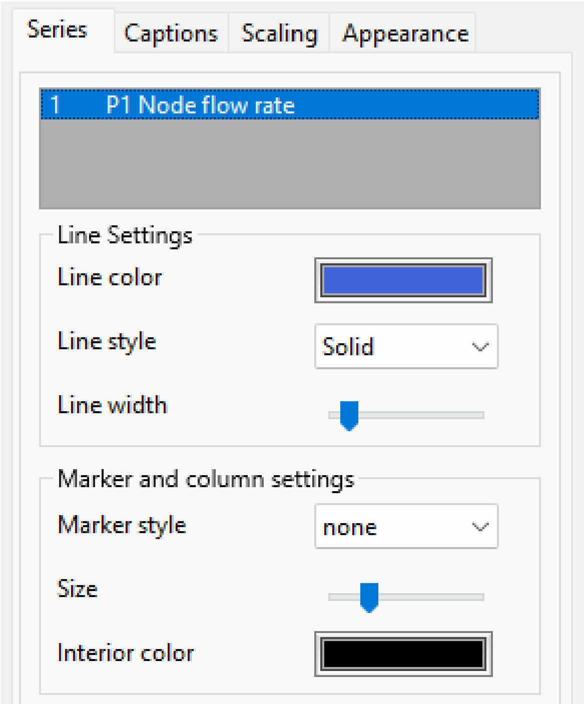
This chapter discusses configuration properties of chart properties which affect chart display on the viewport.

Chart properties are available when the value *Chart* has been selected for the Shape node property. The **Config** button adjacent to the Shape drop-down control toggles the display of **chart properties pane**.

The display properties of a chart are controlled on the chart panel. The chart panel has four tab-pages, each presenting a specific category of chart elements.

6.1.1. Series

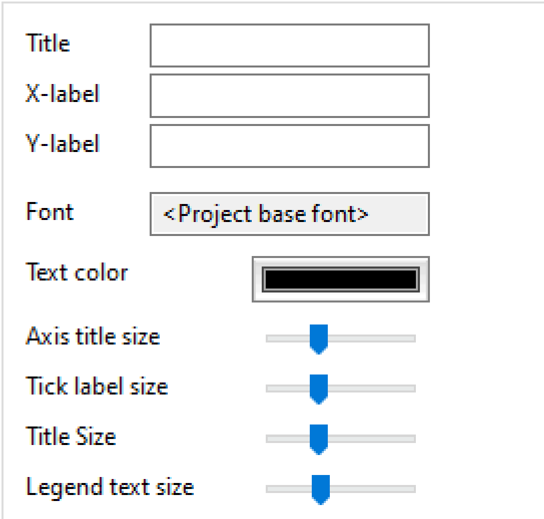
The Series tab controls the styling attributes of the individual data series of the chart. The series list at the top allows creation, deletion, renaming, reordering and selection of a series item.



The properties of a single selected series are managed by the controls in the **Line settings** frame and the **Marker and column settings** frame, which control such properties as line width and the marker symbol used to indicate data points.

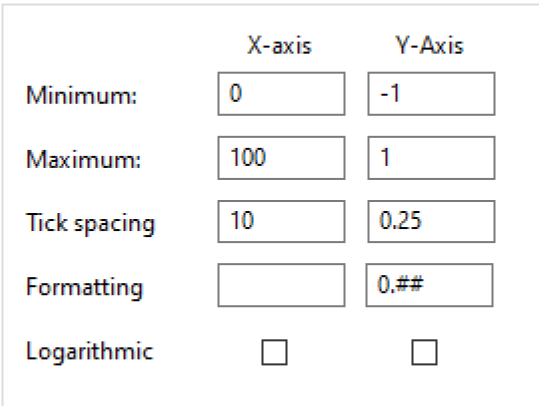
6.1.2. Captions

The Captions tab controls the font, size and color of text displayed on the chart, as indicated by the labels on the tab.



6.1.3. Scaling

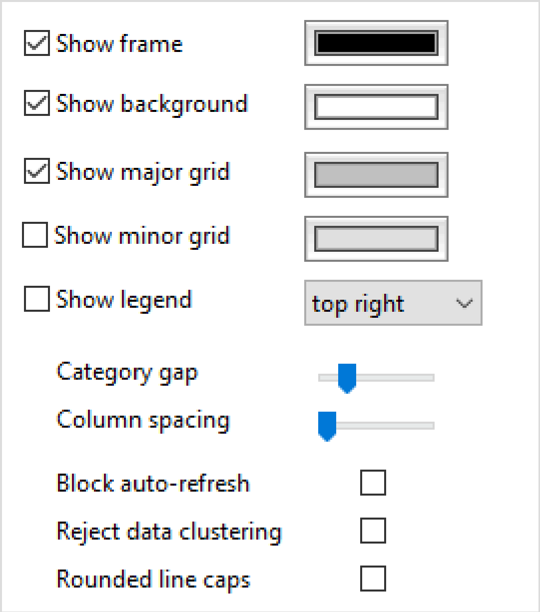
The Scaling tab controls scale ranges of the horizontal and vertical axes of the chart



The text boxes labeled **Formatting** determine the format of the tick labels, specified by Number format characters as also used for clock time display format. The **Logarithmic** checkboxes independently engages a log scale for both axes. The x-axis scale range of line charts can be focused dynamically by rotating the **mouse wheel** while hoovering over the horizontal chart axis in the viewport with the control and shift key both pressed.

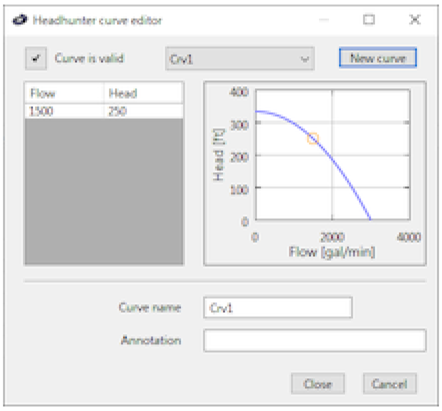
6.1.4. Appearance

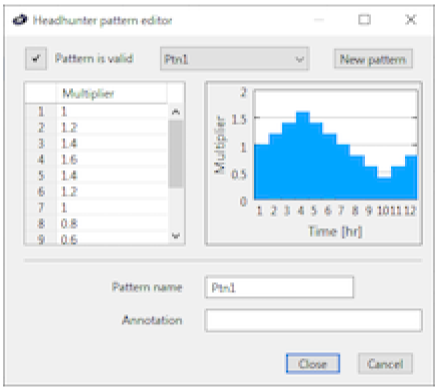
The Appearance tab controls a choice of additional decoration elements and style features of the chart. The check boxes are used to toggle the visibility of specific chart elements. The **Category gap** and **Column spacing** sliders allow adjustment of column width and spacing in column charts. The **Block autorefresh** check box exempts the chart from updates by regular animation update intervals. The **Reject data clustering** check box disables data compression for accelerated plotting of large data sets. The **Rounded line caps** check box affects the line cap smoothness for thick line styles in line plots.



6.2. Curve and Pattern editor

The Pattern editor and Curve editor can be used to create, modify, and delete patterns and curves respectively. Patterns are used to allow quantities to vary in time and to define relative locations along the length of a pipe link. Curves contain data pairs representing a relationship between two quantities.





Validation button

Pressing the validation button at the top left checks if the curve or pattern data supplied in the data list underneath the button is valid. If the curve or pattern is not valid, information is displayed concerning the nature of the detected error or inconsistency. The validation is also performed when a different curve or pattern object is selected by the below-mentioned **Pattern\Curve object** drop down menu and when the utility is opened.

Pattern\Curve object

Selects an object from the drop-down list. The selected pattern item can be designated **Default pattern** by the corresponding context popup-menu, activated by a right mouse button click presents. The Default Pattern is applied to the flow rate property (epanet project: Demand property) of all junction nodes which do not have a time pattern assigned to this property.

New pattern\curve

Press this button to create and select a new curve or pattern object.

Pattern\Curve data list and preview chart

Allows specification of multipliers for each time period of the pattern, or xy-data points for the curve. The length of the time period for patterns is determined by the Pattern time step property in the Scheduling tab of the Model properties console. The current values are displayed on the adjacent preview chart, ignoring empty lines. The mouse or the arrow keys can be used to change the active cell or the selected row of the list. If the last row is selected, pressing the **Ctrl+Down arrow key** adds new rows to the list. The context popup-menu, which is activated by a right mouse button click, presents additional configuration methods for the data list. For the transient valve characteristic curve type the contextual menu allows for switching between Kv and Cv definitions of the flow coefficient, as well as between absolute values and values as fraction of the fully opened valve maximum. Finally, the contextual menu can be used to adopt typical curve data for some common value types, as published in the Val-Matic white paper *Surge Control in Pumping Systems* (relative values only).

Pattern\Curve name
Name of the pattern or curve object (maximum of 24 characters)

Annotation
Text line that represents a description or other information about the pattern or curve object.

Close
Saves the changes made and closes the editor utility window. The selected curve or pattern is assigned to the property for which the editor utility was opened.

Cancel
Closes the curve editor utility window without storing any changes.

6.3. Minor loss evaluation tool

Minor losses are the pressure losses attributed flow through fittings, valves, bends, elbows, tees, inlets, exits and diameter changes. In large networks with long pipes, pressure losses due to such components are minor compared with frictional losses of the pipe itself. The evaluation tool allows minor loss estimation for commonly applied components, based on based on experimentally determined data. The tool presents a list of the available fitting and valves. It is opened by the Minor loss tool button on the Configuration window.

Minor losses

No.	Kf [-]	Fitting	Characteristic	Type
48	**16.5**	Total		
		· Elbows and bends ·		
	0.35	45° elbow		Standard
	0.2			Long radius
16	0.75	90° elbow		Standard
2	0.45			Long radius
	1.3			Square or miter
	1.5	180° bend		Close return
		· Connectors ·		
	1	Tee		Combining or dividing streams
26	0.1			Stagnant branch
	1.2			Elbow flow, entering branch
	1.7			Elbow flow, leaving branch
	0.04	Coupling, union		Screw connection
		· Transitions ·		
1	0.691	Reducer	0.75	120° - 180° sudden contraction ▼
		Expander		45° - 180° sudden enlargement ▼
	0.5	Pipe nozzle entrance		Sharp
	0.05			Rounded
		· Valves ·		
1	0.17	Gate valve		Open
	2.3	Diaphragm valve		Open
	9	Globe valve		Plug disk, open
	2	Angle valve		Open
2	0.05	Ball valve		Full bore, open
	0.24	Butterfly valve		Open
	2	Check valve		Swing, open
	10			Disk, open
	70			Ball, open
		· Supplementary ·		
		Custom A		
		Custom B		
		Custom C		

Line size correction factor 6

The items in the list are sorted by class and category in the third column, with a short type description in the last column. The table cells with a white background can be edited upon a mouse click on the cell. The first column specifies the number of items present in the selected pipe and the second column provides the applied loss coefficient for each item. The top row gives the resulting summations. The Reducer and Expander fitting types require specification of the Ratio d/D of the smaller to the larger pipe diameter in the fourth column, and the transition sharpness qualifier selected by a drop-down menu in the last column. The last three rows contain supplemental items with a user-specified loss coefficient in the second column and description in the last column.

The applied values for the minor loss coefficients are based the following references:

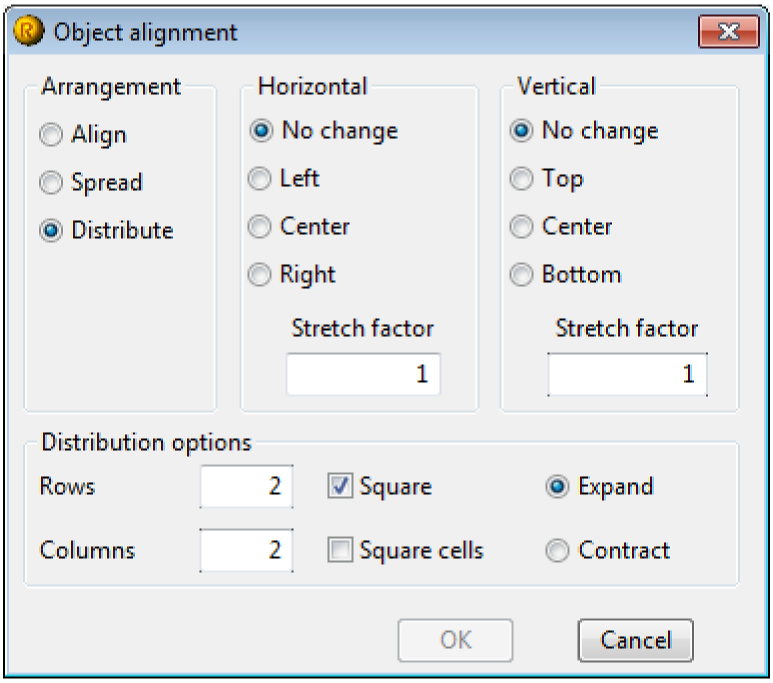
- *Perry's Chemical Engineers' Handbook. New York :McGraw-Hill, 7th ed, 1999*
- *Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410*

The table does not have an entry for the loss coefficient of a pipe exit into a tank or reservoir. The corresponding loss coefficient of 1.0 is automatically applied in the hydraulic model and needs not be specified as a minor loss.

Underneath the table is a text field that permits specification of a multiplication factor for all loss coefficients of a selected pipe. This correction factor permits manual adjustment of the loss coefficients for large pipe diameters, as has been suggested by some publications.

6.4. Object alignment utility

The alignment utility allows alignment of the position of nodes on the Viewport. It is activated by clicking the **More...** item of the Align submenu of the Format menu. Alignment only affects nodes selected in the Viewport. While the alignment utility is activated the Workspace window and all Fusion windows are disabled until the alignment utility is closed.



The frames labeled Horizontal and Vertical determine the alignment orientation. The options in the **Arrangement** frame determine the alignment method. The default *Align* option allows alignment of the node object by the options selected in the adjacent **Horizontal** and Vertical frames. Selection of the *Spread* option allows scaling of the node arrangement by a factor specified in the **Stretch factor** text fields. The *Distribute* option allows rearrangement organized in rows and columns in the configuration specified by the items in the Distribution options frame.

No change

No changes are made to the node positions in either the horizontal or vertical direction.

Left, Center, Right

Aligns the horizontal position of respectively the left-most edges, the center and right-most edges of the selected node images to the node object last selected. If all objects were selected by a rectangular sweep the alignment reference is the horizontal center of the selection.

Top, Center, Bottom

Aligns the vertical position of respectively the top-most edges, the center and bottom-most edges of the selected node images to the node object last selected. If all objects were selected by a rectangular sweep the alignment reference is the vertical center of the selection.

Stretch factor

Scales the realignment by horizontal and vertical scaling factors

Row count
Number of rows formed by the alignment.

Column count
Number of columns formed by the alignment.

Square
Rearranges the node objects into array with square outline.

Square cells
Rearranges the node objects into an array with square cells.

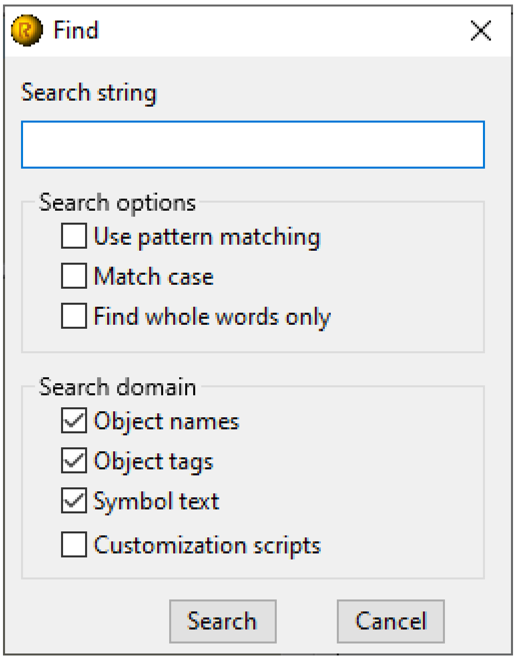
Expand
Increases the width or height of the selection span to obtain square or square cell distribution.

Contract
Decreases the width or height of the selection span to obtain square or square cell distribution.

6.5. Search utility

The search utility can be used to search for objects in the viewport. It is activated by the **Find** item of the Edit menu, or by pressing the shortcut key combination **<ctrl>F** in the Viewport. The utility can be opened from the Workspace window and from a Fusion window. The search domain includes only objects in visible layers of the viewport of the window from which the utility was opened. The utility searches for a user-specified string. All objects that satisfy the search criterion are selected in the Viewport.

When the utility is activated the Find window appears and the Main window and all Fusion windows are disabled until the search utility is closed. The utility window has a **Search text** field, where the search string is entered and two option frames which determine the search domain and method.



6.5.1. Search options

Use pattern matching

Enables pattern matching using Regular expressions.

Match case

Enables case sensitive searching.

Find whole words only

Restricts the search to whole words.

6.5.2. Search domain

Object names

Searches the Name property of node and link objects.

Object tags

Searches in Tags of node and link objects.

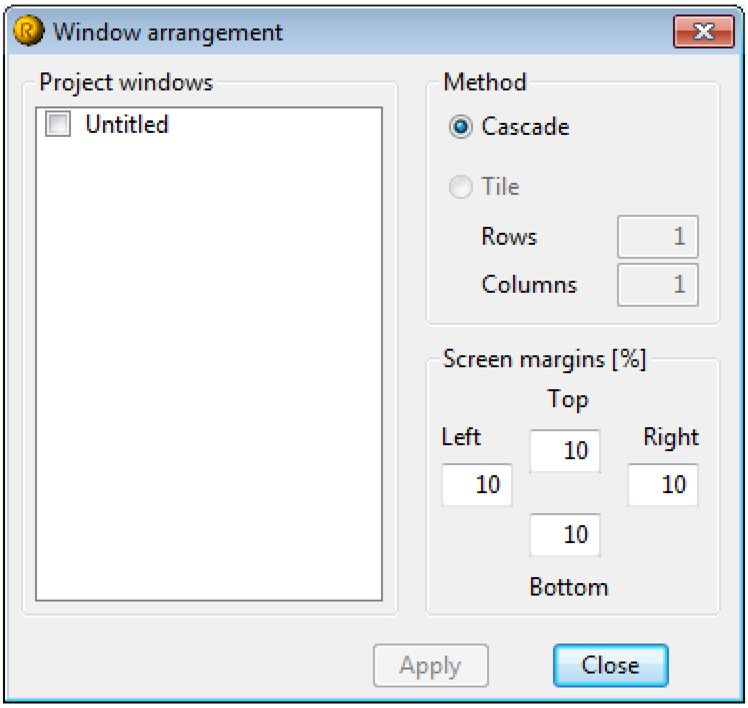
Symbol text

Searches the Text property of node symbol objects.

6.6. Window arrangement utility

The window arrangement utility can be used to arrange the various windows opened by the app on the monitor screen. It is activated by the **Arrange Windows** item of the Window menu.

The utility window contains three frames. The Method frame determines the way the window rearrangement is carried out. The frame labeled *Windows* controls which windows are rearranged and determines the repositioning order of the windows. The Screen margins frame defines a screen area for placement of the rearranged windows. On a system with multiple monitors the windows are placed on the monitor on which the utility window is displayed.



Project windows

The list in the Project windows frame displays the name of each window of the project that may be subjected to rearrangement. The order of the listed items can be modified by drag-drop operations. The arrangement is performed for all items checked in the list.

Arrangement method

Defines the arrangement method. Application of the **Cascade** option stacks the windows in a such manner that overlap of the title bar of the rearranged windows is minimized. The **Tile** option arranges the windows in as many **Rows** and **Columns** as specified in the two associated text fields, in such a way that no window overlaps any other window. For both options the windows are uniformly resized.

Screen margins [%]

Defines the margins on all four edges of the target area in percentages of the screen height or width.

6.7. Picture management

The picture management console is activated by the **Pictures** command of the Tools menu. The picture repository contains pictures that can be used to represent nodes in the viewport. The console has a picture repository to the left and a tab strip on the top right to access different picture property categories for pictures selected in the repository. The picture repository provides a list of the pictures in the project. Pictures can be added from disk by the **New** command of the contextual menu of the repository, by dropping a picture file on the repository, or by pasting a bitmap from the clipboard into the repository. The picture viewer on the bottom right displays the bitmap of the pictures selected in picture repository organized in a grid. A click on a picture in the picture viewer deselects all other pictures.

6.7.1. Masking

Transparency color

Defines the transparency mask color of the selected picture. The transparency color can be selected by the color selection dialog activated by a click on the adjacent color display field. The transparency color can also be selected by a click on a pixel in the picture viewer, provided a single picture is selected in the picture repository. The transparency mask color is opaque and does not preserve any transparency information of the selected pixel.

Layer substitution color

Defines the mask color of the selected picture to be substituted with the color of the object layer. The layer color can be selected by the color selection dialog activated by a click on the adjacent color display field. The layer color can also be selected by a click on a pixel in the picture viewer while pressing the **Shift key**, provided a single picture is selected in the picture repository. The mask color is opaque and does not preserve any transparency information of the selected pixel.

6.7.2. Knots

Add knot

Activates knot creating. Picture knots can be used as anchor for attachment of a link to a node represented by the picture, with the Icon fit value *Filled fit* selected. A click on the picture viewer with activated knot creating results into creation of a new knot in the picture on the mouse position, with the knot location indicated in pixels in the text fields labeled X and Y.

Required link snap opacity

Specifies the minimum opacity level percentage required for attachment of a link to a node represented by the picture.

6.7.3. Transform

Crop transparent edges

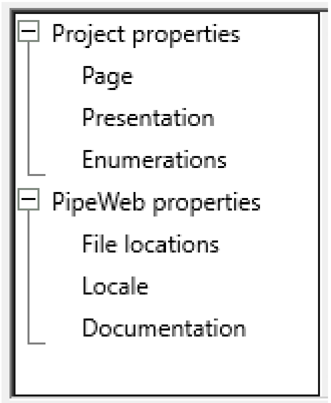
Crops the picture by removing transparent edges on all four sides of the items selected in the picture repository.

Replace

Activates a file browser to select a picture file for replacement of the items selected in the picture repository.

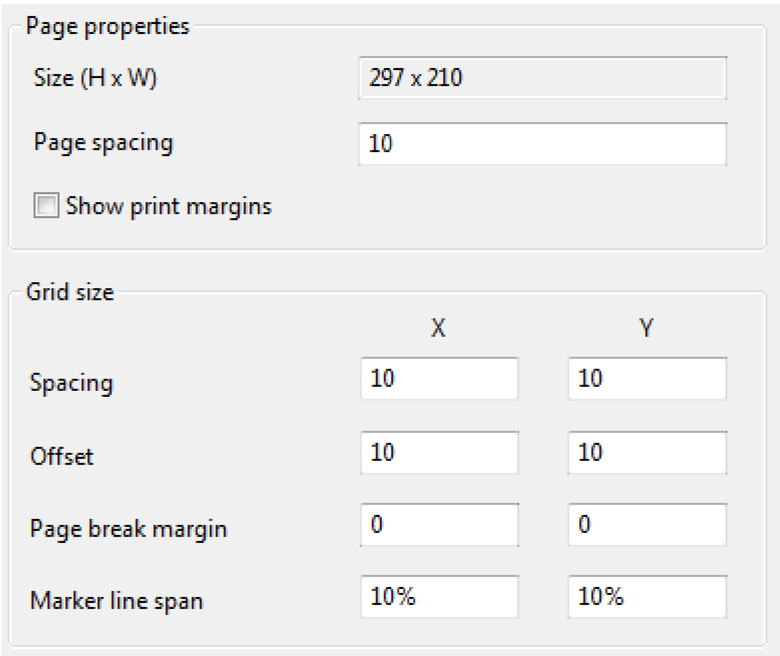
6.8. Preferences

The Preferences console is activated by the **Preferences** command of the Tools menu. Selecting an item in the category list at the left side of the console presents the corresponding properties pane at right.



6.8.1. Page layout

The page properties pane determines the page properties of the project.



Size (H x W)

Displays the page size expressed in the selected for the Project base unit.

Page spacing

Determines the page distance on the viewport in the value selected for the Project base unit.

Show print margins

If checked, the page margins of the selected printer are displayed by a line in the viewport.

6.8.2. Grid size

Spacing

Determines the distance between the grid lines in the viewport specified in the value selected for the Project base unit.

Offset

Determines the distance of the top and left grid line to the page edge specified in the value selected for the Project base unit.

Page break margin

Determines the required minimum distance to the right and bottom page edges for gridlines specified in the value selected for the Project base unit.

Marker line span

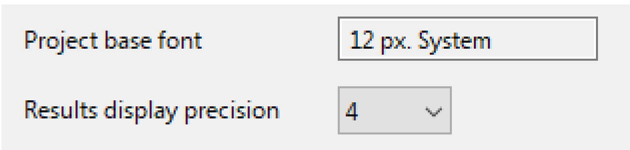
Determines the display length of the gridlines as percentage of the spacing.

Project base unit

Determines the base unit for size display and specification.

6.8.3. Presentation

Determines the display properties of the viewport.



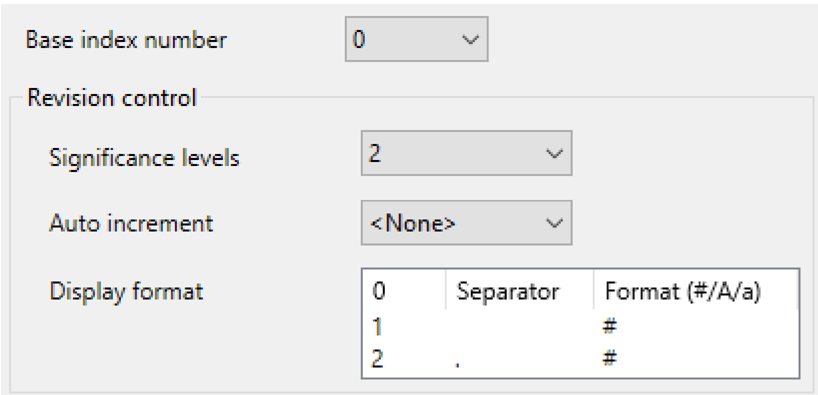
Project base font

Determines the default font for text in the viewport. A click on the text field opens the font picker tool, by which the default font can be changed,

Results display precision

Determines the number of significant digits for display of numerical results.

6.8.4. Enumerations



Base index number

Determines the lowest index number assigned to new link and node objects. The dropdown menu allows selection of the values *0* or *1*.

Revision control
The revision control frame determines the method by which version identification is organized.

Significance levels
Determines the revision enumeration depth. Revision enumeration is applied to distinguish significance levels of revisions. The lowest revision significance level can be updated automatically.

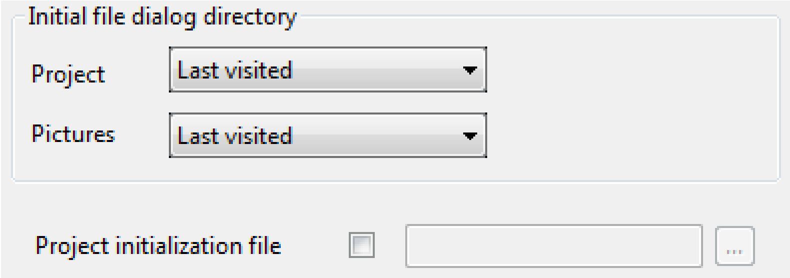
Auto increment
Determines the method for automated increment of the lowest significance enumeration. The dropdown menu allows selection of the values <*None*>, *On save* and *On exit*. The value <*None*> requires manual revision assignment by the contextual menu of the Revision text field.

Display format
Determines the revision display style in the Revision text field and by Viewport object text codes. The string specified in the column labeled *Separator* is shown between the significance level values. The *Format* column requires a selection out of the three characters to specify the revision display format. A series of hashtag characters (#) represent the minimum number of digits displayed for the integer number. The value *A* presents uppercase letters in the scheme A, B, C ... AA, AB... The value *a* presents lowercase letters by the same scheme. If no format is specified the revision enumeration is displayed as unformatted number.

6.8.5. File locations

Initial file dialog directory

Project
Determines the folder for the **Open** and **Save** commands of the File menu.



The three options are: The *Last visited* folder by file operations within the app, the *Documents folder* of the operating system, and a *Specified location*, which is assigned in the adjacent text field that appears when selected.

Pictures
Determines the browsing folder for new pictures created by the Picture management utility.

Project initialization file
Enables loading a project file with every new project. The file and path name are shown in the adjacent text field. The initialization file is selected using the browse button to the right of the text field.

6.8.6. Locale

Units system

Determines the units of measurement preference for the application. The dropdown menu has the values *Metric*, *Imperial* and *None.* The selection affects the default value for the Project base unit and evaluation of standard printer paper dimensions. The values *Metric* or *Imperial* are recommended if high accuracy page size representation is expected for a matching page setup selection.

6.8.7. Documentation

Source Location

Select the value *Local* to use the help files locally installed on the system instead of the default online documentation location. Available for MS-Windows only and must be set each time the application starts.

7. Appendices

This section provides general information associated with the PipeWeb software application for reference.

7.1. Regular expressions

This section describes the syntax of regular expressions.

Pattern	Description
.	Matches any character except newline.
[a-z0-9]	Matches any single character of set.
[^a-z0-9]	Matches any single character not in set.
\d	Matches a digit. Same as [0-9].
\D	Matches a non-digit. Same as [^0-9].
\w	Matches an alphanumeric (word) character - [a-zA-Z0-9_].
\W	Matches a non-word character [^a-zA-Z0-9_].
\s	Matches a whitespace character (space, tab, newline, etc.).
\S	Matches a non-whitespace character.
\n	Matches a newline (line feed).
\r	Matches a return.
\t	Matches a tab.
\f	Matches a form feed.
\b	Matches a backspace.
\0	Matches a null character.
\000	Also matches a null character because of the following:
\	Matches an ASCII character of that octal value.
\x*nn*	Matches an ASCII character of that hexadecimal value.
\c*X*	Matches an ASCII control character.
\metachar	Matches the meta-character (e.g., \, .).
(abc)	Used to create subexpressions. Remembers the match for later backreferences. Referenced by replacement patterns that use \1, \2, etc.
\1, \2,…	Matches whatever first (second, and so on) of parens matched.
x?	Matches 0 or 1 **x'**s, where x is any of above.
x*	Matches 0 or more **x'**s.
x+	Matches 1 or more **x'**s.
x{**m**,**n**}	Matches at least **m** **x'**s, but no more than **n**.
abc	Matches all of a, b, and c in order.
a|b|c	Matches one of a, b, or c.
\b	Matches a word boundary (outside [] only).

\B	Matches a non-word boundary.
^	Anchors match to the beginning of a line or string.
$	Anchors match to the end of a line or string.

7.1.1. Replacement Patterns

The following expressions can only apply to the replacement pattern:

Pattern	Description
$'	Replaced with the entire target string before match.
$&	The entire matched area; this is identical to \0 and $0.
$'	Replaced with the entire target string following the matched text.
$0-$50	$0-$50 evaluate to nothing if the subexpression corresponding to the number doesn't exist.
\0-\50	
\x*nn*	Replaced with the character represented by *nn* in Hex, e.g., ™is ™.
\n*nn*	Replaced with the character represented by *nn* in Octal.
\c*X*	Replaced with the character that is the control version of *X*, e.g., \cP is DLE, data line escape.

7.1.2. Regular Expression Examples

The basic idea of regular expressions is that it enables you to find and replace text that matches the set of conditions you specify. It extends normal Search and Replace with pattern searching.

Wildcards

Some special characters are used to match a class of characters:

Wildcard	Matches
.	Any single character except a line break, including a space.

If you use the "." as the search pattern, you will select the first character in the target string and, if you repeat the search, you will find each successive character, except for Return characters.
The following wildcards match by position in a line:

Wildcard	Matches	Example
^	Beginning of a line (unless used in a character class; see below)	^Phone: Finds lines that begin with "Phone":
$	End of a line (unless used in a character class)	$: Finds the last character in the current line.

Character Classes

A character class allows you to specify a set or range of characters. You can choose to either match or ignore the character class. The set of characters is enclosed in brackets. If you want to ignore the character class instead of match it, precede it by a caret (^). Here are some examples:

Character Class	Matches
[aeiou]	Any one of the characters a, e, i, o, u.
[^aeiou]	Any character except a, e, i, o, u.
[a-e]	Any character in the range a-e, inclusive
[a-zA-Z0-9]	Any alphanumeric character.
[[]	Finds a [.
[]]	Finds a]. To find a closing bracket, place it immediately after the opening bracket.
[a-e^]	Finds a character in the range a-e or the caret character. To find the caret character, place it anywhere except as the first character after the opening bracket.
[a-c-]	Finds a character in the range a-c or the - sign. To match a -, place it at the beginning or end of the set.

Non-printing Characters

You can use the following notation to find non-printing characters:

Special Character	Matches
\r	Line break (return)
\n	Newline (line feed)
\t	Tab
\f	Formfeed (page break)
\x*NN*	Hex code **NN**.

Other Special Characters

The following patterns are wildcards for the following special characters:

Special Character	Matches
\s	Any whitespace character (space, tab, return, linefeed, form feed)
\S	Any non-whitespace character.
\w	Any "word" character (a-z, A-Z, 0-9, and _)
\W	Any "non-word" character (All characters not included by \w).
\d	Any digit [0-9].
\D	Any non-digit character.

Repetition Characters

Repetition characters are modifiers that allow you to repeat a specified pattern.

Repetition Character	Matches	Examples
*	Zero or more characters.	d* finds no characters, or one or more consecutive "d"s. .* finds an entire line of text, up to but not including the return character.
+	One or more characters.	d+ finds one or more consecutive "d"s. [0-9]+ finds a string of one or more consecutive numbers, such as "90404", "1938", the "32" in "Win32", etc.
?	Zero or one characters.	d? finds no characters or one "d".

Please note that, since * and ? match zero instances of the pattern, they always succeed but may not select any text. You can use them to specify an optional character, as in the examples in the following section.

"Greediness"

The "?" is used as a "greediness" modifier for a subpattern in a regular expression. You can place a "?" directly after a * or + to reverse the "greediness" setting. That is, if Greedy is True, using the ? after a * or + causes it to match the minimum number of times possible: For example, consider the following.

Target String	Greedy	Regular Expression	Result
aaaa	True	(a+?) (a+)	$1=a, $2=aaa
aaaa	False	(a+?) (a+)	$1=aaa, $2=a

Extension Mechanism

We also support the regular expression extension mechanism used in Perl. For instance:

(?#text)	Comment
(?:pattern)	For grouping without creating backreferences
(?=pattern)	A zero-width positive look-ahead assertion. For example, \w+(?=\t) matches a word followed by a tab, without including the tab in $&.
(?!pattern)	A zero-width negative look-ahead assertion. For example foo(?!bar)/matches any occurrence of "foo" that isn't followed by

	"bar".
(?<=pattern)	A zero-width positive look-behind assertion. For example, (?<=\t)\w+ matches a word that follows a tab, without including the tab in $&. Works only for fixed-width look-behind.
(?<!pattern)	A zero-width negative look-behind assertion. For example (?<!bar)foo matches any occurrence of "foo" that does not follow "bar". Works only for fixed-width look-behind.

Subexpressions

You can use parentheses within your search patterns to isolate portions of the matched string. You do this when you need to refer to subsections of the matched in your replacement string. For example, you would do this if you need to replace only a portion of the matched string or insert other text into the matched string.

Here is an example. If you want to match any date followed by the letters "B.C." you can use the pattern "\d+\sB\.C\." (Any number of digits followed by a space character, followed by the letters "B.C.") This will match dates such as 33 B.C., 1742 B.C., etc. However, if you wanted your replacement pattern to leave the year alone but replace the letters with something else, you would use parens. The search pattern "(\d+)\s(B\.C\.)" does this.

When you write your replacement pattern, you can refer to the year only with the variable \1 and the letters with \2.

If you write "(\d+)\s(B.C.|A.D.|BC|AD)", then \2 would contain the matched letters.

Combining Patterns

Much of the power of regular expressions comes from combining these elementary patterns to make up complex searches. Here are some examples:

Pattern	Matches
\$?[0-9,]+\.?\d*	Matches dollar amounts with an optional dollar sign.
\d+\sB\.C\.	One or more digits followed by a space, followed by "B.C."

The Alternation Operator

The alternation operator (|) allows you to match any of a number of patterns using the logical "or" operator. Place it between two existing patterns to match either pattern. You can use more than one alternation operator in a pattern:

<table>
<tr><th>Pattern</th><th>Matches</th></tr>
<tr><td>\she\s | \sshe\s</td><td>" he " or " she "</td></tr>
<tr><td>cat|dog|possum</td><td>"cat", "dog", or "possum"</td></tr>
</table>

([0-9,]+\sB\.C\.)|([0-9,]+\sA\.D\.) "'or"' [0-9,]+\s((B\.C\.)|(A\.D\.))	Years of the form "**yearNum** B.C. **or** A.D." e.g., "2,175 B.C." or "215 A.D."

7.1.3. Search and Replace

You use special patterns to represent the matched pattern. Using replacement patterns, you can append or prepend the matched pattern with other text.

Pattern	Description
$&	Contains the entire matched pattern. If "\d\d\d\d\sB\.C\." finds "1541 B.C.", then the replacement pattern "the year $&" results in "the year 1541 B.C.", as the $& contains the string "1541 B.C".
\1, \2, etc.	Contains the matched subpatterns, defined by use of parentheses in the search string. The search pattern "(\d+)\s(B\.C\.|A\.D\.|BC|AD)" looks for any number of digits followed by a space character, followed by either "B.C.", "BC", "A.D.", or "AD". The \1 variable contains the match to the "\d+" portion of the expression and \2 contains the match to the "B\.C\.|A\.D\.|BC|AD" portion.

Credits

Philip Hazel, and copyright by the University of Cambridge, England.

7.2. Text markup escape sequences

This section provides a list of escape sequences available for insertion of object or clock property values in tag or symbol text. An escape sequence consists of the backslash escape character followed by two other characters. Two consecutive backslashes represent a single literal backslash.

7.2.1. Escape sequences for object properties

An escape sequence in object property markup is replaced by a string that expresses a property value in the text displayed in the viewport. The following object escape sequences are defined in PipeWeb:

Esc. seq.	***Description***
\OT	Object type name
\OI	Object index
\Oi	Non-zero object index
\OX	Object index, not displayed if type name is unique for the object
\OP	Object icon picture name (node only)
\DT	Object drawing type name.
\DI	Object drawing index
\Di	Non-zero drawing index
\DX	Drawing index, not displayed if type name is unique for the drawing
\DR	Drawing revision
\PN	Project name
\PR	Project revision

Drawing designation

The drawing container of a node object is determined by the center of the image that represents the node in the viewport. The drawing object of a link is determined by the tag seat location of the link. If an object is positioned in the page break margin area between two pages the corresponding drawing escape sequence is undetermined, indicated by the string “<*Property* ref>”

Page reference representations

The lead characters *O* and *D* are case-sensitive for the index referencing sequence of page reference nodes. Lower case for the first character denotes the index property of the paired node of a paired page reference node, as opposed to the index property of the node itself.