

PIPENET VISION™ Spray/Sprinkler Module User and Reference Manual

Software Revision 1.5



© 2010 Sunrise Systems Limited

Foreword

Spray/Sprinkler User and Reference Manual

Sunrise Systems Limited

This manual is designed to act as a user and reference guide for the Spray/Sprinkler module revision 1.50. It contains a number of tutorial examples which should help both new users and users of previous PIPENET® modules.

PIPENET® and PIPENET VISION are registered trademarks of Sunrise Systems Limited. All other names and services mentioned in this manual that are trademarks, registered trademarks, or service marks, are the property of their respective owners.

Table of Contents

Foreword	0
Part 1 Sunrise Systems	3
1 Welcome	3
2 Sunrise - Product Modules	4
3 Obtaining Support	4
4 Sunrise - How to Contact Us	4
5 Conventions used in this document	5
Part 2 Installation	8
1 Installation	8
2 Prerequisites	8
3 Updating a key license	8
4 Security Key Problems	9
Part 3 A Tour of the User Interface	14
1 Overall view	14
2 The Schematic Window	15
3 Tabular view window	16
4 Properties Window	16
5 Component Properties	17
6 Fittings	19
7 The Schematic overview window	20
8 Menus	21
File Menu	21
Edit Menu	23
View Menu	24
Libraries Menu	25
Options Menu	26
Colouration Menu	27
Calculation Menu	27
Tools Menu	28
Window Menu	29
Help Menu	29
9 Toolbars	30
Spray Toolbar	30
Options Toolbar	31
Calculation Toolbar	32
Tag Toolbar	32
Find toolbar	33

Part 4 Example 1 - A Simple Three Pipe System	36
1 Network Representation	36
2 Labelling the diagram	36
3 Labels	36
4 Inlets and Outlets	37
5 Network Topology	38
6 Simple three pipe system	38
7 Calculation and Design Phases	39
8 Specifying Units	42
9 Creating a pipe type	43
10 Entering the pipe data	45
11 Undefined or invalid components	47
12 Specifications	48
13 Calculating	49
14 Inspecting the results	50
15 Saving and loading the network	51
Part 5 Example 2 - A Small Sprinkler System	54
1 A small Sprinkler System	54
2 Specifying units	54
3 Creating a user defined pipe schedule	54
4 Creating a nozzle type	57
5 Creating a pump library	58
6 Creating a pipe type	60
7 Entering the pipe data	60
8 Entering the pump data	62
9 Entering the nozzle data	63
10 Entering specification data	64
11 Performing calculations	65
Part 6 Example 3 - Condensate Tank Deluge System	68
1 Condensate tank deluge system	68
2 Preliminary considerations	69
3 Initialisation	70
4 Creating a user defined pipe schedule	71
5 Creating a pipe type	72

6	Nozzle characteristics	73
7	The design and calculation phases	74
8	Specifications	74
9	Network data entry	75
10	Calculation and results	77
Part 7	Example 4 - Pressures in the Ring Main	80
1	Pressures in a ring main	80
2	Initialisation	81
3	Creating a pipe type	81
4	Network data entry	82
5	Pump data and the library	84
6	Specifications	85
7	Calculation and results	86
Part 8	Example 5 - A Maintenance Area Sprinkler System	88
1	A maintenance area sprinkler system	88
2	Initialisation	89
3	Specifications	91
4	Calculation and results	92
Part 9	The Schematic	96
1	Schematic Window	96
2	Schematic Underlay	96
3	Selection Tool	97
4	Pan and Zoom Tool	98
5	Area Tool	98
6	Polygon Tool	100
7	Text Tool	101
8	Link Component Tools	102
9	Pipe Component Tools	103
10	Schematic Printing	104
11	Exporting the Schematic	105
12	Moving around the network	105
13	Use of the mouse	107
Part 10	The Tabular View	110
1	Tabular View	110
2	Validation	111

3 Copying Cells	111
Part 11 Specifications	116
1 Introduction to Specifications	116
2 Specification Rules	117
3 Breaks and Blocks	118
4 User Interface	118
5 Nozzle Specification	121
Part 12 Status Checking	124
1 Status checking	124
2 Specification Checks	124
3 Height Checking	125
Part 13 Colour Schemes	128
1 Colour Schemes	128
2 Tagging	132
3 Background Colours	132
Part 14 Elevation Profile and Hydraulic Grade Line	134
1 Elevation Profile Window	134
2 Hydraulic Grade Line	136
Part 15 Multiple Pipe Generation	140
1 Multiple Pipe Generation	140
Part 16 Autolayout and Nozzle Generation	144
1 Autolayout	144
2 Nozzle generation	148
Part 17 Libraries	152
1 Libraries	152
2 Spray Library Editor	153
3 Pipe Schedules	155
4 Nozzles	156
5 Pumps - Coefficients Unknown	157
6 Pumps - Coefficients Known	159
7 Linings	160
8 Deluge Valves	161
9 Editing system libraries	161

Part 18 Specifying Options	164
1 Title	164
2 Spray Options	164
3 Units	167
4 PipeTypes	171
5 Schematic display options	172
6 Calculation options	174
7 Spray Tables	176
8 Defaults	177
Part 19 Modelling	180
1 Remote Nozzle Specification	180
2 Mass Balance Specifications	180
3 Design Facility	180
4 Linings	182
5 Pipe Modelling	182
6 Hazen Williams	183
7 Coulson Richardson Equation Model	184
8 Pipe C-factor	185
9 Pipe hydrostatic pressure loss	185
10 Velocity Pressure	185
11 Equipment items	187
12 Filters	187
13 Deluge Valves	188
14 Non-return valve	189
15 Overboard Dump Valves	190
16 ElastomericValve	190
17 Orifice Plates	191
18 Pumps	193
19 Spray Nozzles	198
20 Fittings	199
Part 20 NFPA Submittal Report	202
1 NFPA Submittal Report	202
Part 21 Exporting the Schematic	206
1 HP-GL/2 Output	206
2 DXF Ouput	206
Part 22 Errors	210

1	Errors	210
2	Basic errors	210
3	Input errors	211
4	Global errors	211
5	Specific component errors	213
6	Numerical errors	214
Part 23 Reference Data		216
1	Bibliography	216
2	Properties of water	216
3	Surface Roughness	217
4	Hazen-Williams Coefficients	218
5	Physical Constants	220
6	FOC Rules	220
7	NFPA Rules	224
8	Chinese Standard (GB)	230
9	Schedule - BS 3505 Class D (uPVC)	230
10	Schedule - BS 1387 Heavy	231
11	Schedule - BS 1387 Medium	232
12	Schedule - ANSI B36.10 Schedule 40	233
13	Schedule - ANSI B36.10 Schedule 80	234
14	Schedule - BS 1211 Class C (cast Iron)	235
15	Schedule - Copper/Nickel (90/10) 14 bar g	236
16	Schedule - Copper/Nickel (90/10) 20 Bar g	237
Index		239

Sunrise Systems

Part

1

1 Sunrise Systems

1.1 Welcome

Welcome to the latest version 1.5 of the PIPENET® modules, collectively referred to as PIPENET VISION™. It contains many new features and improvements.

General enhancements

- The installation process has been simplified, and delivers many new resources:
 - Self-training materials
 - Demonstrations
 - Example datafiles
- Components now only change colour from blue to black when all mandatory attributes have been given valid values.
- The Options>Units tab now initially appears unexpanded, to make it easier to find and change the units of interest.
- Formatting of decimal numbers has been improved, and the requested precision is now saved correctly between sessions in all cases.
- Component types now appear alphabetically in the Data View window “Browse” selector.
- The size of network that can entirely be displayed within the view window has been increased.
- Large underlays now display correctly.
- A number of other fixes have been incorporated.

Standard and Spray/Sprinkler module enhancements

- New **elastomeric valves** achieve the required input pressure, output pressure, pressure drop or flow rate without the need to input the valve characteristic data.
- **You can use the right mouse button menu to turn nozzles within a defined area on/off (applies to Spray/Sprinkler only).**
- A new stand-alone program to convert a results file (SRF) to a spreadsheet is provided.
- The Edit Pipe type dialog now checks that increasing bores correspond to increasing flow-rate (calculated from the velocity and the pipe internal bore.)

Transient module enhancements

- A new graphing tool is provided, with improved user interface and capabilities including superimposition of multiple graphs.
- Open channel cavitation algorithms have been significantly upgraded - covering bubbly, slug, and stratified flow types.
- New geometric shapes are possible for:
 - Simple tank types (vertical cylinder, horizontal cylinder, sphere, cone frustum).
 - Accumulator types (vertical cylinder, horizontal cylinder, sphere).
 - Surge tank types (vertical cylinder, horizontal cylinder, uniform section, cone frustum).
 - Receiving vessel types (vertical cylinder, horizontal cylinder, uniform section, cone frustum).
- The new pipe bundle is a single component representing the hydraulic characteristics of a bundle of hundreds/thousands of pipes).

- A new Signal Selector component can find the minimum/maximum of two input signals, or switch from one input signal to another after a defined time period.
- A new Limiting Power Ramp option for Transfer function enables different ramp up and ramp down response rates.
- The receiving vessel model has been extended to allow the specification of various types of weir.
- Vacuum breakers can now be defined to operate only within a given time period, staying closed outside that period.
- Pressure and pressure-difference sensors are now recognized for both the analogue and digital options, whereas previously they were always assumed to be analogue.

1.2 Sunrise - Product Modules

The PIPENET suite of programs has been designed to enable the accurate simulation of the flow of fluid through a network of pipes and other components. The full suite of programs consists of the following modules:

Standard Module	For the analysis of the single phase flow of liquids and gases.
Spray Module	For the analysis of fixed fire-protection systems employing water.
Transient Module	For the analysis of transient flow in all types of network employing a liquid.

1.3 Obtaining Support

Support queries, or details of any problems experienced, should be emailed to:

support@sunrise-sys.com.

If you are experiencing problems with a specific PIPENET network please remember to attach the data files, together with any associated library files, to the email. For PIPENET VISION files include the .SDF data file and the associated .SLF file, For PIPENET Classic files include the .DAT file together with the any associated library files, .PDF, .PMP, .UFL, and .VLB. Please also include any other information that might help in locating the source of the problem.

For queries relating to installation, in particular security key problems, include the key number. For USB keys this is a five digit number of the form *1nnnn* or *2nnnn*. For parallel port keys the number is located in the bottom right-hand corner of the key label (not the Sunrise address label). Before sending any queries relating to the installation of the software make sure that you have read the trouble-shooting section in this document and provide details of any error messages encountered during installation.

1.4 Sunrise - How to Contact Us

Sunrise Systems Limited may be contacted by post, by fax, by email or via our website:

Sunrise Systems Limited

Sunrise Business Park
Ely Road
Waterbeach
Cambridge
CB25 9QZ
United Kingdom

Telephone +44 1223 441311
Fax: +44 1223 441297
email pienet@sunrise-sys.com
web site <http://www.sunrise-sys.com>

1.5 Conventions used in this document

The following are the conventions used in this manual:

- Items such as **File | New** shown in bold indicate the selection of an option from a menu. The item before the vertical bar is the main menu item and the item after the vertical bar is the specific menu option. For example; **File | New** indicates that the menu option **New** is to be selected from the **File** menu.
- Capitalised item shown in bold, for example **Apply**, generally indicate the selection of a button or item in a dialog.

Installation

Part

2

2 Installation

2.1 Installation

Installation of a PIPENET module will have been described in the installation guide, which accompanies the CD-ROM. This chapter deals with problems that may occur once the software has been installed from CD-ROM.

2.2 Prerequisites

To run any of the PIPENET modules, you need the following:

- A valid PIPENET license provided by an appropriate HASP security key
- A release CD-ROM
- A PC with an Intel Pentium IV (or equivalent) processor
- 1 G Byte memory
- At least 250 M Bytes of free disk storage
- Microsoft Windows 2000, XP, Vista or Windows 7
- A display monitor with a resolution of 1024 x 768 or even 1280 x 1024

2.3 Updating a key license

Depending on the type of licence, a security key may be restricted in its use in some way. For example, it may only license some of the PIPENET modules, it may be restricted to run a specific release, or it may have an expiry date applied to one or more modules licensed by the key.

Keys can be re-programmed without needing to returned to Sunrise Systems Limited for re-programming, using an encrypted file which can be sent by email to the customer.

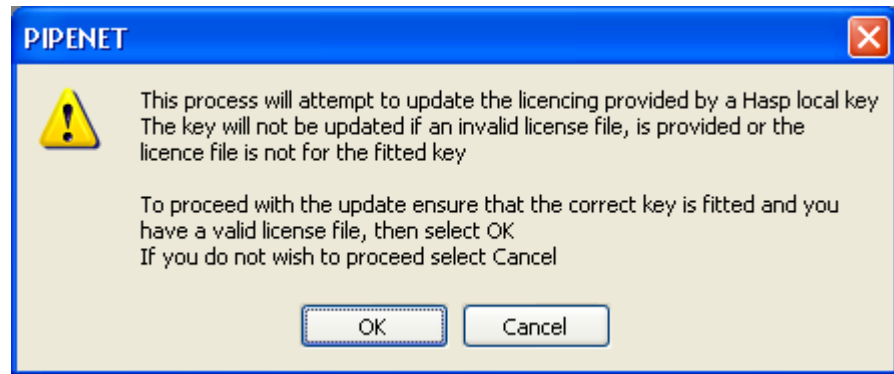
Expiry dates

If a module has been licensed with an expiry date, then any attempt to use the module beyond the expiry date will result in the key no longer being recognized as a valid key. If a key expires then Sunrise Systems should be contacted, requesting an update. If approved, a small encrypted text file will be sent to the customer by email.

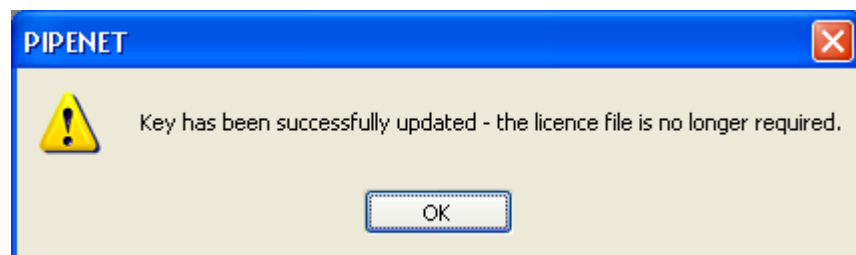
Updating a licence file

When you are in receipt of an encrypted licence file then proceed as follows to update the security key:

1. Make sure that you have started PIPENET and there is no network open
2. Select the menu option **Help | Update key** and the following message will be displayed:



3. Selecting ok will display a dialog for navigating to the encrypted licence file
4. As soon as you have selected the licence file a second dialog will be displayed, hopefully indicating the success of the operation:



5. If the operation fails then Sunrise Systems should be contacted for further assistance, which may necessitate the return of the key. This may occur with security keys that are several years old.

Note that the encrypted licence file is only valid for a specific key, as identified by the number in the bottom right-hand corner of the key's label. For parallel port keys this will be a four digit number, for USB keys a 5 digit number greater than 10000. Note also that a licence file can only be used once to update a key, if an attempt is made to update a security key more than once, the second and subsequent attempts will fail with an error

2.4 Security Key Problems

If you have got as far as installing the software then the most common problems encountered are those to do with licensing. If you can load the software but cannot open or create a problem because of an error message relating to a security key or licensing problem then proceed as follows.

Security access rights

It is important that the installation of PIPENET modules be performed with Administrator access rights and privileges. These rights and privileges are required for:

- Installation of the drivers for the security key
- Addition and/or updating of system files in the Windows System directory. (note

that PIPENET will never replace an existing file in this directory if it is more recent than the one shipped with PIPENET).

- Access to the Windows' System registry

If you did not have these rights when you installed the software then the key drivers will not have been correctly installed and thus the software cannot be run. Re-install the software having obtained the necessary rights and privileges. Contact your IT support group if you are unsure as to how to proceed.

Fitting the security key

A USB key must be fitted to an available USB port and a parallel key to port LPT1:. If you are using a parallel port key then the key may be fitted in series with security keys provided by other software manufacturers.

Licensing provided by a key

Each key is specifically programmed for the needs of each user. Depending on the licensing agreement, the key will be programmed:

- for one or more PIPENET modules,
- for a specific number of runs or unlimited number of runs
- for use before a programmed expiry date or for unlimited use.

If the key is not programmed for the module you are attempting to run, or the number of runs has expired, or the key has passed its expiry date then any error messages displayed will inform you of the specific error.

If the key is correctly fitted and the drivers have been correctly installed then the status of the key and the licences available can be checked via the [Help | Key Status](#) menu option. If the security key is correctly fitted then this will display the key details, including key number and customer name, in addition to licensing details.

Checking installation of security key drivers

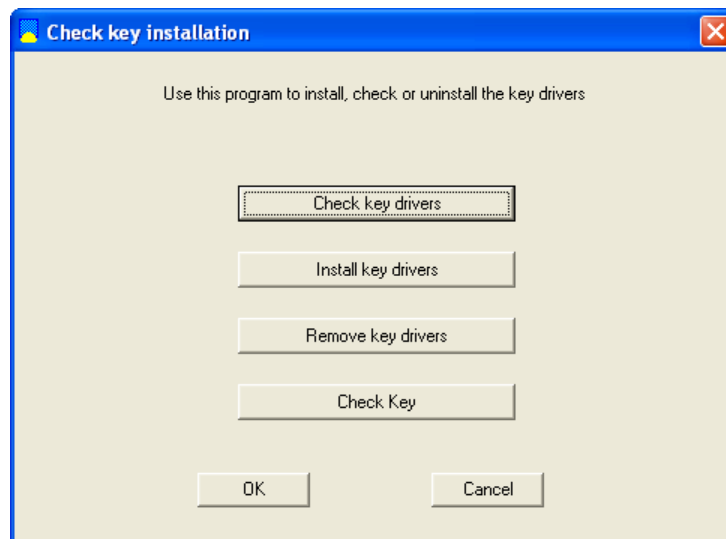
If you have followed all of the instructions above and you have a key licence for the module you are trying to run then you should have no further problems. However, if you are still having problems, please perform the following checks and email the results to Sunrise Systems. Be sure to include the key number. If you cannot run Key Status then the key number is also printed in the bottom right-hand corner of the key's label.

Trouble shooting

The most common reasons for failure are not having the correct key fitted or the key drivers were not installed correctly. If the key drivers were not installed then the most likely reason for the failure is that you did not have the necessary access rights during installation. The following summarizes the checks you should perform to locate the source of the problem:

1. Check that you are using a security key appropriate to the module you are trying to run.

2. Check that the key is correctly fitted to a USB or parallel port on the computer where you are attempting to run the PIPENET software.
3. You must have Windows' Administrator privileges to install the key drivers since changes are made to the System Registry. If you do not have these rights you will have to contact your IT department to set up your account details. It is recommended that you contact your IT department anyway to confirm any other rights and restrictions there may be.
4. Check that you have read and write access rights to the drive where the software will be installed (by default C:) and where the temporary files will reside (also by default drive C:). This is necessary since some organisations prohibit their users from accessing the local disk and selected network drives, other than for read access. Again if you do not have these rights then you will have to contact your IT department.
5. Please check (re-install if necessary) that the software and key drivers are correctly installed.
6. The installation of the key drivers can be checked by running the program KEYSETUP.EXE which can be found in the keydriver sub-directory of the PIPENET installation directory. Running this program produces the display:



Select the **Check key drivers** button to confirm installation. Entering this command will display the status of the key drivers, which if correctly installed will display the date of installation and other information on printer port, version number of driver etc.

7. If step 5. reports that the key driver is not installed then terminate any running PIPENET module and select the three buttons in the sequence **Remove key drivers**, **Install key drivers** and **Check key drivers**.
8. The last of the four buttons in the dialog, **Check key**, will attempt to read the key, displaying the internally stored key number and the customer name. The customer name may not be present on very old keys but the key number should always be readable if the key drivers are correctly installed.

The key check utility is stored on your hard drive during installation in the sub-directory

keydriver, the key check utility program is named KeySetup.exe.

A Tour of the User Interface

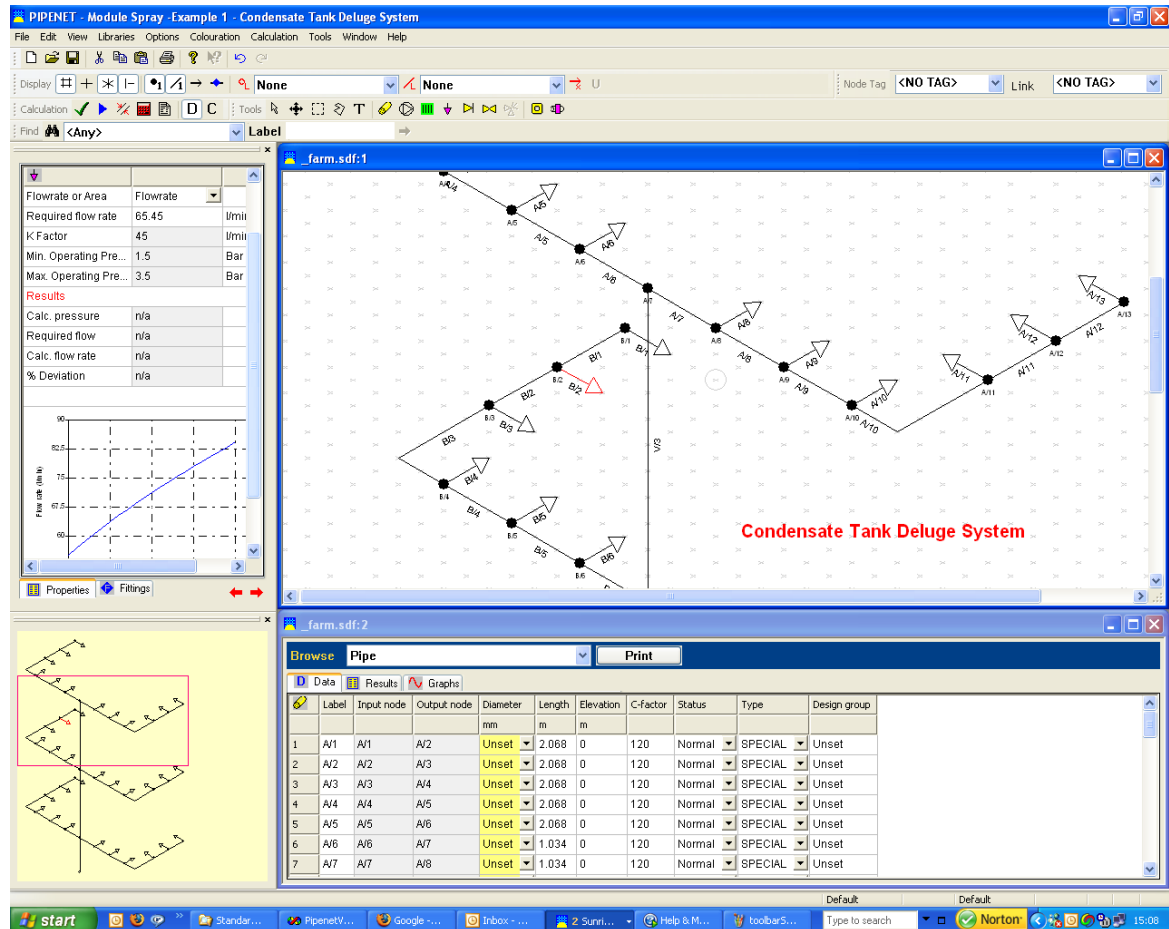
Part

3

3 A Tour of the User Interface

3.1 Overall view

The user interface is common to all PIPENET modules, although the toolbars and menus may vary slightly to reflect the different features available. When the program is started and a PIPENET data file is opened the screen will appear thus:



The four main windows depicted are as follows:

- Upper-left: a [Properties Window](#) used for displaying the attributes of the currently selected component. This window replaces the dialogs of earlier versions of the software since data can be entered and edited in this window.
- Lower-left: [Schematic Overview Window](#) showing an overall view of the schematic, with a rectangle showing the region covered by the main schematic. The rectangle may be dragged, with the main schematic window being automatically scrolled to reflect the changes. Conversely, scrolling the main window or changing the zoom factor will change the position and/or size of the rectangle in the Overview window.
- Upper-right: the [Schematic Window](#), essentially as in the previous versions of PIPENET, but allowing colour coding, multiple selections, an improved Area

Tool with copy, paste, delete, flip and invert operations, and an unlimited undo/redo facility.

- Lower-right: a [Tabular View](#) of the database is provided by a browse window, via which the user can display and edit component properties, and display results.

The Tabbed Properties and Schematic Overview windows can be closed by clicking the button at the top-right of each window. To re-display the windows select either the **View | Properties** or **View | Schematic Overview** option.

The Properties Window and the Schematic Overview can be moved to the right-hand side of the screen or "floated" by clicking and dragging the top of each window.

At the top of the screen is the usual assortment of menus and toolbars and at the bottom the status bar.

Menu styles

The menus are arranged differently to the previous PIPENET programs, although users of other Windows' programs may find them more familiar. To revert to the old style of menu select the menu option **Window | Use Pipenet menu style**. To revert to the new window style select **Window | Use Windows menu style**. See also the [Window Menu](#).

3.2 The Schematic Window

The Schematic window is the primary means of entering and viewing networks. It closely resembles the schematic window of earlier products but has a number of improvements.

When the window is first displayed it is presented with a light-grey background suitable for general viewing. The background colour may be changed to white or black. However, for coloured links and text, it will generally be found that a white background is unsuitable for viewing.

New elements are added by selecting the appropriate element tool from the tool palette and then placing and drawing the component using the mouse.

All labeled elements created via the schematic are automatically assigned a unique label. Labeled elements include nodes, link elements and attribute elements. Numeric labels are used (no tags) with each component type having its own set of unique labels.

The background colour and the font sizes used for labeling components can be changed using the Display Options dialog.

Schematic Underlay

A facility has been included whereby a graphic may be imported and displayed as a background to the main schematic. Display of this [underlay](#) is enabled and disabled via the [View menu](#). The underlay may be zoomed independently of the main network to achieve relative scaling and registration. Zooming the network, zooms the underlay by the same selected zoom factor. In normal use the procedure to use an underlay commences with a new network:

1. Import and display the underlay.
2. Select a suitable zoom size for the underlay.
3. Commence laying out the PIPENET components using the underlay as a guide.

3.3 Tabular view window

The [Tabular View](#) window provides details of attributes and results for network components. Data may be entered and edited via this view, columns re-arranged, and attributes/results printed. In fact the grid provides many of the properties (but no calculation facilities) associated with a spreadsheet product such as Windows Excel or Lotus 123. Copy/paste facilities are provided with unlimited undo/redo capability.

More than one Tabular View window may be open at a time, each showing the same or a different component type.

Tabular View windows are opened via the View menu.

3.4 Properties Window

This is a tabbed set of windows showing:

1. The Properties of the currently selected component, including where appropriate a graph, for example, a pump curve or a filter profile. Unlike earlier versions of PIPENET, properties can be entered and edited via this window. When a calculation has been performed this window will also contain the results for the component. Properties are displayed in three columns; the first is the name of the property, the second the current value of the property and the third the units (where appropriate). Properties are either entered explicitly as values or text strings or a value is selected from a drop-down list of acceptable values. If a property cell is greyed out it indicates that the value of the property cannot be edited.
2. A fittings window for assigning fittings to pipes.

The Tabbed properties window is normally displayed but if closed it can be re-opened via the View menu.

Pump operating point

Following a calculation, the **operating point** for each pump will be displayed on the pump curve in the properties window. The operating point is indicated by a small red triangle and will be at the point where the pressure and flow match the calculated results for the pump.

Nozzle operating point

Following a calculation, the **operating point** for each nozzle will be displayed on the nozzle characteristic curve in the properties window. The operating point is indicated by a small red triangle and will be at the point where the pressure and flow match the calculated results for the nozzle.

3.5 Component Properties

This window displays the properties (and possibly results) associated with the currently selected component. If no component is selected the window is blank.

When a component has been drawn on the schematic it will be assigned default parameters. These defaults may be edited in the **Properties** tab. Data entry fields in the properties window are either straightforward numeric or text entry, or of the selection style when there is a limited number of options available. Fields which cannot be edited will be "greyed" out.

The properties for all components are displayed in the same general manner: In the top left-hand corner of the grid is the symbol for the component, below this a number of rows, one for each attribute or result. Each row comprises of three columns:

- Column 1 - Name of attribute or result.
- Column 2 - Displays the value of the attribute or result. The value displayed here may be edited unless it has been "greyed" out.
- Column 3 - Units where appropriate.

Attribute	Value	Units
Label	1	
Input node	1	
Output node	2	
Diameter	Unset	mm
Length	0.50000	m
Elevation	0.0	m
C-factor	120.00000	
Status	Normal	
Type	Schedule 80	
Design group	Unset	
Results		
Input pressure	n/a	
Output pressure	n/a	
Friction loss	n/a	
Velocity	n/a	
Flow rate	n/a	

The above example shows the attributes for a pipe. All attributes can be edited: The length and elevation of the pipe are shown in feet and the diameter in inches. The status of the pipe is selectable from a drop-down list, and will be one of Normal, Blocked or Broken.

To edit a field click in the appropriate row in the second column and enter the new value,

or select from the available options. To accept the value enter Tab to move down to the next field or Enter on the keyboard. Values with a light-grey background cannot be edited. Changes made in the window can be undone and redone using the undo and/redo keys:



The left key is Undo and the right key Redo.

Scroll buttons

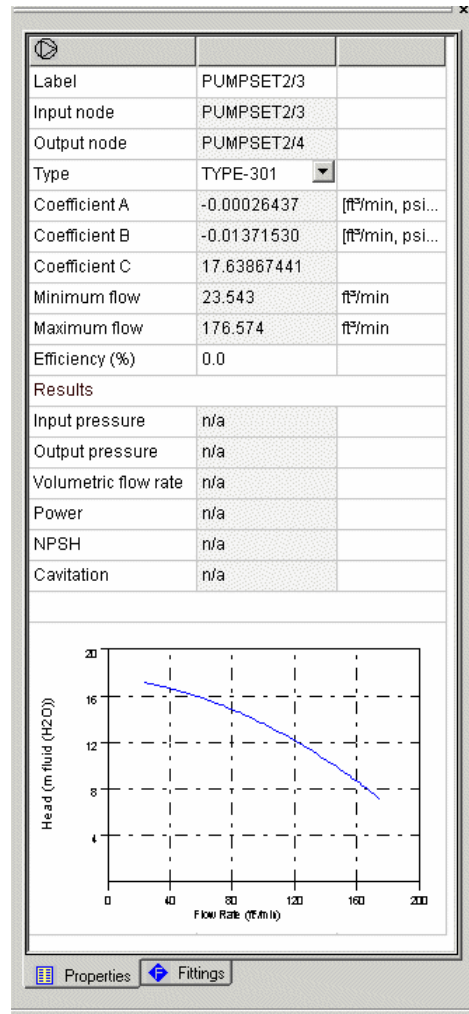
In the bottom right-hand corner of the window are two scroll buttons, these are used to move from the currently displayed component to the next or previous component of the same type.



For example, if a pipe is currently displayed, selecting the left button will move to the previous pipe (if there is one). Pressing the right button will move to the next component. Components are ordered in the sequence in which they were created.

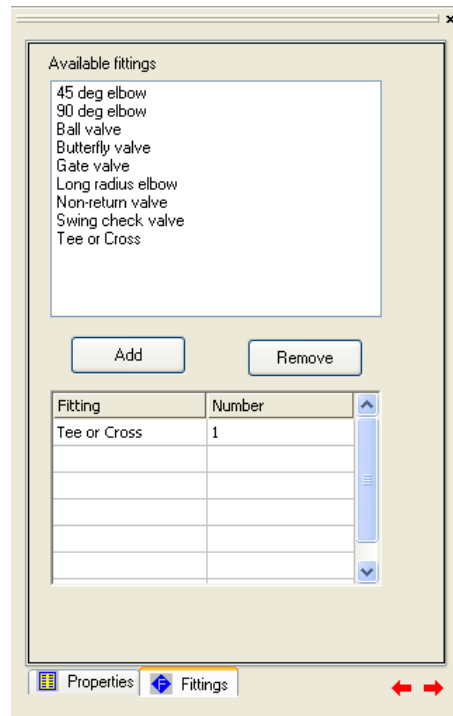
Display of accompanying graph

Some components may have an accompanying graph. If so, this is also displayed in the Properties window. For example, selecting a pump in the network will display the pump curve along with the pump parameters. Currently graphs are displayed for pumps and filters.



3.6 Fittings

If the currently selected component is a pipe then this window will display the fittings on the pipe.



The top window displays a list of available fittings, the bottom a list of the fittings currently selected on the pipe. The lower window displays for each fitting type; the fitting name, the K-factor and the number of fittings selected on the pipe.

Adding a fitting

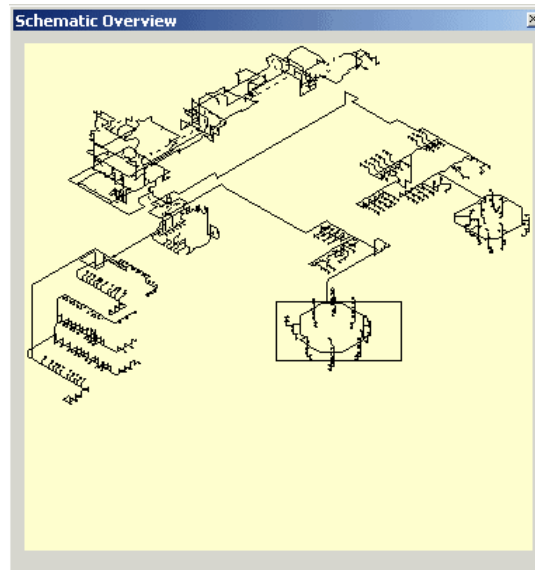
To add a fitting to a pipe select the desired fitting from the top window, and then the **Add** button. Each selection of the **Add** button adds one fitting of the selected type to the pipe.

Removing a fitting

To remove a fitting select the desired fitting type in the bottom window and then the **Remove** button. Each selection of the **Remove** button removes one fitting of the selected type.

3.7 The Schematic overview window

Provides an overall view of the schematic with a rectangle showing the part of the network currently displayed in the schematic window. The rectangle tracks changes in size and position of the main schematic window but the rectangle in the overview can also be dragged to effect a scroll of the main schematic window.



Display of the Overview window, which is not shown on starting the program, is done via the **View | Schematic Overview** menu option.

3.8 Menus

3.8.1 File Menu

The available options will be from the following. The displayed options will vary depending on whether or not a network is open.

New

Creates a new network. If licenses are available for more than one module type a prompt will be displayed requesting the PIPENET module be selected; Standard, Spray/Sprinkler or Transient.

Open

Opens an existing data file - files may be old format data files with a .DAT file extension or new style files with a .SDF file extension.

Close

Closes the network - if changes have been made to the network you will be prompted to save the file first before closing.

Save

Saves the current network and continue working

Save As

Saves the current file under a different file name.

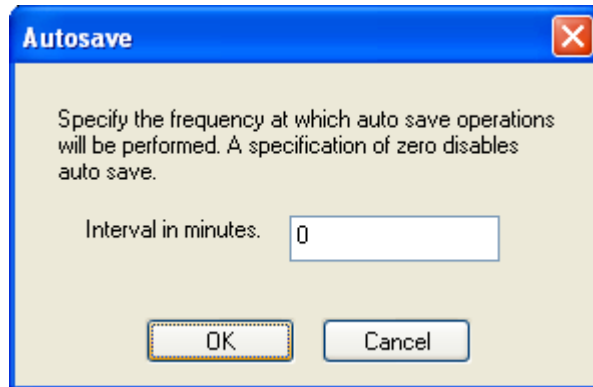
Export...

This will export the current network as an old style .DAT file together with any associated old-style library files, that is .PDF files, .UFL files, etc. When

selecting this option you should be aware that some graphical information will not be saved, for example colour schemes. When prompted for a file name enter the name of the .DAT file, any associated library files will be saved with the same name, but with a different file extension.

Autosave...

AutoSave is a feature that can be used to automatically save your edits and modifications periodically. Select this option to set the interval, in minutes, at which the network is to be automatically saved. The default is zero indicating that the autosave feature is disabled.



Open Library

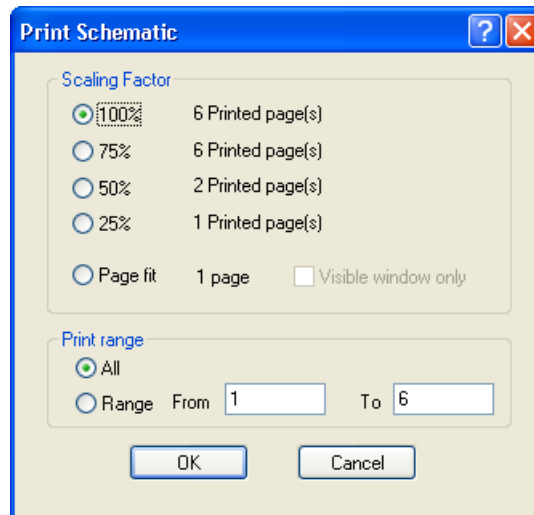
Opens an external [system library](#).

Import Library

Imports a library file - opens a system library or old format library file (e.g. .pmp pump library file) and import its definitions into the [local user library](#).

Print

Print the schematic or the grid - which is printed will depend on which of the two windows is selected. If in doubt, click in the desired window before selecting Print. If the grid is selected, the current grid page will be printed. If the schematic is selected then a dialog box is displayed via which the user can select the scaling and hence the number of pages required to print the schematic.



Print Preview

Previews appearance of printed output.

Print Setup

This option displays a standard Windows dialog from which the user can select a particular printer, landscape or portrait mode, etc.

Exit

Exits PIPENET.

Recently used file list

Provides a list of recently opened files.

3.8.2 Edit Menu

This menu provides a number of edit related functions:

Undo

Undoes last operation - this command will undo the last change made in the Schematic, Properties or Tabular View window. Following the word Undo is a brief description of the last operation performed.

Redo

Redoes last undone operation - as with the Undo command, following the word Redo is a brief description of the last undo operation.

Cut

Combination of a copy operation followed by a delete.

Copy

If the select tool is in operation this will copy the attributes of the selected component. If the [Area tool](#) is in use it will copy all components within the selection rectangle.

Paste

If the select tool is in operation this will paste the last copied attributes to the selected component as long as the components are of the same type, i.e. both pipes, both pumps, etc. If the Area tool is used it will paste all components copied by the last copy operation.

Paste in column

This option is only available in the [Tabular View](#). When a single cell in a column is selected, it will reproduce the contents of the cell in all cells in the same column.

Mirror

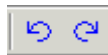
Used with the [Area Tool](#) to mirror the contents of the selected area left-right.

Invert

Used with the [Area Tool](#) to invert the contents of the selected area.

Undo/redo on the toolbar

Note the undo and redo operations are available via two shortcut buttons in the toolbar:



The left button is the Undo and the right button Redo.

3.8.3 View Menu

The view menu controls some aspects of the overall appearance of the various windows.

Toolbar

Displays or hides the main toolbar containing the file and edit related buttons - this is best left displayed at all times.

Status Bar

Displays or hides the Windows status bar - this is best left displayed at all times.

Palette

Displays or hides the [palette bar](#) which is used to select the tool for drawing within the schematic window - this is best left displayed at all times.

Schematic Window

Opens the schematic window if for some reason it has been closed - only one schematic window can be open.

Data Window

Opens a tabular view window - more than one tabular view window can be open at any one time.

Properties

Displays or hides the [Properties Window](#) which contains the tabbed set of windows providing information on the currently selected component and its fittings.

Schematic Overview

Displays or hides the [Schematic Overview Window](#).

Zoom

Used to zoom the network and/or the graphical underlay.

View graphical underlay

Selecting this option will display a graphical underlay which has been imported from a Windows extended metafile (.EMF), a Windows metafile (.WMF) or an AutoCAD .DXF file.

Import graphical underlay

Imports a graphical underlay from a Windows enhanced metafile, file extension .EMF. These files can be produced by many graphics programs and there are utilities available for converting from some CAD formats to extended metafiles.

The imported graphic is displayed near the centre of the window scaled to fit the current size of the window. The size of the imported graphic can be controlled by zoom (see above). Its position is currently not controllable. However using the [Area tool](#) to move the network, the registration between the underlay and the PIPENET network can be adjusted. The intensity of the displayed image, specified as a percentage, can be controlled via the underlay tool (The **U** button) on the [Options toolbar](#).

3.8.4 Libraries Menu

Before attempting to use libraries, the user should first read the chapter on [libraries](#).

The Libraries menu contains a number of module specific entries. Those for the Standard module include:

Schedules - Define or edit pipe schedules in the local user library.

Nozzles - Define or edit pipe spray nozzles in the local user library.

Pumps - coefficients unknown - Define or edit pumps with unknown coefficients in the local user library.

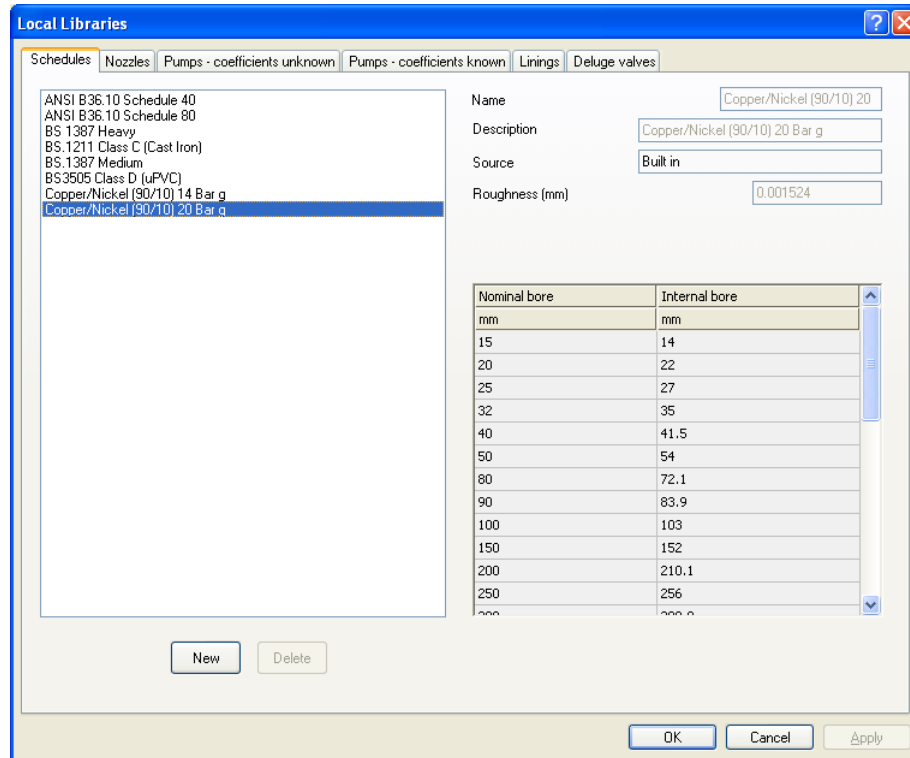
Pumps - coefficients known - Define or edit pumps with known coefficients in the local user library.

Linings - pipe linings - Define or edit pipe linings

Deluge Valves Define or edit deluge valves.

Selecting any of these options displays a tabbed dialog with the appropriate library edit selected. It is important to note that these edits are applied to the definitions in the [Local User Library](#).

To edit System libraries select the last option Edit system Libraries.



The final option in the Libraries menu opens a very similar window except that in this case edits are performed on a named System Library. It is important to re-emphasize the distinction between [Local User Libraries](#), where changes made only affect the current user, and [System Libraries](#), where changes may affect other users.

3.8.5 Options Menu

Options displays a tabbed dialog for viewing and setting the various options in use:

- [Title](#) - Title for the network, up to four lines of text may be specified.
- [Module Options](#) - Various modelling options.
- [Units](#) - Units to be used.
- [Pipe Types](#) - Pipe types.
- [Display options](#) - Schematic display options.
- [Calculation](#) - Calculation options, including tolerances.
- [Output tables](#) - Selection of which output results are to appear in the browser output.
- [Defaults](#) - Default information for pipes and nozzles (module dependent).

3.8.6 Colouration Menu

This menu contains three items which can be used to setup the colouring of nodes and/or pipes based on the value of one or more attributes. The options are described in detail in [Colour Schemes](#).

Simple rules

This option is used to define the colouring of links and or nodes based on the value of a single attribute or result, for example node elevation, pipe length, velocity of fluid through pipe, and so on.

Complex nodes

This option is used to define more complex colouring rules for nodes based on the values of more than one attribute or result.

Complex links

This option is used to define more complex colouring rules for pipes based on the values of more than one attribute or result.

3.8.7 Calculation Menu

This menu provides a number of calculation and output related commands:

Check

Check conditions for calculation - checks that specifications are complete and consistent, and if pipe elevations are in use, that node heights are consistent. This option opens a status window (if it is not already open) as described in [Status Checking](#).

Spec. for Calculation

Displays the Options dialog with the Calculation options tab displayed.

Input data

Runs validation in calculator, producing an output browser file. Validation does not consume a calculation for those users with metered licenses.

Pipe Sizing

Runs the pipe sizing phase of the calculation only, this phase will calculate the bore of any pipes with an undefined bore. Validation does not consume a calculation for those users with metered licenses.

Calculate

Runs a calculation, consuming one calculation for those users with metered keys.

Browse

Browse the output of the last calculation or the last request for input

NFPA Submittal Report

Hydraulic calculation details can be written to a file in tabular form in order to assist in the preparation of a fire protection submittal of the kind required by the NFPA. Following a successful calculation select this option to produce an [NFPA Submittal Report](#) that can be loaded into most word processors, including Word.

3.8.8 Tools Menu

The tools menu provides a number of useful tools:

Tag selected items

Used in colour coding to [tag](#), or mark, selected components as being significant in some way so that they can be [colour coded](#).

Remove tags

Marks all [tagged](#) components as untagged.

Make Path

Used in the construction of an [elevation profile](#) to construct a path through two or more selected nodes..

Export clipboard

Copies schematic to clipboard - from where it can be pasted into other applications.

Export HP-GL/2

Exports schematic as an HP-GL/2 file containing instructions for off-line plotting. HP-GL/2 is Hewlett-Packard's standardized Graphics Language supported by many CAD and graphics programs and peripherals. On selecting this option you will be prompted to supply an output file name.

Export DXF file

Exports schematic as an AutoCad™ DXF file, which can be used as input to many CAD programs. On selecting this option you will be prompted to supply an output file name.

Add multiple pipes

This tool provides a means of defining a run of pipes based on a table of elevations and distances. From this data, if supplied in a suitable form, the PIPENET module can automatically generate a sequence of pipes with the correct lengths and rises (elevation changes). For further information on this facility see the [Add multiple pipes section](#).

Autolayout

The [autolayout tool](#), which is available in the PIPENET spray/sprinkler module, can be used to automate the task of designing spray/sprinkler systems for vessels or for area protection.

3.8.9 Window Menu

In addition to the standard Windows window menu offering options to Tile or Cascade windows, this menu also offers an option to switch the appearance and ordering the menus between two styles: New style, which closely resembles other windows programs, and PIPENET style, which has a similar ordering to earlier versions of the program. However, there are some obvious differences. For example there is no view menu and most options are displayed on a tabbed dialog. To revert to the old style of menu select the menu option **Window | Use PIPENET menu style**. To revert to the new window style select **Window | Use Windows menu style**.

3.8.10 Help Menu

Help topics

Display help contents.

About PIPENET VISION

Details of PIPENET version numbers - note that unlike previous versions of PIPENET the calculators will have version numbers different from those displayed in the graphical user interface. With the first releases of the software the version numbers will be of the form:

Graphical user interface - version 1.nn

Spray Calculator - version 4.nn

Key Status

Status and information relating to Hasp security key and associated licenses. Selecting this option will display the current licenses available on the fitted key, the following is an example of the display:

Security Key Status

Number: 10025 Customer: Sunrise - Robert

Key type: Old style regular key Issued: 2005-8-3

Standard

Licenced Expires: 2006-8-3 Runs: Unlimited

Spray/Sprinkler

Licenced Expires: 2006-8-3 Runs: Unlimited

Transient

Licenced Expires: 2006-8-3 Runs: Unlimited

Note all dates are in ISO format YYYY-MM-DD.

OK Save

Selecting **OK** closes the window, whilst selecting **Save** will save the displayed details to a text file (the save feature is not implemented).

Update Key

This option is only available in the Help menu when no PIPENET file is open. Activating this option will initiate an [update to a local HASP security key](#) using a license file supplied by Sunrise Systems.

Sunrise on the Internet

This option will open a new page in your default web browser set to the Sunrise Systems web page.

3.9 Toolbars

3.9.1 Spray Toolbar



This toolbar contains, from left to right, the following tools

A group of five general tools

[Select](#) - Used to select an item on the schematic, for example to display properties of a component, to drag a node, to select an item for deletion.

[Pan and zoom](#) - pan and zoom

[Area tool](#) - Used to select a rectangular area for copy/paste operations.

[Polygon tool](#) - Similar to the Area tool except that it allows an irregular shaped area to be selected

[Text tool](#) - For creating text elements on the schematic.

Then there are seven tools for creating the different types of link components (i.e. components with an input and output node) available with the Spray module:

Pipe - note this button will be greyed out (not selectable) until at least one [pipe type has been defined](#). In the Spray/Sprinkler module it is mandatory that all pipes belong to a [pipe type](#).

Pump

Filter

Nozzle

Non-return valve

Deluge valve

Elastomeric valve

Overboard dump valve (see note below)

Finally, there are two component tools - components must be placed on a pipe:

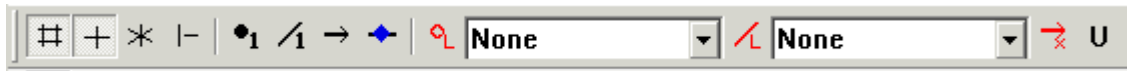
Orifice plate - only one per pipe

Equipment item

Note that if the **Remote nozzle specifications** or **Mass balance specifications** option has been selected in the [Calculation options dialog](#) then the overboard dump valve icon on the toolbar will be greyed out (non-selectable). This is because overboard dump valves can only be used with the **User-defined specifications** option.

3.9.2 Options Toolbar

The options toolbar provides a number of convenient shortcuts to schematic related options. For example, display of node labels, grid style, etc.



The toolbar consists of four button groups, from left to right the button options are:

Group 1 - grid related options:

- Display grid
- Select Orthogonal grid
- Select Isometric grid
- Snap to Grid

Group 2 - label options:

- Display node labels
- Display component labels
- Display component direction, arrow is drawn pointing from the pipe input towards the pipe output
- Display pipe fittings present symbol

Group 3 - results related options:

- Display node colouring legend button - if this is selected a node colour coding legend is displayed in the schematic window.
- A drop-down box from which the parameter to be displayed on nodes can be selected. For example, elevation and pressure.
- Display pipe/duct colouring legend button - if this is selected a link colour coding legend is displayed in the schematic window.
- A drop-down box from which can be selected the parameter to be displayed on pipes, for example flow rate, velocity.
- Following a calculation, arrows indicate flow direction, which may be in opposite direction to the component direction.

Group 4 - Underlay control:

A single button is provided. Selecting the button will display a dialog where

the intensity of the underlay can be specified as a percentage.

When an option is selected the button is shown as depressed, as illustrated for the Select Orthogonal grid option (second button in the above diagram).

3.9.3 Calculation Toolbar

This toolbar contains a few buttons related to a calculation and the calculation output.



Check

Checks conditions for calculation - checks that specifications are complete and consistent, and if pipe elevations are in use, that node heights are consistent. This option opens a status window (if it is not already open) as described in [Status Checking](#).

Input data

Runs validation in calculator, producing an output browser file. Validation does not consume a run for users with metered keys.

Pipe sizing

Runs the pipe sizing, or [design phase](#), of the calculation only. Pipe bores will be calculated for those pipes with undefined sizes and returned to the user interface for display with the rest of the pipes' attributes. Pipe sizing does not consume a run for users with metered keys. Pipe sizing is only possible if there is at least one pipe type and no ducts present in the network.

Calculate

Runs a calculation, producing a browser output file that can be viewed in the supplied browser, Word or Write.

Browse

Browses the output of the last calculation or the last request for input.

Design phase

Places the front-end in [design phase](#), in this phase pipe sizing operations can be performed repeatedly.

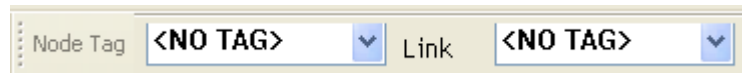
Calculation phase

Selection of this button places the calculator in [calculation mode](#), any pipe sizes calculated in the design phase are fixed for all future calculations.

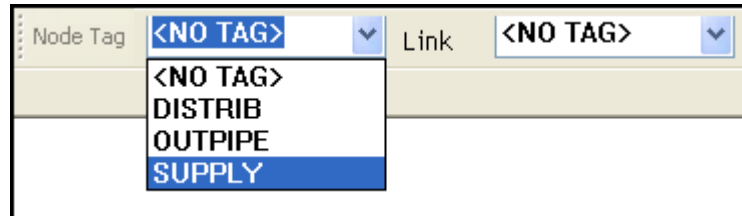
3.9.4 Tag Toolbar

This toolbar displays two drop-down lists of the label tags used in the current network for nodes and for links. It can also be used to select which tag is to be used as the default for all new nodes and links. To add a tag to a list simply enter the tag in the edit field

followed by return.



Selecting the arrow to the right of the text field will drop down a list of currently used tags - selecting a tag from this list sets that tag as the default.



If you have a tag selected in one of the drop-downs then whenever you create a new component, its automatically assigned numerical id will be prefixed with the specified tag and a '/'.

Unused tags cannot be explicitly deleted but each time a file is re-opened the tags in use are re-evaluated, and only those in use will be displayed.

3.9.5 Find toolbar

This toolbar is useful for finding components in the schematic window, particularly for large networks.



The toolbar consists of:

- a **Find** button
- a drop-down list of component types including the special type **<Any>** that can be used to find a component of any type with the specified label
- an edit box for entering the label of the component to find
- a **Next** button - this button is only enabled if the component type is **<Any>** and a find has been performed. Having found one component with a matching label, selecting this button will find the next component with the same label (assuming there is one).

If a component is not found then an information message will be displayed on the status bar at the bottom of the main window.

Example 1 - A Simple Three Pipe System

Part



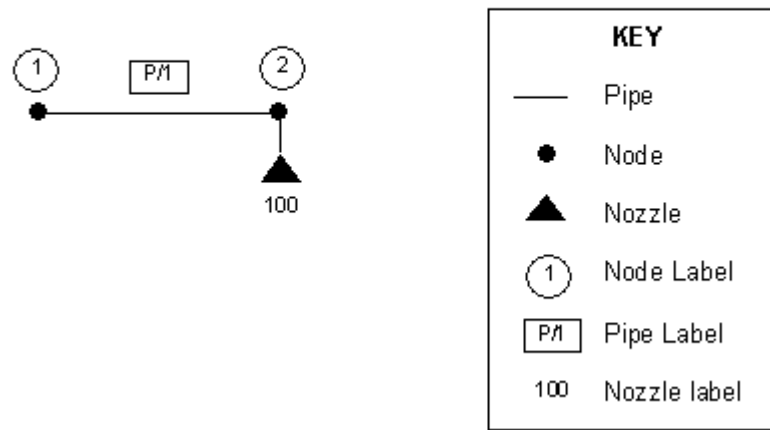
4

4 Example 1 - A Simple Three Pipe System

4.1 Network Representation

Schematic Diagrams

In general, networks consist of a number of components (Pipes, Pumps, Valves, Filters and Nozzles) all connected together. The points at which the components may be joined to other components are referred to as nodes. Consider, for example, the simple system shown in the figure below, which consists of a single pipe with a nozzle on one end. A liquid enters at the open end of the pipe and is discharged through the nozzle. The network can be represented schematically by the diagram shown below:



Other Network Data

As well as the topology of the network, PIPENET must be given data on the physical characteristics of the components in the network. For example, the lengths of the pipes and the discharge coefficients of nozzles in the network.

4.2 Labelling the diagram

When preparing a network for simulation, every component and every node must be given a [label](#) which identifies it uniquely. The production of a fully labeled schematic diagram is an essential part of any simulation.

4.3 Labels

Each component and each node in the network must be given a label which identifies it uniquely. Labels may either be tagged or untagged. Untagged labels are simply a number in the range 0-32767. Tagged labels consist of a 'tag' (a string of up to 8 characters) followed by a slash (/) and a number in the range 0-999. Tags can be used to make labels more meaningful, and to allow sections of large networks to be more easily identified. Up to 100 different tags can be used in one system. Tags must begin with a letter, and may contain only letters and numbers. For example, the following are all legal labels:

Untagged:

1
1273
9999

Tagged:

JETTY6/1
JETTY6/876
P/12

Notes:

1. Untagged labels greater than 999 and tagged labels may not be used in the same network.
2. Tags not followed by a slash and a number are not valid labels. For example, XYZ is a valid tag but not a valid label when used alone.
3. Tags are case insensitive, that is, RING is taken to be the same as Ring and ring.

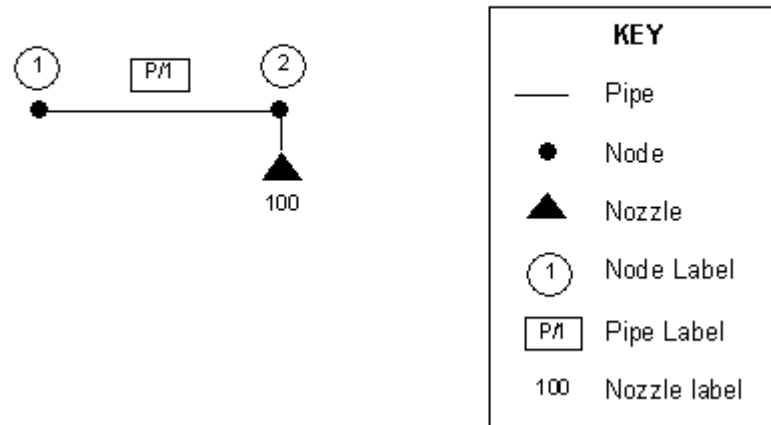
4.4 Inlets and Outlets

When drawing a schematic diagram of a network almost all components, for example a pipe, pump, valve or filter component should have two nodes-one at each end. One of these nodes is designated the component's input node and the other is designated its output node, collectively input and output nodes may be referred to **I/O nodes**. Note that fluid does not necessarily flow from the input node to the output node. PIPENET uses the convention that flow from input node to output node is referred to as positive, and flow from output to input is referred to as negative. In displayed results a negative value for flow indicates flow from the output towards the input.

Some components have only an input node, and although the component may appear to be drawn between two nodes, the output node is not visible and therefore it can have no other component connected to it. For example, a spray nozzle does not have an output node because the output from a nozzle cannot be attached to any other component- all nozzles are assumed to discharge to atmosphere.

4.5 Network Topology

The topology of the network is defined by specifying the input and output node of every component in the network. Thus we can define the topology of a simple network such as:

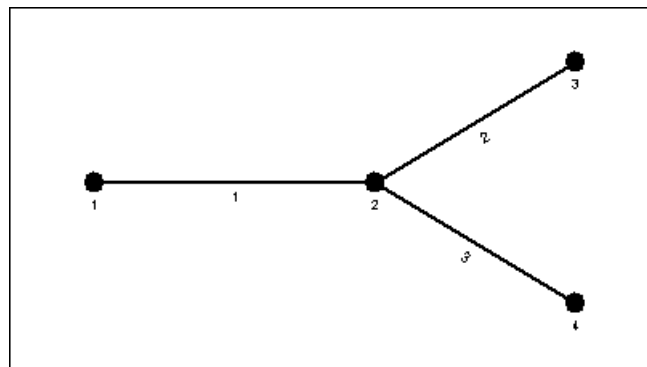


as having:

- Pipe P1 has input node 1 and output node 2.
- Nozzle 100 has input node 2.

4.6 Simple three pipe system

The Network under consideration is shown in the figure below.



Water is flowing into pipe 1 at node 1, and out of pipes 2 and 3 at nodes 3 and 4. The two outlets will be required to have a flow rate of 150 litres per minute, and a pressure of 1 bar G. In practice only one outlet has its pressure specified, but the symmetry of the network ensures that the other node is similar. PIPENET will calculate the pressures and flow rates throughout the network.

The details of the three pipes are as follows:

Pipe label	Input node	Output node	Diameter (mm)	Length (metres)	Elevation (metres)	C-factor	Fittings
1	1	2	32	1	0	120	None
2	2	3	20	1	0	120	None
3	2	4	20	1	0	120	None

Note that node and link labels will be assigned automatically as the network is entered. Your network will only agree with the above diagram if the pipe labeled in the diagram above is drawn first, then the pipe labeled 2 and finally the pipe labeled 3.

Note also that pipes have a notional direction from input to output, which does not necessarily correspond to the direction of flow. The elevations shown here are changes in elevation of the pipes as measured from the input to the output. Thus a positive value means that the elevation increases as we go from the input to the output and a negative value, that the elevation is decreasing. A value of zero indicates that there is no elevation change. Note that elevation can be specified as elevation changes on pipes, or as absolute elevations on nodes (see [Spray model options](#) for further details).

The three pipes will come from the built-in [schedule Copper/Nickel \(90/10\) 14 Bar G](#), with a design velocity of 5 m/s.

In this first example the details of entering a network, performing a calculation and viewing the results will be described in some detail with few references to other parts of the help. Subsequent examples will contain references to topics already covered in the example, only going into detail when discussing new features.

4.7 Calculation and Design Phases

The Design Phase

When designing a network it is generally required that the velocity of the fluid in each pipe does not exceed a given value (known as the design velocity of the pipe). The velocity of the fluid in a pipe depends on:

- The flow rate through the pipe.
- The diameter of the pipe.

It is thus important that all the pipes are correctly sized, so that the fluid velocity does not exceed the design velocity. This problem is addressed by the design phase of the simulation.

Given the required flow rates in and out of the network, PIPENET will find optimum diameters for each pipe in the network, so that the velocity of the fluid does not exceed the design velocity. The user must supply the required flow rate for all nozzles and for all but one of the I/O nodes in the network. PIPENET can then find the flow rates required throughout the network, and thus calculate optimal sizes for the pipes.

The following should be noted:

- If desired, the diameter of some (or all) pipes in the network can be set by the user. PIPENET will then size only those pipes whose diameter has not been set.
- If the user wants certain pipes in the network to be given the same diameter, then the Pipe Groupings facility should be used.
- When sizing a pipe (or group of pipes), PIPENET will choose the smallest pipe size which ensures that the design velocity is not exceeded by the fluid.
- The design phase assumes that all nozzles discharge at the minimum required rate. In most systems there will be some nozzles which actually discharge at a rate greater than the minimum requirement, and so flow rates and velocities in the system will rise. This may cause the velocity of the fluid to rise above the design velocity in some pipes in the system. These pipes will be identified during the calculation phase and a warning will be issued. To solve this problem the user should set the diameters of these pipes to slightly larger than the designed diameters.
- Calculated diameters are displayed in the [Properties window](#) and the [Tabular View](#) with a yellow background to the text.

More than one simulation may need to be performed in order to size all the pipes in a network.

The Calculation Phase

In the calculation phase all the diameters of the pipes are known -either set by the user or found by PIPENET during the design phase. PIPENET simulates the behaviour of the network under pressure and flow rate conditions set by the user. All nozzles in the network are assumed to discharge to atmospheric pressure. The user must supply flow rates and/or pressures at various parts of the network by making Calculation Phase Specifications (see next section). PIPENET will then calculate the pressures and flow rates throughout the network.

Typically the calculation phase is used to do one or more of the following:

- Determine what pressures are needed to produce the required flow rates.
- Select suitable pumps for the network.
- Check that all demands made on the network can be satisfied. A warning will be issued for any nozzle supplying at less than its required rate.
- Check that the fluid velocity in each pipe does not exceed the design velocity of the pipe. A warning will be issued for any pipe in which the fluid velocity exceeds the design velocity.

User Interface and the Design and Calculation phases

When a calculation is performed a Pipe Sizing operation will be performed if there are unset pipe sizes and the user has requested that the Design Phase be run. However, the user will usually perform a pipe sizing operation, possibly make some changes to the network, and then perform another pipe sizing operation, repeating these operations steps as required. Only when he/she is happy will they perform a calculation.

The controls relating to the two phases are on the Calculation toolbar:



and the sequence of operations will typically proceed as follows:

1. The two buttons D and C correspond to the Design and Calculation phases. If a design phase is required then the D button is selected by default and the pipe sizing button (the third button) is enabled.
2. The user enters the network and if he/she requires that PIPENET performs a Design phase then pipes may be entered with undefined bores.
3. When the user has entered the network they will select the pipe sizing button to size the undefined pipes. The calculated pipe sizes are returned to the user where they are displayed along with other pipe attributes, but note if another pipe sizing operation is performed the sizes may change if changes are made to the network.
4. Steps 2 and 3 are repeated as required.
5. When the user is happy with the network they select the C button (pipe sizing button is disabled) to fix the designed pipe sizes and then perform a calculation (fourth button).
6. The user can revert to the design phase by selecting the D button. However, the bores of pipes fixed in step 5 do not become unset.

Pipe grouping

The Grouping facility allows the user to specify sets of pipes in the network which are required to have the same diameter. For example, when designing a ring main, it might be desirable for the pipes in the primary main to share a common diameter. The facility is used by setting up groups of pipes, labeled 1, 2, 3 etc. When the facility is used PIPENET will ensure that all the pipes in the same group are given the same diameter.

The following should be noted:

- Pipe groups should be labeled consecutively, starting with '1'.
- All pipes in each group must be of the same pipe type.
- No more than one pipe in each may have its diameter set by the user (as setting the diameter of one pipe in the effectively fixes the diameter of all other pipes in the group).

A pipe is added to a group by specifying the design group number in the attributes for the pipe:

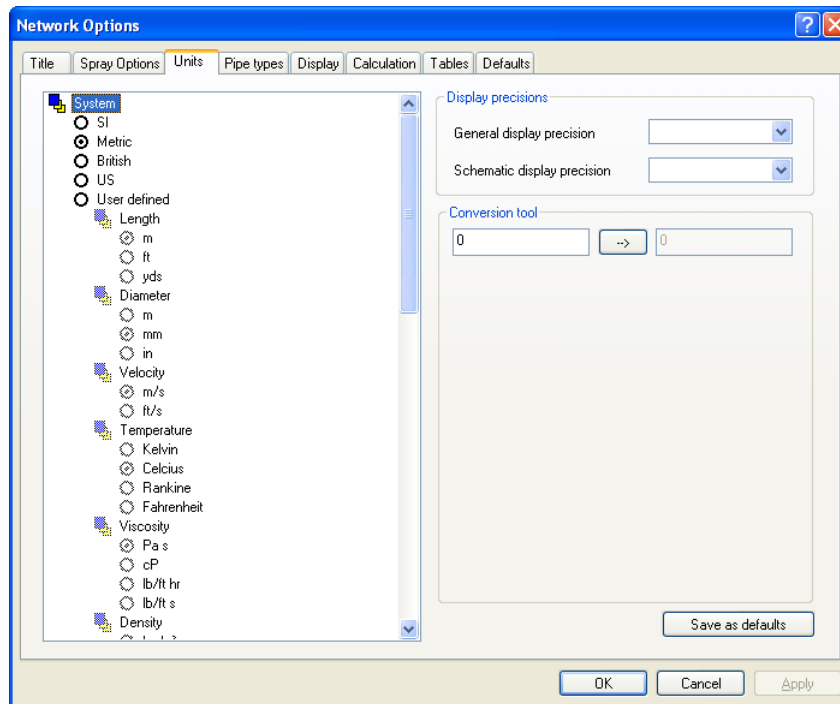
Label	1	
Input node	1	
Output node	2	
Diameter	Unset	mm
Length	0.5	m
Elevation	0	m
C-factor	120	
Status	Normal	
Type	Schedule 80	
Design group	1	
Results		
Input pressure	n/a	
Output pressure	n/a	
Friction loss	n/a	
Velocity	n/a	
Flow rate	n/a	

In this example, pipe 1 belongs to Design group 1. Note the default design group is unset, indicating that the pipe does not belong to a design group.

4.8 Specifying Units

By default the system assumes that the units are SI, we will change this to metric where pipe bores are specified in mm rather than the SI unit of metres and flow is in litres /min

Select the menu option **Options | Units** and then from the dialog that appears select the unit system as **Metric** from the system options in the upper part of the left-hand window. Information on the other features available in this dialog are discussed in [Specifying options - Unit options](#):



Select **OK** to close the dialog.

4.9 Creating a pipe type

In the Spray/Sprinkler module it is mandatory that all pipes belong to a [pipe type](#), which is simply a reference to a pipe schedule, the bores that can be used from the schedule, and usually the maximum velocity allowed in each usable bore. If no pipe type is defined then the pipe drawing tool on the toolbar is disabled and the status message **No Pipe Type** appears in the status bar at the bottom of the screen. This status will disappear when a pipe type is defined and the pipe button will be enabled:



We are going to use a built-in pipe schedule so all we need to do is to define the pipe type.

1. Select the menu option **Options | Pipe type**.
2. Select the **New** button to create a new pipe type.
3. Select the [schedule Copper/Nickel \(90/10\) 14 Bar G](#) from the drop-down list in the right-hand part of the dialog.
4. Use the **tab** key or the cursor to move to the [C-factor](#) field and enter the value as 120, the dialog should appear as follows:

Network Options

Title Spray Options Units **Pipe types** Display Calculation Tables Defaults

Name: Copper/Nickel (90/1)

Schedule: Copper/Nickel (90/1) 14 Bar g

Description: Copper/Nickel (90/1) 14 Bar g

C-factor: Unset

Lining (optional)
Material: None
Thickness (mm): Unset

Design information
 Velocities Pressure drops
Use in design Avoid in design

Nominal bore	Internal bore	Max. velocity
mm	mm	ft/sec
15	14	Unset
20	22	Unset
25	27	Unset
32	35	Unset
40	41.5	Unset
50	51	Unset

New Delete

OK Cancel Apply

We now need to set the maximum velocities. These are entered in the grid in the bottom right-hand corner. We could do this by typing the value in every cell in the third column. However, there is a much quicker way to do this. Enter the value 5.00 in the first cell of the third column followed by Return. Now select the cell with the mouse and whilst holding the button down, drag the mouse down the column so that all cells in the column are highlighted. On releasing the mouse button all cells will be set to the same value of 5.00. The dialog now appears as:

Network Options

Title Spray Options Units **Pipe types** Display Calculation Tables Defaults

Name: Copper/Nickel (90/1)

Schedule: Copper/Nickel (90/1) 14 Bar g

Description: Copper/Nickel (90/1) 14 Bar g

C-factor: 120

Lining (optional)
Material: None
Thickness (mm): Unset

Design information
 Velocities Pressure drops
Use in design Avoid in design

Nominal bore	Internal bore	Max. velocity
mm	mm	ft/sec
15	14	5.0
20	22	5.0
25	27	5.0
32	35	5.0
40	41.5	5.0
50	51	5.0

New Delete

OK Cancel Apply

Finally select **OK** to save the changes and close the dialog.


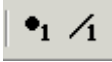
Other parts of this dialog box allow the user to line the pipes with some material, or to restrict the calculation to a subset of the full schedule.

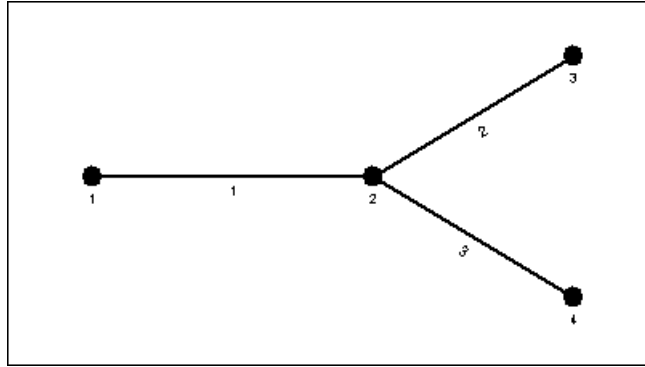
Note that if a pipe is defined as using a pipe type, then instead of explicitly entering the pipe diameter, the diameter is selected from a combo box listing the available sizes together with the **unset** value.

4.10 Entering the pipe data

With the schematic and the editable [Properties Window](#) this is a simple task. You can either draw the first pipe, enter the data for the first pipe, draw the second pipe, enter its data and so on, or draw all three pipes and then add the data for all three pipes. We will choose the latter method.

To draw the first point proceed as follows:

1. From the toolbar select the pipe drawing tool 
2. Place the cursor at the point where you want the input node to appear and left click.
3. If the selected point coincides with an existing node, that node becomes the input node; otherwise a new node is created and displayed at the selected point.
4. A line representing the link element is drawn and tracks mouse movements.
5. Place the cursor at the point where you want the output node to appear and left click.
6. As with the input point, if the selected output point coincides with an existing node then that node will become the output node. Otherwise a new node is created.
7. If, between defining the input node and the output node, you want to abort creation of the link, select the Escape key.
8. Now draw the second pipe in a similar manner ensuring that on the first click the cursor is over the output node of the first pipe. Finally draw the third pipe in the same manner as the second pipe. Note that if a pipe is coloured red this means the pipe is selected and its properties will appear in the [Properties Window](#).
9. Select the node and pipe label buttons  from the display toolbar if they are not already selected and the network should appear as:

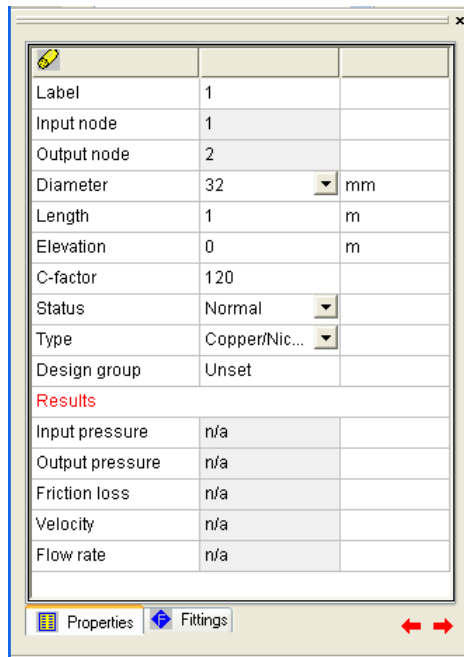


Now we can add the properties to the pipe, for example the bore and length of each pipe. Click on the first pipe and its current properties appear in the Properties Window. Observe that since we are using a pipe type some of the information is already filled in:

Label	1	
Input node	1	
Output node	2	
Diameter	Unset	mm
Length	Unset	m
Elevation	0	m
C-factor	120	
Status	Normal	
Type	Copper/Nic...	
Design group	Unset	
Results		
Input pressure	n/a	
Output pressure	n/a	
Friction loss	n/a	
Velocity	n/a	
Flow rate	n/a	

Properties Fittings ← →

Click in the value field for the pipe diameter and select the value 32 from the drop-down list of available pipe sizes, followed by Tab to move to the next field; the length. Enter a value of 1.0 for the length of the pipe followed by either Tab or Return to accept the value. You can now click on the second and third pipes entering the appropriate diameters and lengths. Alternatively, having entered the values for the first pipe use the red right arrow button in the bottom left-hand corner of the Properties Window to move to the next pipe (for long time users of PIPENET this is equivalent to the **Next** button). Clicking the left arrow button moves to the previous pipe if there is one. Note pipes are assumed to be in order of creation.



To view all three pipes we can use the [Tabular View](#). Select the menu option **View | Data** window to display the window:

Browse Pipe										
Print										
Data Results Graphs										
	Label	Input node	Output node	Diameter	Length	Elevation	C-factor	Status	Type	Design group
				mm	m	m				
1	1	1	2	32	1	0	120	Normal	Copper/Nickel (90/10) 14 Bar g	Unset
2	2	2	3	20	1	0	120	Normal	Copper/Nickel (90/10) 14 Bar g	Unset
3	3	2	4	20	1	0	120	Normal	Copper/Nickel (90/10) 14 Bar g	Unset

Data can be edited in this window simply by clicking in the appropriate cell. Entering a new value or selecting from a list of valid options, and pressing either Tab to move to the next field or Return.

Both the Schematic Window and the Tabular View can be displayed simultaneously by selecting the menu option **Windows | Tile horizontally** or **Windows | Tile vertically**.

It is a good idea to save the network at this point by selecting the menu option **File | Save**.

4.11 Undefined or invalid components

When a pipe, or any link component, is first drawn in the schematic it is coloured blue to indicate that either the component has not had its attributes specified and/or it has an invalid combination of attributes, for example a zero length pipe. When one or more attributes are entered for the component its colour changes to black (or white if the background is black).

The colour of a component may revert to blue in either of the following situations:

1. If a check is performed using the check button on the calculation toolbar and the component is found to be invalid.
2. If the file is saved and re-opened, and on re-loading the component it is found to be invalid.

4.12 Specifications

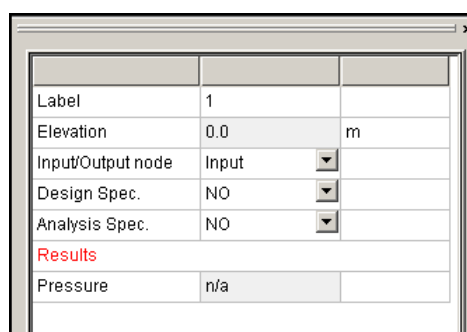
Before sending the network to the calculator, we need to specify conditions of pressure and flow rate at various nodes. Without these, the problem is not mathematically tractable. Also, we must designate certain nodes as input and output nodes. We achieve this simply by adding some attributes to some of the nodes.

Click on the first node, that is the node with label 1 on the extreme left of the schematic and we observe in the Properties Window that it has no specifications and is not designated as an input or output node. The same applies for nodes 3 and 4.

Since all the pipes have had their diameter set, the [design phase](#) of the calculation will not have much to do. However, it must still run and we must still provide appropriate specifications for it. The [rules](#) for design phase specifications say we must supply the flow rates at all but one of the I/O nodes. We will set nodes 3 and 4 to have design phase flow rates of 50 lit/min.

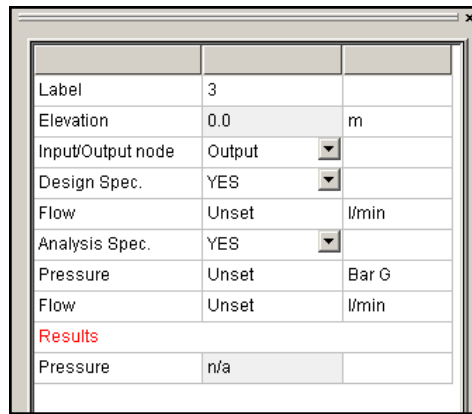
The rules for [calculation phase](#) specifications say that there must be as many specifications as there are I/O nodes, and at least one of them must be a pressure specification. In this case then we need three specifications, and at least one of them must be a pressure. We will specify a flow rate of 50 lit/min on each on the two outlets, and a pressure of 1 bar G on node 4.

Select the first node and in the properties window, click on the drop down in the Input/Output node, change the selection from **No** to **Input** and then hit the Return key. The properties window should appear as:

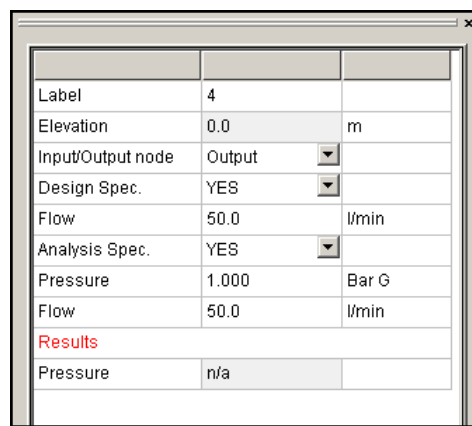


Label	1	
Elevation	0.0	m
Input/Output node	Input	
Design Spec.	NO	
Analysis Spec.	NO	
Results		
Pressure	n/a	

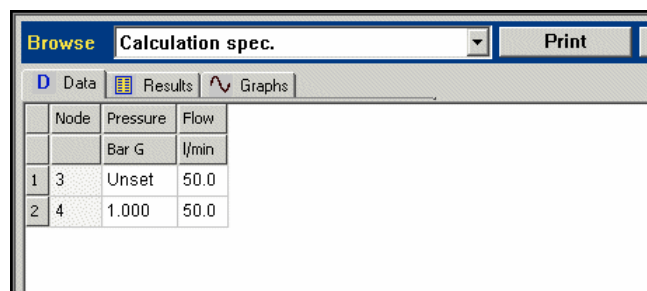
Now select node 3, then in the properties window change the input/output node status to **Output** and the Design and Analysis fields both from **NO** to **YES**. The dialog should appear as:



Now enter the values for node 4 and the Properties Window should appear as:



If you have the Tabular View window open you can select Design or analysis specifications from the drop-down list of components and inspect all specifications:



4.13 Calculating

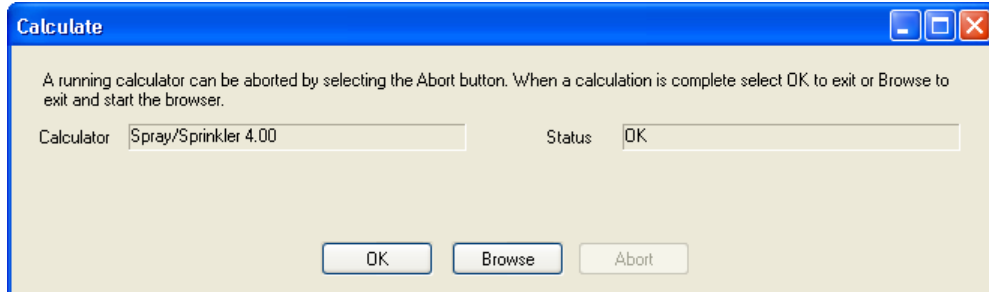
Having created and specified the network, we are now in a position to calculate its pressures and flow rates.

Select the menu option **Calc | Calculation**, or click on the Calculate toolbar button .

If you want to check that the data is correct without performing a calculation then (useful for those users with limited run licenses) you can use the **Calc | Check** menu option

instead, or click on the Check toolbar button .This option will check that the input data is valid without performing a calculation.

For all but very large networks (200 pipes or more) the calculation will complete almost immediately. Whilst the calculator is active the following dialog appears:



On completion of the calculation the dialog will display the completion status. Select the **OK** button to close the dialog or the **Browse** button to close the dialog and display the output data. Selecting **OK** simply closes the dialog, the browser can subsequently be started from the calculation toolbar.

4.14 Inspecting the results

Selection of the **Browse** button on completion of a calculation, or selecting the **Browse** option from the calculation toolbar, displays the following dialog:

Allowing inspection of the results using the PIPENET supplied browser, Write or Word; we shall use the PIPENET browser.

The calculation results are held in a temporary file but may be saved to a more permanent file by selecting the menu option **File | Save As**.

The report file is suitable for printing with most printers with the page orientation set to landscape. The scroll bars can be used to move around the report. At the top and bottom of the vertical scroll bars, page icons can be found. These page icons will allow users to move from one page to the next. On the left side of the horizontal scroll bar, an annotation appears indicating the current page number.

Using the browser text may be searched for using the Search menu option provided in the browser window.

When you have finished viewing the output, exit the Output Browser by selecting **File | Exit**.

Using the Tabular View to view results

The output produced by the calculation phase is more suited to printing than on-line viewing. Results can be viewed in the Tabular view by selecting the Results tab for the appropriate component type. Selecting the Results tab for pipes results in:

Browse		Pipe		Print		Add to report		
D Data		Results		Graphs				
	Label	Input node	Output node	Input pressure	Output pressure	Friction loss	Velocity	Flow rate
				Bar G	Bar G	Bar/m	m/sec	l/min
1	1	2	1	0.965	0.952	0.01305	1.73	99.9
2	2	3	2	1.000	0.965	0.03474	2.19	50.0
3	3	4	2	1.000	0.965	0.03474	2.19	50.0

4.15 Saving and loading the network

The network can be saved at any time in a data file from the PIPENET main window, as follows:

1. Select the **File | Save as** menu option,
2. The default file type is .SDF and is the one required. Enter the file name as SPR.SDF,
3. Click on the **OK** push button or press Return.
4. After saving a data file, it is safe to close PIPENET. If an attempt to close PIPENET is made at any other time, the user is warned of unsaved work and asked to confirm whether they want to save the changes, or not, or whether they want to return to PIPENET.

The network can be loaded again very simply. When loading a data file, PIPENET will load any related library files automatically.

Open PIPENET Spray/Sprinkler

Select the **File | Open** menu option, or the equivalent button. When the file open dialog is displayed, enter the desired file name or double click on SPR.SDF. Note the file type can be changed to .DAT to enable reading of old style data files.

Example 2 - A Small Sprinkler System

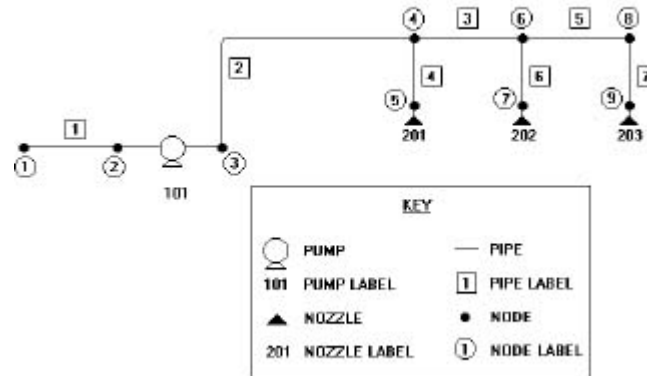
Part

5

5 Example 2 - A Small Sprinkler System

5.1 A small Sprinkler System

The second example consists of a pump connected to three nozzles. It demonstrates the use of user-defined pipe schedules, nozzle types and pump types. The pipe bores are generated in the design phase and the calculation phase makes use of the [Remote Nozzle Specification](#) option for specifications. The example is available in complete form under the name exam2.dat in the examples directory. Schematically the network appears thus:



5.2 Specifying units

Units will be specified as metric as in the [previous example](#).

5.3 Creating a user defined pipe schedule

Having loaded the PIPENET Spray/Sprinkler module we will first input a user defined pipe schedule. As we discovered in the first example, the library is used to contain data on user-defined pipe schedules, lining types, nozzle types and deluge valve types. The file is created using the Library pull-down menu, and saved using the File pull-down menu.

The data for the user-defined pipe schedule is shown below.

Schedule Name : Schedule 80

Roughness : 0.4572 mm

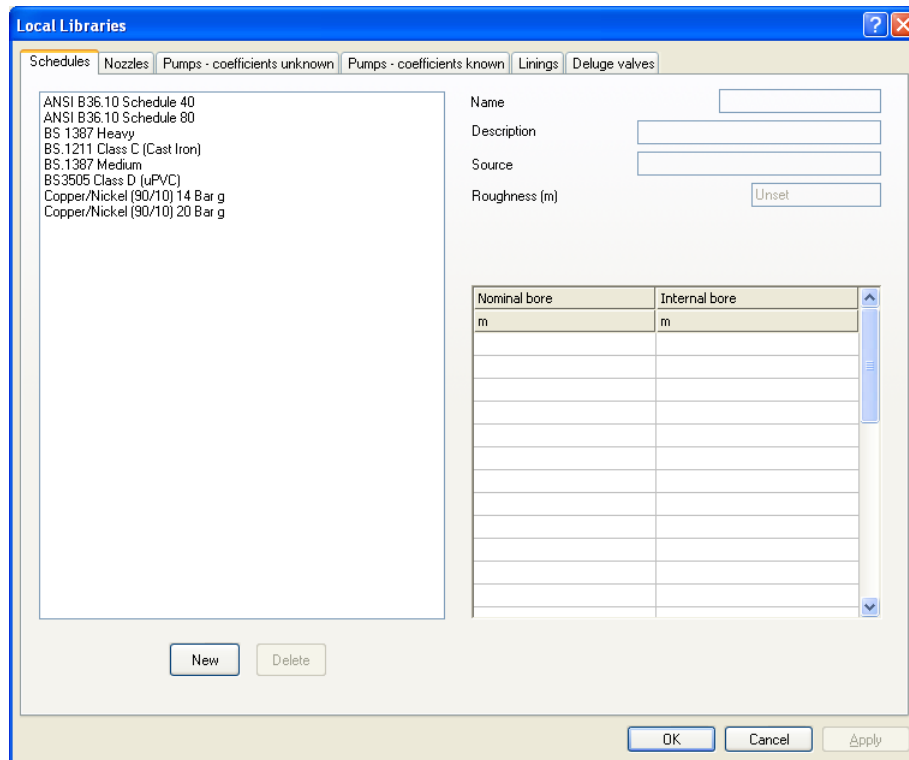
Bore Data :

Nominal Bore (mm)	Internal Diameter (mm)
15	unset
20	unset
25	24.003
32	unset
40	37.973

Nominal Bore (mm)	Internal Diameter (mm)
50	48.9966
65	unset
80	73.9902
90	unset
100	97.0026
125	unset
150	145.999
200	194.005
250	243.23
300	289.001
350	318.008
400	364.007
450	410.007
500	456.006
600	548.005
700	unset
750	unset
800	unset
850	unset
900	unset

Entering the Pipe Schedule

Select the menu option **Libraries | Schedules**; a tabbed dialog set appears with schedules displayed:



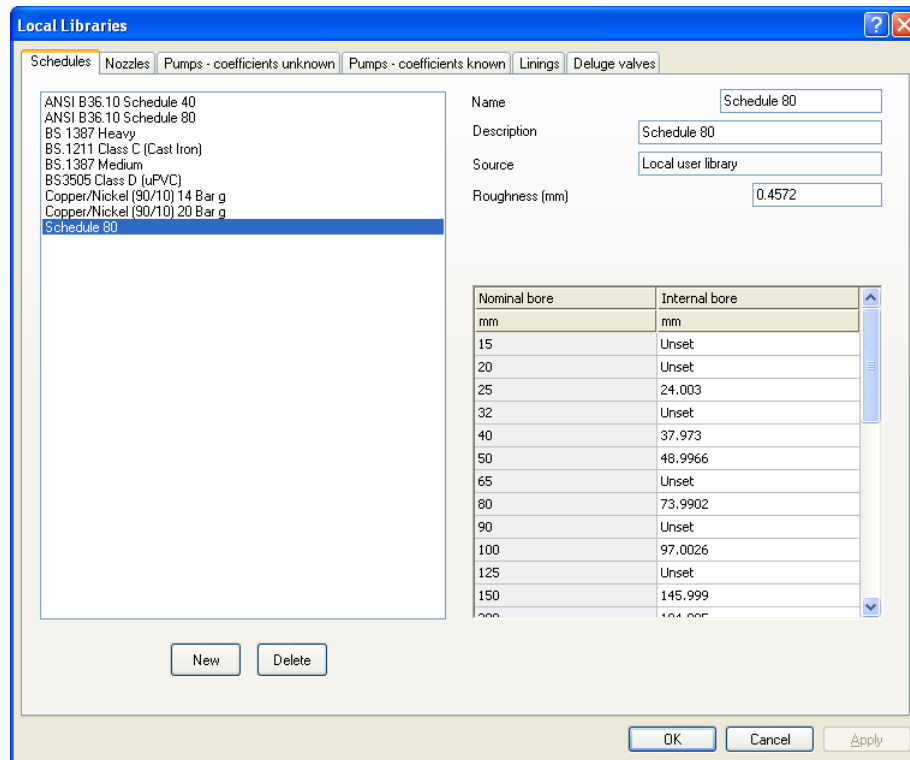
In the left-hand window is a list of all currently defined schedules. Selecting an item in this list displays the schedules properties on the right-hand side of the dialog. Data for eight pipe schedules are built into the PIPENET Spray/Sprinkler module. A non-built-in schedule in the list can be edited by simply selecting it in the left-hand window and editing the various attributes that appear in the right-hand side. Note that built-in schedules cannot be edited.

To add data for a schedule select the **New** button. Provide:

- The schedule name, Schedule 80 (this is the name that will appear in the left-hand window when the data entry is complete).
- An optional description (if not specified the name will be the same as the schedule name).
- A roughness value of 0.4572 mm.

The **tab** key can be used to move from one field to the next. Note that in selecting **New** the data grid in the bottom right-hand side lists all internal bores as **unset**. This simply means that no value has been supplied for the corresponding nominal bore.

Now place the cursor in the internal bore field corresponding to a nominal bore of 25.00 mm and enter the value 24.003 (from table above). Selecting tab moves to the next field which we can skip (leaving the value as unset. Selecting tab again moves to the next field where we enter the value 37.973. Continue in this way until the last diameter has been entered. Select the **Apply** button - changes are accepted and the new schedule appears in the list in the left-hand window.



If you are happy with the displayed results you can either select **OK** to quit the dialogs or, since we going to define other library items, simply select another tab, specifically the Nozzles tab since we are going to define a library nozzle in the next section,

5.4 Creating a nozzle type

The data for our nozzle is as follows:

- Nozzle Descriptor: Example nozzle.
- K-factor: 75 (flow rate units: lit/min, pressure units: bar G)
- Minimum pressure: 1.3 bar G.
- Maximum pressure: 9.5 bar G.
- Spray divergence angle - this is used in [autolayout and nozzle generation](#) and is not used in this example.

Nozzles are entered in a similar way to schedules. A dialog is displayed in the left-hand side with a list of available nozzles and the data is entered in the right-hand side. This is the same for all library components except pumps as we shall see in the next section.

Select **New** to add a new nozzle, provide the data above and select **Apply** to accept the changes. The dialog should then appear as:

The screenshot shows the 'Local Libraries' dialog box with the following details:

- Tab:** Nozzles
- List:** Example Nozzle
- Name:** Example Nozzle
- Description:** Example Nozzle
- Source:** Local user library
- K-factor (l/min, Bar^{1/2}):** 75
- Minimum pressure (Bar G):** 1.3
- Maximum pressure (Bar G):** 9.5
- Spray divergence angle - optional (deg):** 0
- Buttons:** New, Delete, OK, Cancel, Apply

5.5 Creating a pump library

For this example, a pump is also needed. This data is entered using the menu option **Libraries | Pumps - Coefficients Unknown** (or if you already have the library dialogs selected by selecting the Pumps - Coefficients Unknown tab)

Pumps in PIPENET are modelled with a quadratic performance curve and the required data are the three coefficients A, B and C (refer to [Modelling - Pumps for details](#)). If these coefficients are known for the pump then they may be specified directly using the Coefficient Known tab instead. However, these coefficients are not usually known, and the manufacturer generally supplies, with each pump, a graph of the pressure increase against throughput. In this case coordinates from the graph can be entered and the Pump/Fan Pre-Processor will find the coefficients by regression analysis.

The points to be entered are:

Flow rate l/min	Pressure Bar (H ₂ O)
0	3.95
115	3.75
395	3.15
575	2.75
860	2.05
1100	1.5

1450	0.6
------	-----

To specify the pump details proceed as follows:

1. Select the **New** button.
2. Select the units for flow (lit/min) and pressure (bar water) from the drop-down lists.
3. Provide the name for the pump (this is the name that will appear in the drop-down box at the top right-hand on the dialog when data entry is complete).
4. Provide an optional description.
5. Provide a minimum flow rate of 114.6 litres/minute and a maximum flow rate of 1450 litres/minute.
6. Now place the cursor in the first cell in the data entry grid in the lower left-hand side of the dialog and enter the first flow rate value of 0.0.
7. Tab to the next field and enter the corresponding pressure of 3.95.
8. Continue in this way entering the remaining data pairs, using tab to move from one cell to the next.
9. When all the data pairs have been entered, select the **Apply** button to calculate and display the pump curve.

The completed pump specification should appear as:

The screenshot shows the 'Local Libraries' dialog box with the 'Pumps - coefficients unknown' tab selected. The 'Example Pump' is defined with a quadratic curve type and a reference fluid density of 998.2343 kg/m³. The flow rate is in lit/min and head is in bar. The working range is set from 114.6 to 1450 lit/min. The curve coefficients are Constant: 3.968288, Linear: -0.002011912, Quadratic: -2.162733e-007, and Cubic: 0. The data table and graph show the resulting pump curve.

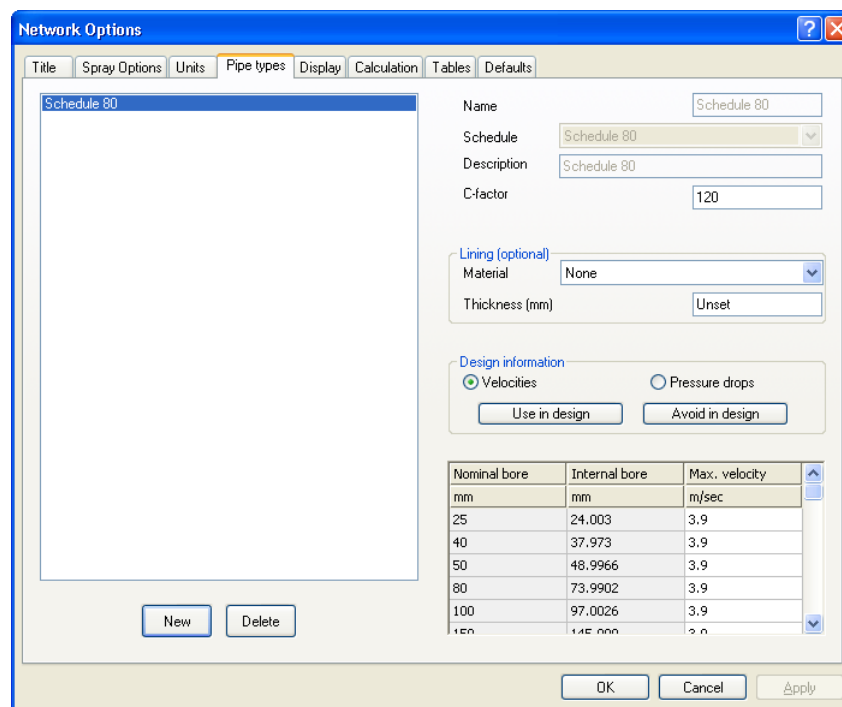
Flow rate	Head (input)	Head (curve)	Head (calculation)
l/min	Bar fluid (water)	Bar fluid (water)	Bar fluid (current)
0	3.95	3.968	3.968
115	3.75	3.734	3.734
395	3.15	3.14	3.14
575	2.75	2.74	2.74
860	2.05	2.078	2.078
1100	1.5	1.493	1.493
1450	0.6	0.596	0.596

5.6 Creating a pipe type

Before we can create any pipes we must first create a pipe type. This is done using the menu option **Options | Pipe types** (or if you already have the library dialogs selected by selecting the Pipe types tab). Proceed as follows:

1. Select **New**
2. Select the corresponding schedule from the schedule drop-down. This is the first editable box on the right-hand side of the dialog - the pipe type name becomes the schedule name
3. Enter a C-factor value of 120
4. Click in the first cell of the column labeled Max. Velocity in the data entry grid and enter the value 3.9 followed by a tab or Return. Select this cell and with the mouse button held down, drag the mouse down to select all cells in the maximum velocity column. On releasing the mouse button, all cells will have the value 3.9. Alternatively you could enter each value independently.
5. Select the **Apply** button to accept the pipe type.

The dialog should appear as:



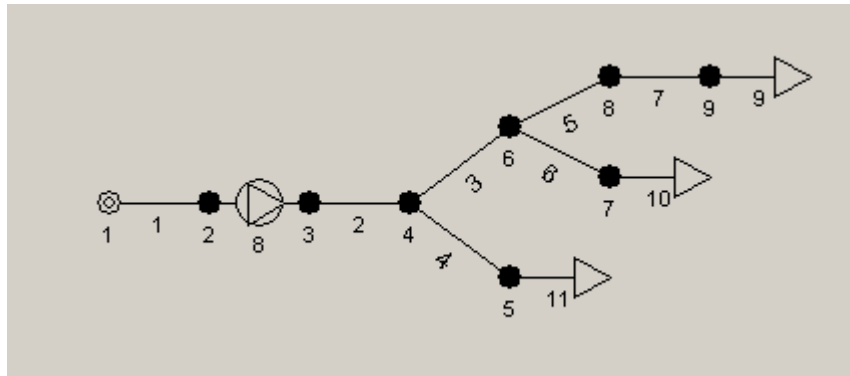
5.7 Entering the pipe data

Lay the pipes out in the schematic window as we did in the previous example, the pipes being as follows (note leave space between the output node of pipe 1 and the input node of pipe 2 for a pump):

Pipe	Input	Output	Bore	Length	Elevation	C-factor	Fittings

label	node	node	(mm)	(metres)	(metres)		
1	1	2	unset	0.5	0	120	None
2	3	4	unset	10	5	120	90° elbow
3	4	6	unset	5	0	120	None
4	4	5	unset	0.5	-0.5	120	None
5	6	8	unset	5	0	120	None
6	6	7	unset	0.5	-0.5	120	None
7	6	9	unset	0.5	-0.5	120	None

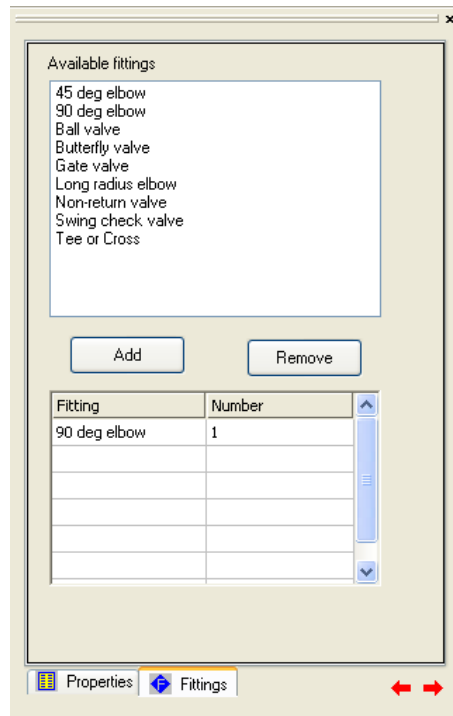
All pipe bores should be left unset, since the pipes are to be sized by PIPENET during the [Design phase](#). Note if the pipes are drawn in the order specified above you should have input and output designations as shown above. If you draw them in a different order then the designations may not agree, but as long as the network, when completed, is topologically the same as the following results will be the same..




To add the fitting to pipe 2:

1. Select pipe 2.
2. Select the Fittings tab in the properties window.
3. Select the 90° elbow in the upper window.
4. Select the + button to add the fitting.

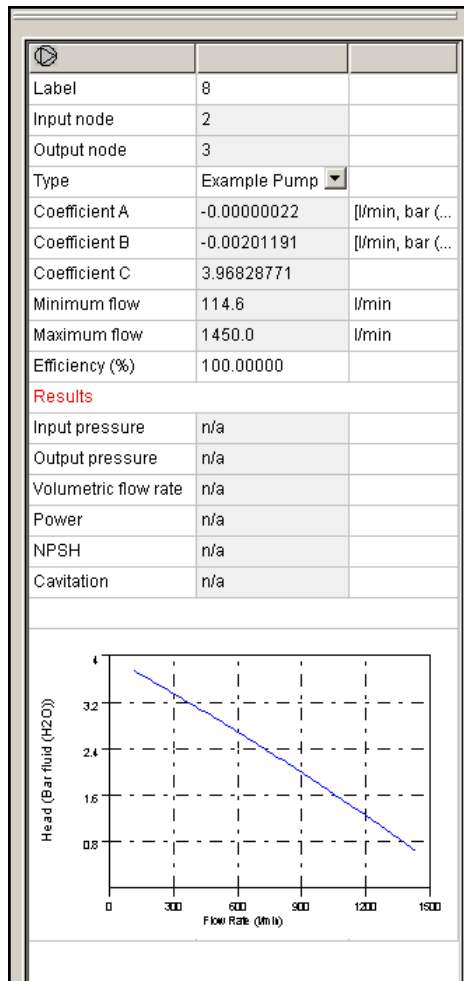
The fittings window now appears as:




5.8 Entering the pump data

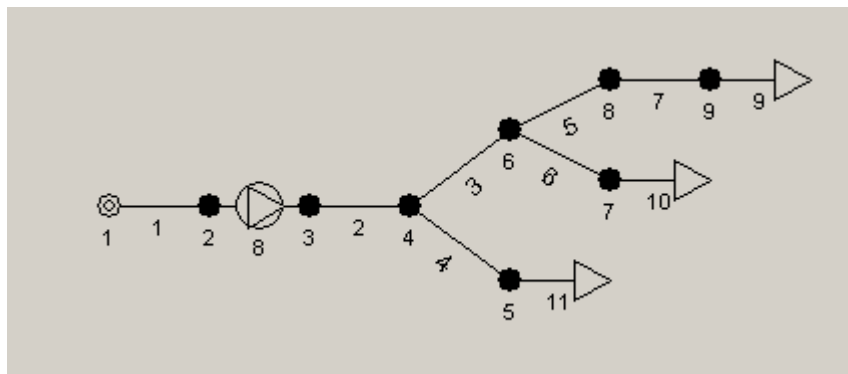
Select the pump tool  from the tool bar and draw a pump between the output node of pipe 1 and the input node of pipe 2. To achieve this with the pump tool selected click on the output node of pipe 1 and whilst holding the mouse button down move the cursor to the input node of pipe 2 and release the mouse button. The pump is the selected item and its properties appear in the properties window.

Select the pump type from the drop down list as **Example pump** followed by Return. The properties window changes to display the pump data from the library and the pump curve is reproduced in the bottom of the Properties window:



5.9 Entering the nozzle data

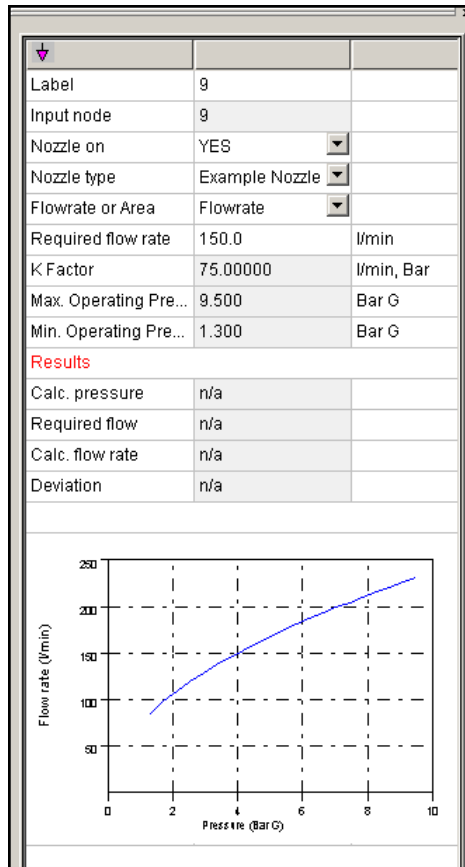
Select the nozzle tool  from the toolbar and draw three nozzles, connected to the nodes numbered 9, 7 and 5 (again node numbering could depend on the order in which the pipes were drawn), but the final schematic should appear topologically the same as:



For each nozzle in turn:

1. Select the nozzle so that its properties appear in the Properties window.

2. Select the nozzle type as **Example Nozzle** followed by Enter - attributes from the library item appear in the Properties window.
3. Set the required flow rate to 150 litres/minute - a characteristic curve appears in the lower part of the Properties window.



5.10 Entering specification data

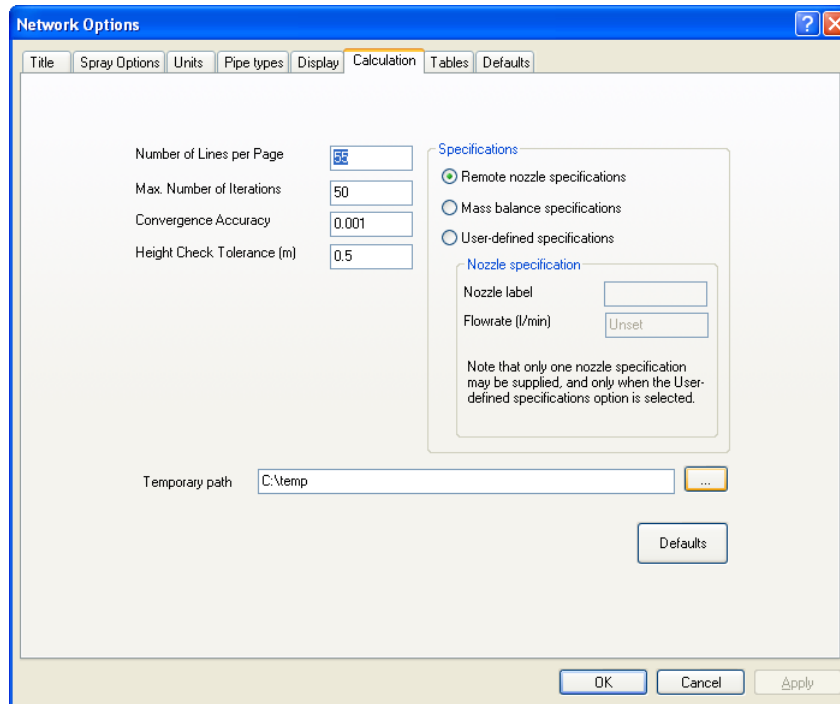
Node 1 is the only I/O node and must be specified as an input node. Select the node and change it to an input/output node, leaving its flow rate and pressure specifications unset. The number of design phase specifications must be one less than the number of I/O nodes. In this case then there are no design phase specifications.

Associated with each bore in the pipe type is a maximum velocity, and associated with each nozzle is a required flow rate. The design phase will, for each pipe, select the smallest bore from the schedule which can deliver the required flow rate without exceeding the maximum velocity.

As there is one I/O node, the user must supply one calculation phase specification. When the system is operating it is required that each nozzle is discharging at its required flow rate or higher. To ensure this condition is satisfied we use the Remote Nozzle Specification option. The calculator will find the hydraulically most remote nozzle, and specify that its flow rate is exactly the required flow rate. In general, this will mean increasing the total flow through the system, and that may, in turn, cause the fluid velocity

to rise above the design velocity. In such circumstances it will be necessary for the user to set the bores for some pipes.

The Remote Nozzle Specification option provides the system with one specification, and so this is all that is required. Select **Options | Calculation** and select the Remote Nozzle specifications:



5.11 Performing calculations

The specifications are now complete, and a calculation may be performed as in the first example and the results inspected via the Browser.

It is easy to perform several simulations on a network under different conditions, once the basic network data has been input. Specifications can easily be edited, and the different data files saved under different names. The same library private data file and pump library can be used for all simulations. It is recommended that the user experiments with this network. Try setting different sets of specifications altering the flow rate required through the nozzles, editing the lengths of pipes, etc.

Example 3 - Condensate Tank Deluge System

Part

6

6 Example 3 - Condensate Tank Deluge System

6.1 Condensate tank deluge system

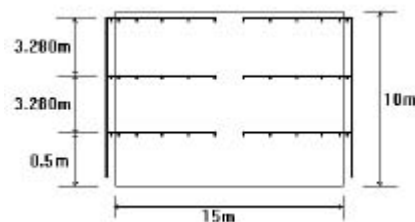
In this more realistic example we look at the fire protection system on a tank farm. Each tank is to be protected by a pair of external deluge systems, and we will investigate the design of the system. The example illustrates the following:

- Use of PIPENET to [size pipes](#) in a network.
- Use of the [Remote Nozzle Specification](#) option.
- Creation of a [pipe schedule](#) and [nozzle library](#) data.

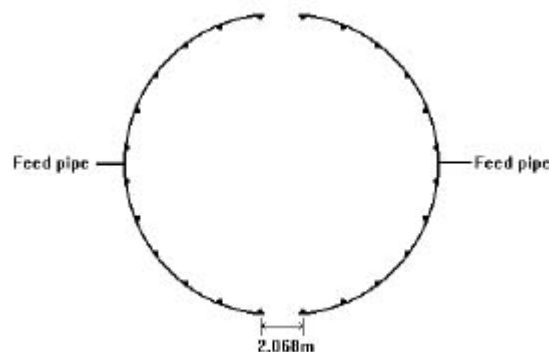
The files relating to this example are supplied with PIPENET, and are _farm.sdf, together with its associated library file in the examples sub-directory.

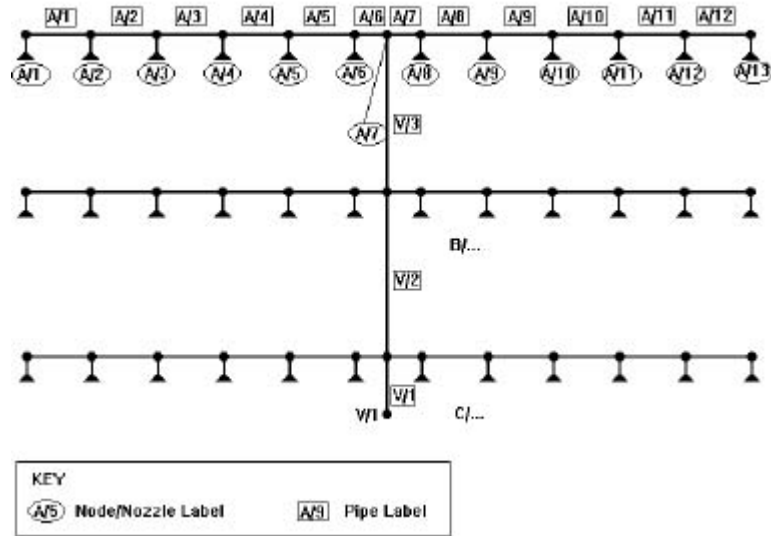
The Network

Each deluge system will consist of three horizontal semicircles, spaced at 3.28 m intervals vertically. Each semicircle will have 12 nozzles, 6 each side of the vertical feed pipe. Each tank has two such semicircular deluge systems. The total number of nozzles protecting a tank is therefore $(3 \times 2 \times 12) = 72$ nozzles. An elevation of a single tank, a plan of a single deluge ring and a labeled schematic are shown in the following figures.



The semicircles are held at a distance of 0.4 m from the side of the tank.





We will use PIPENET to:

- Find appropriate diameters for the pipes in the system.
- Determine the pressure and flow rate needed at the system inlet to ensure that all nozzles in the system discharge at or above the required rate.

6.2 Preliminary considerations

Before we can run the simulation, we have to work out some of the parameters that PIPENET will be using.

Lengths of pipes

$$\begin{aligned}
 \text{Nozzle spacing} &= \pi \times D / 24 \\
 &= \pi(15 + 0.8) / 24 \text{ m} \\
 &= 2.068 \text{ m}
 \end{aligned}$$

The pipes between nozzles will be treated as straight, of length 2.068 m

Required Nozzle discharge rate

$$\begin{aligned}
 \text{Surface area of tank} &= \pi \times D \times H \\
 &= \pi \times 15 \times 10 \text{ m} \\
 &= 471.24 \text{ m}^2
 \end{aligned}$$

Required flow rate through each nozzle

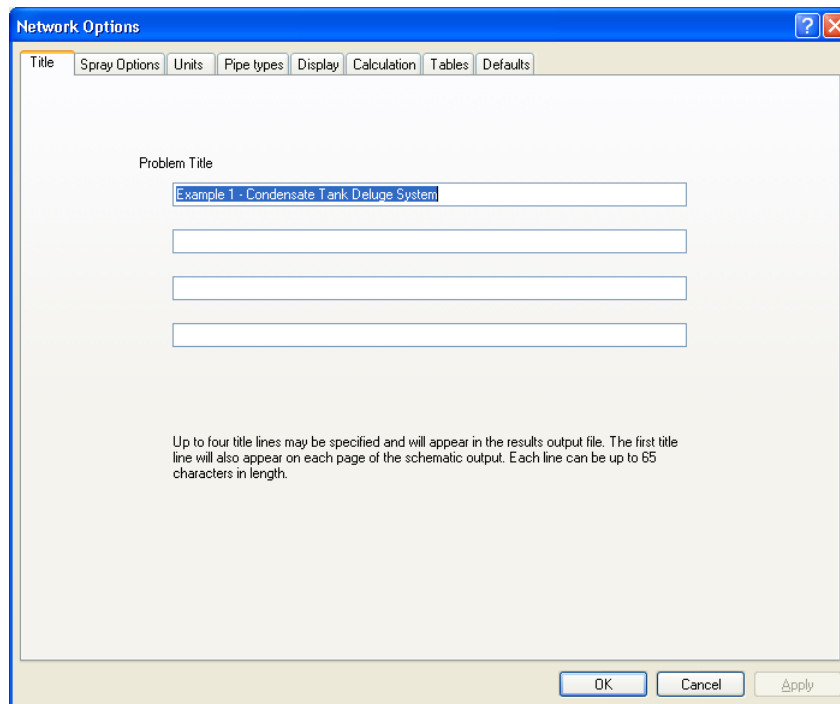
$$\begin{aligned}\text{Required flow rate through each nozzle} &= \text{Demand} / \text{No of nozzles} \\ &= 4712.4/72 \text{ lit/min} \\ &= 65.4498 \text{ lit/min}\end{aligned}$$

The flow rate required through each nozzle forms part of the nozzle data in the input file.

6.3 Initialisation

Title

The **Options | Title** option is used to enter a title for the run as Example 1 - Condensate Tank Deluge System.

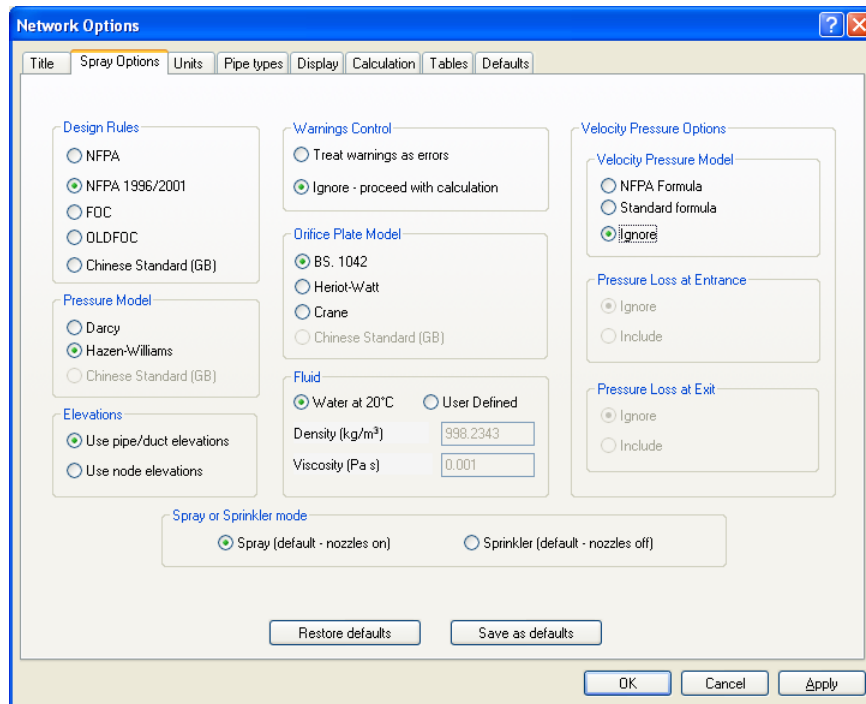


Parameter options

In this example we will use the Hazen-Williams equation with NFPA rules for fittings. The default values for the density and viscosity of water will be used.

Specifications data

In this system the specifications are particularly simple. Note that there is just one I/O node - the input node V/1. We will look separately at the specifications required for the design and calculation phases. Parameters are defined via the menu option **Options | Module** options:



Specifying units

Units will be specified as metric as in the previous two examples.

6.4 Creating a user defined pipe schedule

This will be created as in the previous example using **Libraries | Schedules**. The data for the user-defined pipe schedule is shown below. Note the value "unset" simply indicates that no value has been assigned for this nominal bore since the nominal bore is not used.

Schedule Name : SPECIAL.

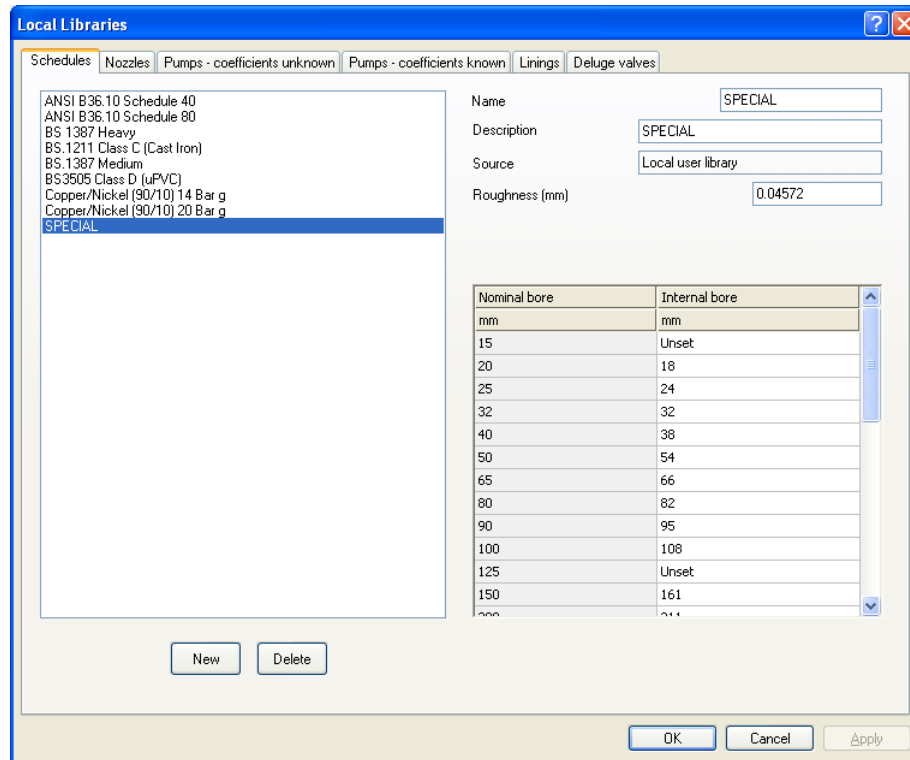
Roughness : 0.4572 mm.

Bore Data : See table below:

Nominal Bore (mm)	Internal Diameter (mm)	Nominal Bore (mm)	Internal Diameter (mm)
15	unset	400	393
20	18	450	unset
25	24	500	unset
32	32	600	unset
40	38	700	unset
50	54	750	unset
65	66	800	unset
80	82	850	unset
90	95	900	unset
100	108	950	unset
125	unset	1000	unset

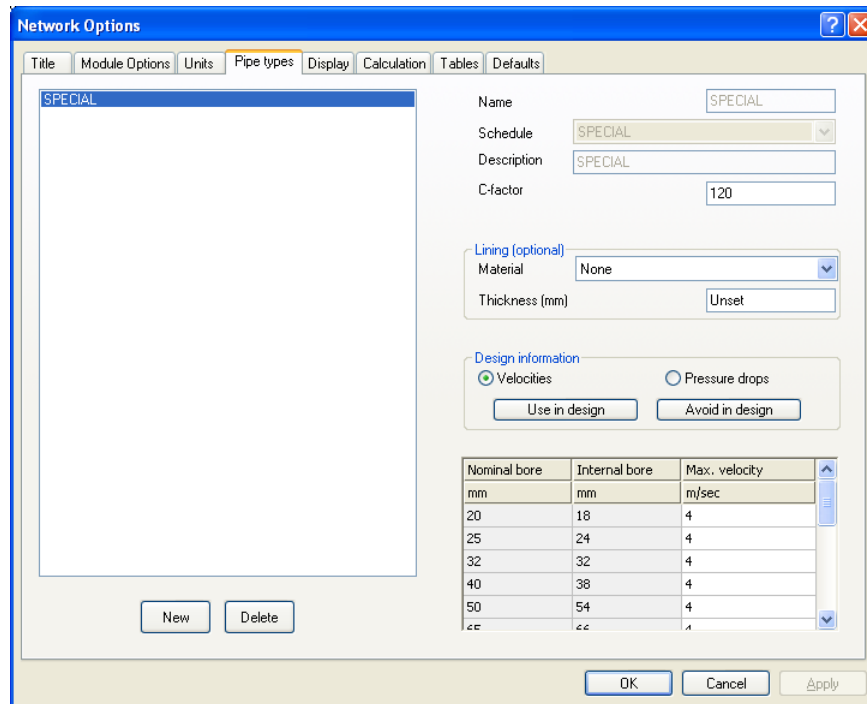
150	161	1050	unset
200	211	1100	unset
250	260	1150	unset
300	311	1200	unset
350	342		

When completed the Schedule dialog should appear as:



6.5 Creating a pipe type

The pipe type is created in the same way as in the previous example, with a C-factor of **120** and a maximum velocity on each pipe of **4 ms⁻¹**.



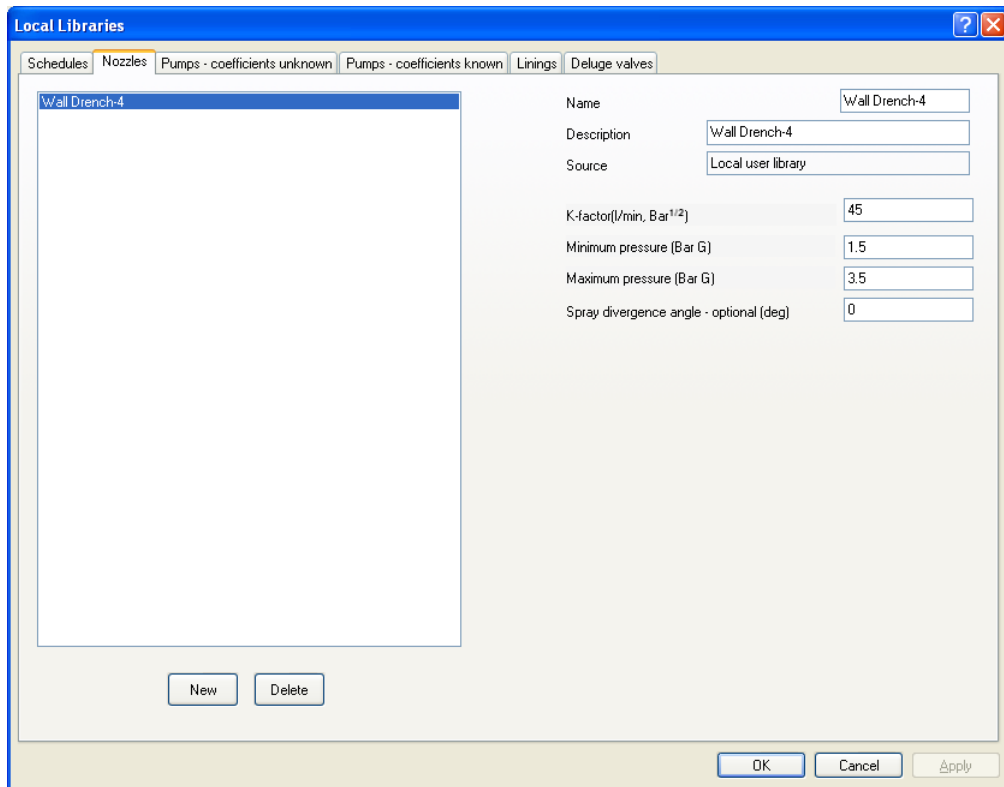
6.6 Nozzle characteristics

The pressure drop, P , across a nozzle is given by:

$$P = (Q/K)^2$$

Where Q is the flow rate through the nozzle and K is a constant. The nozzles in this system have a K -value of 45.0 (with P measured in bar and Q measured in lit/min). The minimum and maximum operating pressures are 1.5 and 3.5 bar G respectively. This nozzle performance data is contained in the library along with the pipe schedule data.

As in the previous example, nozzles are entered into the library via the **Libraries | Nozzles** menu - a dialog is displayed in the left-hand side with a list of available nozzles and the data is entered in the right-hand side. Select **New** to add a new nozzle, provide the data above and select **Apply** to accept the changes. The dialog should then appear as:



6.7 The design and calculation phases

As always, the simulation is run in two phases - the [design phase](#) and the [calculation phase](#).

In the design phase PIPENET will find appropriate sizes for pipes in the network based on the assumption that each nozzle is discharging at a rate of 65.4498 lit/min, as calculated above. The design velocity will be 4m/s for all pipes.

In the calculation phase PIPENET will find the pressures and flow rates throughout the system when every nozzle is discharging with a flow rate of at least 65.4498 lit/min. This is accomplished using the [Remote Nozzle Specification](#) option which ensures that every nozzle discharges fluid at or above its required rate. From the results of this phase we will be able to determine:

- The pressure required at the inlet (node V/1) to ensure that all nozzles discharge fluid at or above the required rate.
- Whether, under these conditions, the fluid velocity in every pipe in the designed network remains below the pipe's designed velocity.

6.8 Specifications

Design phase specifications

During the design phase PIPENET will assume that all the nozzles discharge at exactly their required rate (in this case 65.4498 lit/min). The user must supply the flow rates at all

I/O nodes except one. Since there is only one I/O node the user needs to provide no design specifications in this case.

Calculation phase specifications

We are going to use the [Remote Nozzle Specification](#) option for our calculation phase. This instructs PIPENET to ensure that all nozzles are discharging at or above their required rate (65.4498 lit/min). The *most remote nozzle* will supply water at exactly the required rate, whilst other nozzles in the system will supply at a slightly higher rate. Under this option the user must supply one less specification than there are I/O nodes in the system. However, in this case, there are none. In this example, the use of most remote nozzles adds one flow specification to the calculation phase. For further information on how to determine the number of specifications see [Specification rules](#).

Even though we are making no user-defined specifications, we must still remember to declare node V/1 as an input node.

6.9 Network data entry

This example includes many more pipes and nozzles than previous examples so we have chosen to create components with [tags](#), this is simply no more than placing a name before the numeric pipe label, for example with previous examples we had component labels like 1,2, 3... now we can have labels with tags C/1, C2, B1....

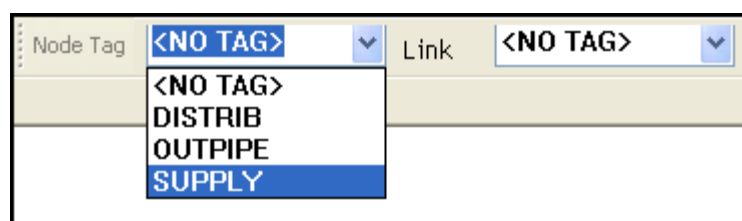
Using tags

We could lay out the network as in the previous examples and then using either the properties window or the tabular view change all of the labels manually. However there is a way to avoid this using the Tag tool bar:



Tags can be assigned separately for nodes and components, by default both have a setting of <NO TAG>. You can either:

- Click in the data entry field and a new tag followed by a Return.
- Select an existing tag by selecting the drop-down button to the right of the data entry field and selecting from the list of available tags.



If you have a tag selected in one of the drop-downs then whenever you create a new component, its automatically assigned numerical id will be prefixed with the specified tag and a '/'.

Pipe data

The following table shows the data for the A branches, the other two branches will have a similar pattern of connection.

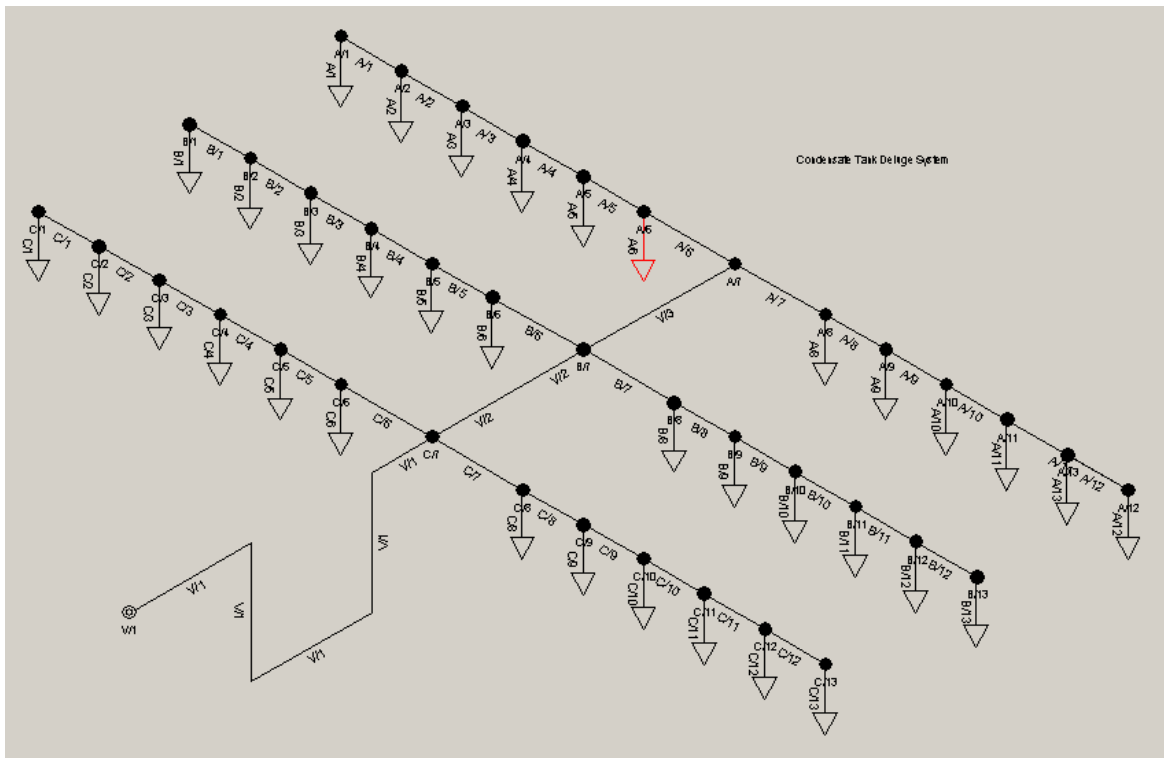
Pipe label	Input node	Output node	Length (metres)	Elevation (metres)	Fittings
V/1	V/1	C/7	0.5	0.5	Tee or Cross
V/2	C/7	B/7	3.28	3.28	Tee or Cross
V/3	B/7	A/7	3.28	3.28	Tee or Cross
A/1	A/2	A/1	2.068	0	
A/2	A/3	A/2	2.068	0	
...					
A/5	A/6	A/5	2.068	0	
A/6	A/7	A/6	1.034	0	
A/7	A/7	A/8	1.034	0	
A/8	A/8	A/9	2.068	0	
...					
A/12	A/12	A/13	2.068	0	

Note that no bores are provided since we will use the design phase to do the sizing for us.

Entering the network the hard way

It will probably be very difficult to ensure that you end up with a network which has the same numbering as that shown below, but the main points to note are that:

- The schematic has been laid out on an isometric grid.
- The three pairs of branches containing the nozzles have been labeled with tags of A B and C.
- The main line has been labeled with a tag of V.



An easier way

There is another way to generate the 3 pairs of branches. This is using the [automatic nozzle generation tool](#) on the Tools toolbar. We can use this to speed up data entry because all of the nozzles are of the same type and all of the pipes between the nozzles are of the same length. There are several ways we could do this. One way is to select the nozzle generation tool, when the dialog is displayed:

1. Set the tag to be used for pipes (A/B/C).
2. Set the tag to be used for nozzles (A/B/C).
3. Select the nozzle type from the right-hand window.
4. Set the length of the pipe between two nozzles as 2.068.
5. Set the angle at which the line is to be drawn as either -30 degrees or +150 degrees.

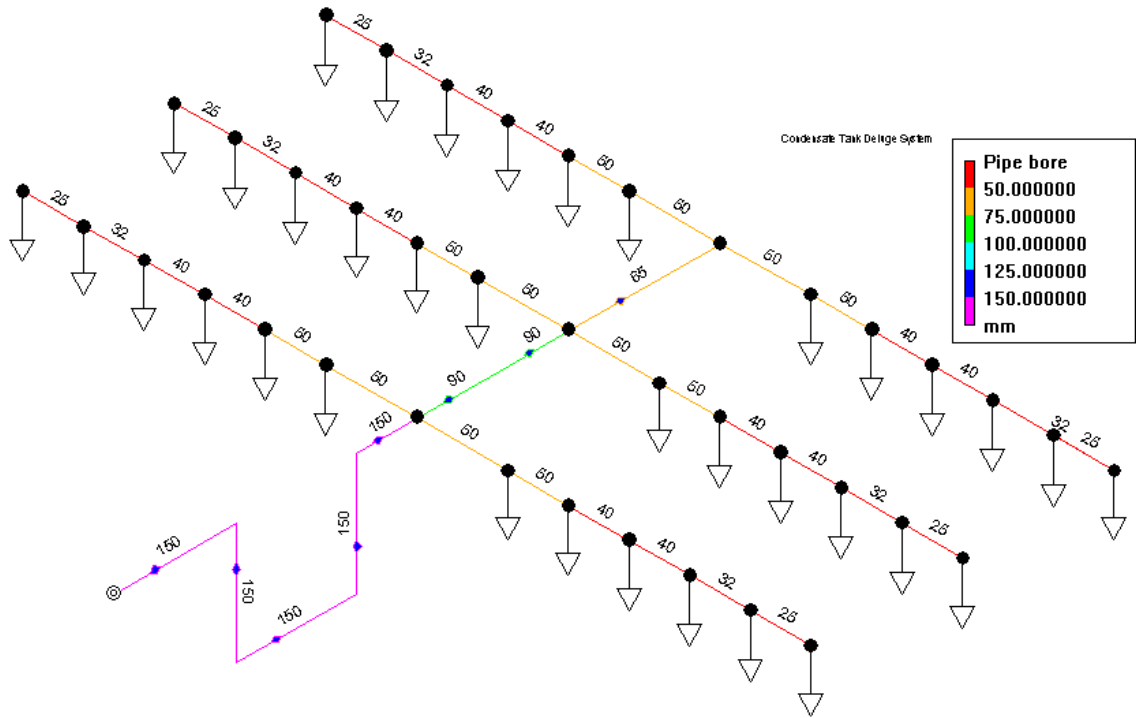
Note the tag fields will be used independently of any setting on the [tag tool bar](#).

This is done for each branch, i.e 6 times. Yet another way is to create one branch pair, say the A branch and then use the [Area tool](#) to reproduce the other two branch pairs. It only remains to position the branches and add in the main line.

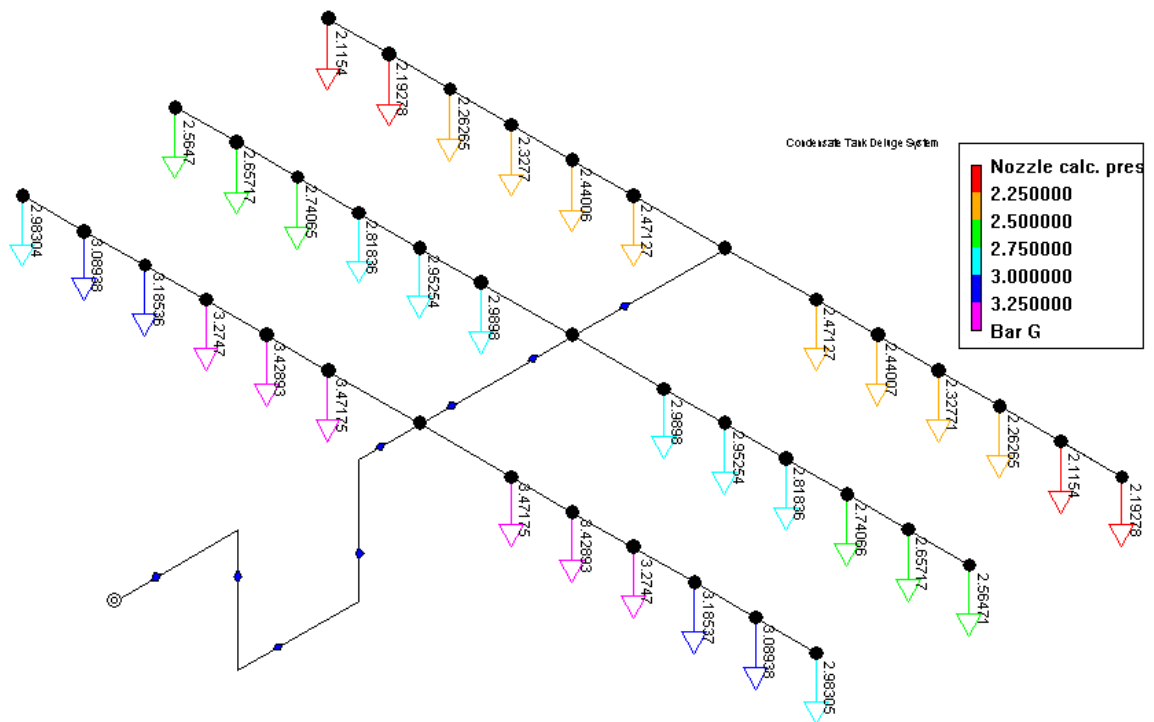
6.10 Calculation and results

The specifications are now complete, and a calculation may be performed as in the first example and the results inspected via the Browser or the Tabular view:

Useful information may be obtained using [colour coding](#), for example the following illustrates the bores calculated in the design phase:



Similarly the following illustrates the calculated pressure at each nozzle:



Example 4 - Pressures in the Ring Main

Part



7

7 Example 4 - Pressures in the Ring Main

7.1 Pressures in a ring main

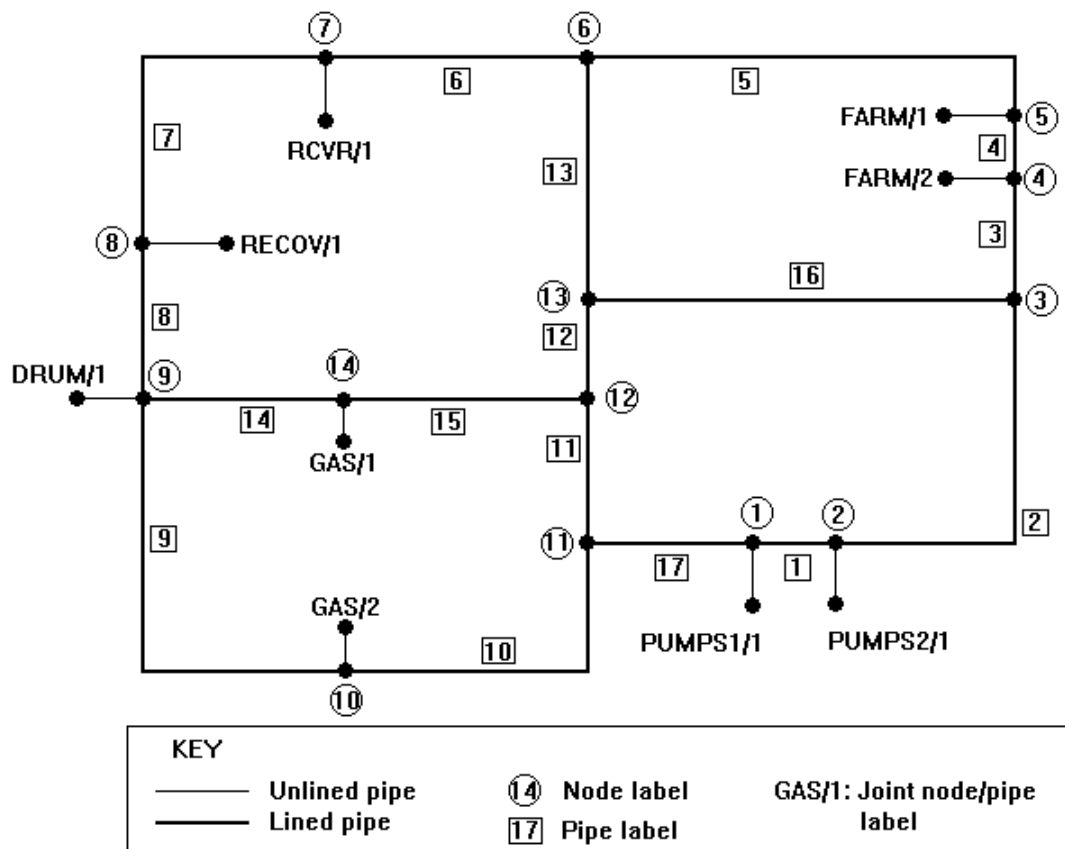
In this example we look further at the design of the ring main supplying the fire protection system in example 3. The example illustrates the following:

- Use of lined pipe schedules.
- Creation and use of a pump library file.
- Use of orifice plates to create a pressure drop.

The files relating to the first stage of this example are: `_ring1.dat` and `_ring.pdf`

The Network

The network is a ring main in a typical process plant, feeding a number of potential fire-hazard areas. The system is to be designed to protect five zones, one of which is the condensate tank farm investigated in the previous example. Only one of the zones may be discharging at any given time, but the system must be designed to cater for any of them. The pipes in the network are cast iron. The pipes in the primary main are below the ground and are lined with cement.



In example 3 we found that in order to supply the deluge systems for the Condensate Tank Farm, a pressure of 3.52 bar G is required at the deluge system inlet. In this example we

investigate how this supply requirement can be met by a pump at PUMPS2/1. This is done in two stages:

Stage 1

In stage 1 of this example we find the pressure required at the pump outlet (PUMPS2/1) in order to produce a pressure of 3.52 bar G at the node FARM/2.

Stage 2

In stage 2 of this example we add a pump to the network at PUMPS2/1 (Input node IN/1, output node PUMPS2/1). We will investigate the pressure it produces when supplying FARM/2. An orifice plate is used to ensure the correct pressure of 3.52 bar G is attained. The orifice plate will be sized for us by PIPENET. Given the pressures required at the other outlets, similar calculations could be carried out for them.

7.2 Initialisation

Title

The Title option is used to enter a title for the run ('Example 2 - Pressures In the Ring Main').

Units

The Units option can be used to select the units to be used. In this case we use Metric units.

Module parameters

We will use the Hazen-Williams equation with NFPA rules for fittings. The default values for the density and viscosity of water will be used.

Pipe schedule and Pipe Type data

The pipes are all taken from the built-in schedule 6, Cast Iron to BS.1211 Class C. The pipes in the primary main, which are underground, will be lined with a 2 mm thickness of cement. They will have a C-factor of 90 and a design velocity of 10 m/s. The outlet feed pipes and pump outlet pipes will be unlined. They have a C-factor of 120 and a design velocity of 8 m/s.

7.3 Creating a pipe type

The Pipe Type is set up in a manner similar to the preceding examples. The schedule in each case is Cast Iron BS.1211 Class C. The first pipe type is unlined, with a C-factor of 120 and design velocity of 8 m/s. The second type is lined with 2 mm of cement, has a C-factor of 90 and a design velocity of 10 m/s.

The dialogs for the two pipe types should appear as:

Network Options

Title | Spray Options | Units | **Pipe types** | Display | Calculation | Tables | Defaults

BS.1211 Class C (Cast Iron)
BS.1211 Class C (Cast Iron):Cement

Name: BS.1211 Class C (Ca
Schedule: BS.1211 Class C (Cast Iron)
Description: BS.1211 Class C (Cast Iron)
C-factor: 120

Lining (optional)
Material: None
Thickness (mm): Unset

Design information
 Velocities Pressure drops
Use in design | Avoid in design

Nominal bore	Internal bore	Max. velocity
mm	mm	m/sec
80	80.77	8
100	106.17	8
125	132.59	8
150	158.5	8
200	210.31	8

New | Delete

OK | Cancel | Apply

Network Options

Title | Spray Options | Units | **Pipe types** | Display | Calculation | Tables | Defaults

BS.1211 Class C (Cast Iron)
BS.1211 Class C (Cast Iron):Cement

Name: BS.1211 Class C (Ca
Schedule: BS.1211 Class C (Cast Iron)
Description: BS.1211 Class C (Cast Iron):Cement
C-factor: 90

Lining (optional)
Material: Cement
Thickness (mm): 2

Design information
 Velocities Pressure drops
Use in design | Avoid in design

Nominal bore	Internal bore	Max. velocity
mm	mm	m/sec
80	80.77	10
100	106.17	10
125	132.59	10
150	158.5	10
200	210.31	10

New | Delete

OK | Cancel | Apply

7.4 Network data entry

The following table shows the data for the A branches, the other two branches will have a similar pattern of connection. Refer to the [previous example](#) for the use of tags instead of pure numeric identifiers.

Pipe label	Input node	Output node	Bore (mm)	Length (metres)	Elevation (metres)	Fittings*
PUMPS1/1	PUMPS/1	1	250	33.45	-3.45	LRE x 2 BFLY
PUMPS2/1	PUMPS2/1	2	250	33.45	-3.45	LRE x 2 BFLY
RECOV/1	8	RECOV/1	200	15.0	5.0	LRE TEE BFLY
FARM/1	5	FARM/1	125	5.75	3.00	LRE x 2 TEE BFLY
FARM/2	4	FARM/2	125	5.75	3.00	LRE x 2 TEE BFLY
RCVR/1	7	RCVR/1	100	10.0	0.0	TEE BFLY
DRUM/1	9	DRUM/1	100	80.0	1.5	LRE TEE BFLY
GAS/1	14	GAS/1	150	15.0	1.5	LRE TEE BFLY
GAS/2	10	GAS/2	150	15.0	1.5	LRE TEE BFLY
1*	1	2	150	15.0	0.0	TEE x 2 BFLY
2*	2	3	150	122.5	0.0	LRE TEE BFLY x 2
3*	3	4	150	25.0	0.0	BFLY
4*	4	5	150	30.0	0.0	
5*	5	6	150	260.0	0.0	LRE BFLY
6*	6	7	150	75.0	0.0	BFLY x 2
7*	7	8	150	135.0	0.0	LRE BFLY x 2
8*	8	9	150	60.0	0.0	BFLY x 2
9*	9	10	150	162.0	0.0	LRE BFLY x 2
10*	10	11	150	92.5	0.0	LRE BFLY x 2
11*	11	12	150	55.0	0.0	BFLY x 2
12*	12	13	150	20.0	0.0	BFLY x 2
13*	13	6	150	95.0	0.0	TEE BFLY x 2
14*	9	14	150	75.0	0.0	TEE BFLY x 2
15*	14	12	150	75.0	0.0	TEE BFLY x 2
16*	13	3	150	220.0	0.0	TEE x 2 BFLY x 2
17*	1	11	150	82.5	0.0	TEE BFLY x 2

Fittings* TEE - Tee or Cross; BFLY - Butterfly valve; LRE - Long radius elbow

The pipes in this example are taken from a built-in schedule. However, some of the pipes are lined with cement (pipe labels with an * after the name), which is not a built-in lining. The private data file must thus include data for cement. This data is entered by choosing the Pipe Lining option from the **Libraries | Linings** menu. The data required is the lining name and roughness value. As we are using the Hazen-Williams equation the roughness will not actually be used, but a value must be specified (we use 10^{-5} mm).

7.5 Pump data and the library

The performance coefficients of the pump are unknown so we must take coordinates from the performance curve and use the menu option **Libraries | Pumps - Coefficients Unknown** to find values for the coefficients A, B and C. The data required is as follows:

Pump descriptor: ELECTRIC_PUMP
 Minimum flow rate = 2000 lit/min
 Maximum flow rate = 13000 lit/min

Coordinates from the curve:

Flow rate (lit/min)	Pressure (Bar)
2000	12.5
5000	11.0
9000	8.5
11000	7.0
13000	5.5

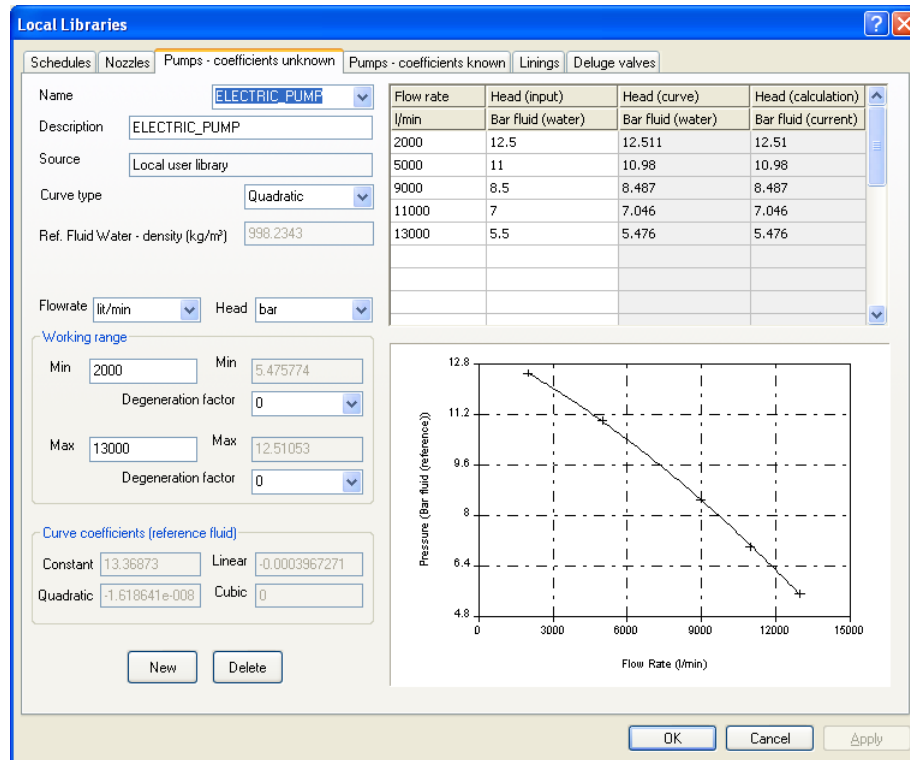
Adding the pump to the library

For this example, a pump is also needed. This data is entered using the menu option **Libraries | Pumps - Coefficients Unknown** (or if you already have the library dialogs selected by selecting the Pumps - Coefficients Unknown tab).

To specify the pump details proceed as follows:

1. Select the **New** button.
2. Provide the name for the pump (this is the name that will appear in the drop-down box at the top right-hand on the dialog when data entry is complete).
3. Provide an optional description.
4. Provide a minimum flow rate of 2000 litres/minute and a maximum flow rate of 13000 litres/minute.
5. Now place the cursor in the first cell in the data entry grid in the lower left-hand side of the dialog and enter the first flow rate value of 2000.0.
6. Tab to the next field and enter the corresponding pressure of 12.5.
7. Continue in this way entering the remaining data pairs, using tab to move from one cell to the next.
8. When all the data pairs have been entered, select the **Apply** button to calculate and display the pump curve.

The completed pump specification should appear as:



The screenshot shows the 'Local Libraries' dialog box for an 'ELECTRIC_PUMP'. The 'Pumps - coefficients unknown' tab is active. The configuration includes:

- Name: ELECTRIC_PUMP
- Description: ELECTRIC_PUMP
- Source: Local user library
- Curve type: Quadratic
- Ref. Fluid Water - density (kg/m³): 998.2343
- Flowrate: lit/min
- Head: bar
- Working range: Min 2000, Max 13000, Degeneration factor 0
- Curve coefficients (reference fluid): Constant 13.36873, Linear -0.0003967271, Quadratic -1.618641e-008, Cubic 0

The table of flow rate vs. head data is as follows:

Flow rate (l/min)	Head (input) (Bar fluid (water))	Head (curve) (Bar fluid (water))	Head (calculation) (Bar fluid (current))
2000	12.5	12.511	12.51
5000	11	10.98	10.98
9000	8.5	8.487	8.487
11000	7	7.046	7.046
13000	5.5	5.476	5.476

The graph shows Pressure (Bar fluid (reference)) on the y-axis (ranging from 4.8 to 12.8) and Flow Rate (l/min) on the x-axis (ranging from 0 to 15000). The curve is a downward-sloping line with data points at (2000, 12.511), (5000, 10.98), (9000, 8.487), (11000, 7.046), and (13000, 5.476).

7.6 Specifications

Design phase specifications

As all the pipe diameters are already set, we are not interested in the results of the Design phase of the simulation. However, we must still supply a set of design phase specifications (in this case we choose to set all flow rates to the same as in the calculation phase, as given below).

Calculation phase specifications

We set the following calculation phase specifications:

Stage 1

FARM/2: Pressure 3.52 bar g.

Flow rate 5364 lit/min.

Other outlets: Flow rate 0 lit/min

PUMPS1/1: Flow rate 0 lit/min

Stage 2

(The network having had a pump added with input node IN/1, and output node PUMPS2/1)

FARM/2: Flow rate 5364 lit/min

Other outlets: Flow rate 0 lit/min

PUMPS1/1: Flow rate 0 lit/min

IN/1: Pressure 0 bar g.

7.7 Calculation and results

Stage 1 Results

The results (see table below) show that in order to produce a pressure of 3.52 bar G at the node FARM/2, with a flow rate of 5364 lit/min, a pressure of 5.752 bar G is required at the pump outlet, PUMPS2/1.

Flow at inlets		
	Inlet Node Pressure (bar G)	Flow rate (lit/min)
PUMPS1/1	5.543	.0000 *
PUMPS2/1	5.752	5364.

Note: A * after a value indicates that this is a specification

Stage 2 Results

When the pump is added a pressure of 8.543 bar G is produced at the node FARM/2. In order to reduce this pressure to the required pressure of 3.52 bar G, an orifice plate can be added to the pipe FARM/2. This is done using the Orifice Plate tool. The pressure drop required across the orifice plate is $(8.543 - 3.52) = 5.023$ bar G. This forms part of the data for the orifice plate. Given this required pressure drop, PIPENET can size the orifice plate for us if we carry out another calculation run — the required diameter is 69.3567mm.

The files relating to the solution to Stage 2 of this problem are also stored in the Examples directory as:

- pipenet\examples_ring2.dat
- _ring.pdf
- _ring2.out
- _ring2.pmp

Example 5 - A Maintenance Area Sprinkler System

Part

8

8 Example 5 - A Maintenance Area Sprinkler System

8.1 A maintenance area sprinkler system

The previous examples have shown how to create new networks from scratch. These have been simple examples with few pipes. In practice, the engineer will often be dealing with pre-existing networks, and they can typically have several hundred or more pipes. This example considers the alteration of a previously existing system that is more elaborate than the previous one. The example illustrates the following points:

- The alteration of existing networks, and the addition of new components.
- The modelling of a SPRINKLER system with only a few nozzles working at any one time.
- The ease with which both loop and tree systems may be mixed.
- Comparison of the Mass Balance and Remote Nozzle Specification options.

The files relating to this problem are supplied with PIPENET and are: `_maint.dat` and `_maint.pdf`.

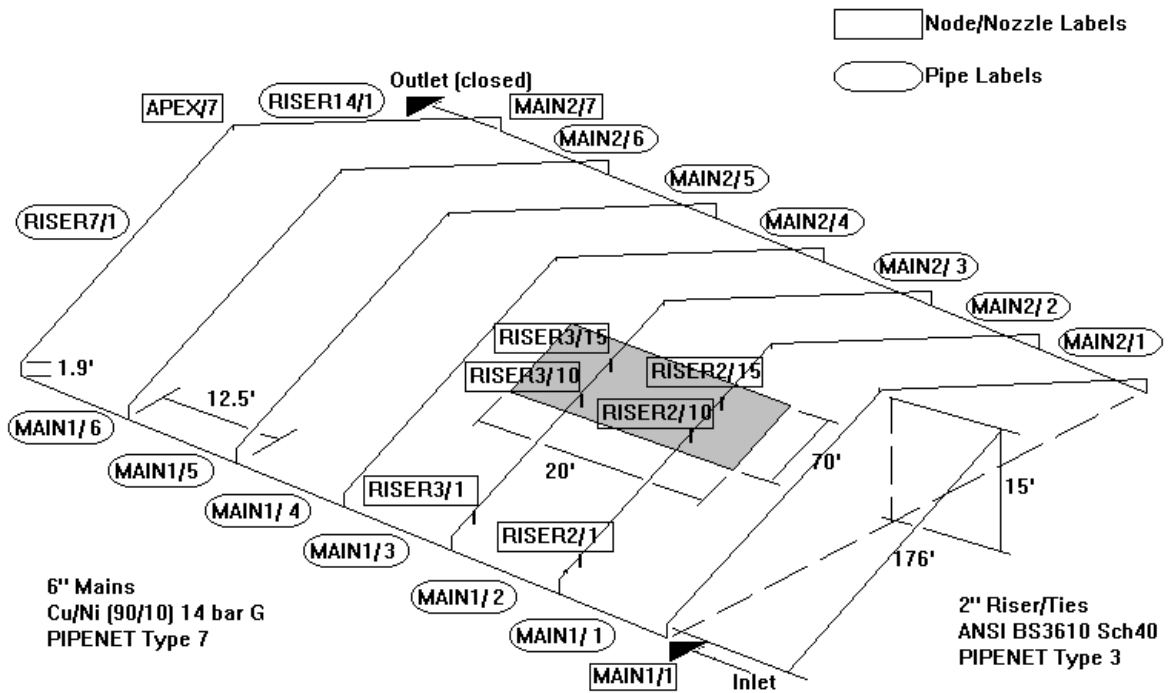
Description of the Problem

The system is a regular network, consisting of 2 long parallel mains, 6" in diameter, made of 90/10 cupro-nickel 14 bar G running along the walls of the Maintenance Area, with risers running along the roof to the apex at 12.5 foot intervals. The nozzles are fixed to the risers at 10 foot intervals. The apex of the roof is 15 feet above the mains, which is located 1.9 feet below the top of the wall. There are 18 nozzles on each riser and a tie set.

In this example, we will assume that the basic system is installed, and that a new fire hazard over an area measuring 20 feet by 70 feet is to be covered with a larger application rate than was originally assumed. To achieve the greater cover, we first have to show that the installed system will not be able to cope, and then add some new pipes and nozzles to cover the new hazard.

In this example, we are not concerned in this example with the pump(s) used to feed the system, although it would usually be necessary to consider if the installed ones are inadequate for the new hazard.

The system is illustrated below. Note that not all the nozzles are marked, and some pipe and node labels have not been shown.



8.2 Initialisation

Entering the network details

Each node with the tag NEW1 has a nozzle with the same label (e.g. node NEW1/6 has a nozzle NEW1/6). The two risers NEW2 and NEW3, which spring from slightly different points on MAIN3, are otherwise identical to NEW1.

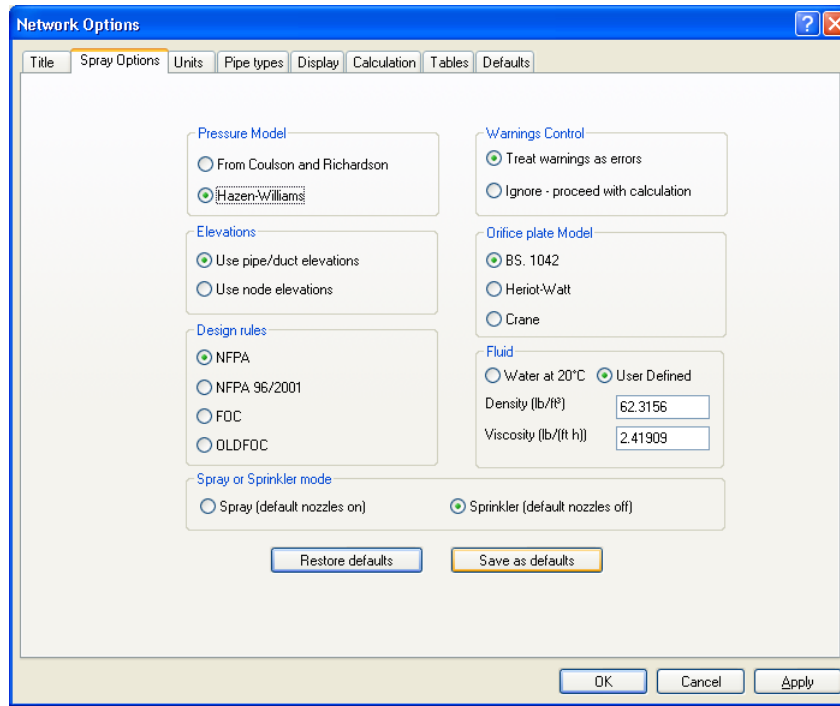
Pipe label	Input node	Output node	Bore (in)	Length (ft)	Elevation (ft)	Fittings*
MAIN3/1	MAIN1/1	MAIN3/1	6	99.15	8.65	EL90 x 2 TEE
MAIN3/2	MAIN3/1	MAIN3/2	6	12.5	0	
MAIN3/3	MAIN3/3	MAIN3/3	6	12.5	0	
NEW1/1	MAIN3/1	NEW1/1	2	5	1	TEE
NEW1/2	NEW1/1	NEW1/2	2	2.9	1.975	EL45
NEW1/3	NEW1/2	NEW1/3	2	2.9	1.975	EL45
NEW1/4	NEW1/3	NEW1/4	2	2.9	1.975	EL45
NEW1/5	NEW1/4	NEW1/5	2	2.9	1.975	EL45
NEW1/6	NEW1/5	NEW1/6	2	2.9	1.975	EL45
...						

Fittings* EL90 - standard 90 Elbow; TEE - Tee or cross; EL45 - standard 45 elbow.

Modelling of a sprinkler system

The essential difference between a SPRAY system and a SPRINKLER system is in the

nozzles. In a SPRAY system, all the nozzles are usually open, and will discharge when water is supplied to them, whereas in a SPRINKLER system, each nozzle is individually controlled, either with a fusible plug or electronically. When modelling SPRINKLER systems it is necessary to specify some nozzles as being turned on. To change from the default of a SPRAY system select the menu option **Options | Spray options** and then the Sprinkler option:



Nozzle Data

The existing nozzles are all of the same type with a K-factor of 95. The operational range is 1.3–9.5 bar G. Note that these data are in metric units, while the rest of the data is in Imperial.

Preliminary Calculations

First, we must convert the nozzle data to the same units as the rest of the problem. In UK gal/min and psi G, the K-factor is 5.4871 and the operational range 18.856–137.79 psi G.

The required application rate over the new hazard is 0.6 UK gal/ft²min

$$\text{Area of application} = 70 \times 20 = 1400 \text{ ft}^2$$

$$\text{System demand at inlet} = 0.6 \times 1400 = 840 \text{ UK gal/min}$$

$$\text{Number of nozzles} = 12 \text{ (RISER2/10 - RISER2/15 and RISER3/10 - RISER3/15)}$$

$$\text{Required flow from each nozzle} = 840/12 = 70 \text{ UK gal/min}$$

$$\text{Required nozzle inlet pressure} = (70/5.4871)^2 = 162.7 \text{ psi G}$$

This is above the maximum operating pressure for this nozzle type, so we can conclude that the present system cannot cope. With 18 additional nozzles, the required flow rate per nozzle becomes:

$$\begin{aligned} &= 840/(12+18) \\ &= 840/30 \\ &= 28 \text{ UK gal/min} \end{aligned}$$

$$\text{Required nozzle inlet pressure} = (28/5.4871)^2 = 26.04 \text{ psi G}$$

This is within the operating range for the nozzles as $18.856 < 26.04 < 137.79$ psi G

Parameter options

In this example we will use the Hazen-Williams equation with NFPA rules for fittings. The default values for the density and viscosity of water at 20°C will be used.

8.3 Specifications

Design Rules require that the area to be protected receive a certain total flow rate. In this case, this is 0.6 UK gal/ft²min over 1400 ft². As calculated above, if the 30 nozzles in the area each discharge 28 UK gal/min, this requirement will be met. PIPENET can deal with this situation in two ways:

- Ensure that the total flow rate into the system (and as the nozzles are the only output, the total flow rate into the area) is equal to the sum of the flow rates required at each nozzle (mass balance option).
- Ensure that each nozzle individually meets its required flow rate, so that the area is covered uniformly (remote nozzle specification option).

In this example we will investigate both, to see if there is a significant difference in the results.

Design phase specifications

All the pipes in the network are already sized, so we are not interested in the results of the design phase. However, we must still provide design phase specifications. The network has two I/O nodes, and therefore requires one flow rate specification. This will be 0 on node MAIN2/7, indicating that the outlet is closed.

Calculation phase specifications

If we use the Mass Balance option for our calculation phase, PIPENET automatically assigns a calculation phase specification to every I/O node in the network based on the design phase flow rate (See the Technical Manual for a full description of the Mass Balance option and making specifications).

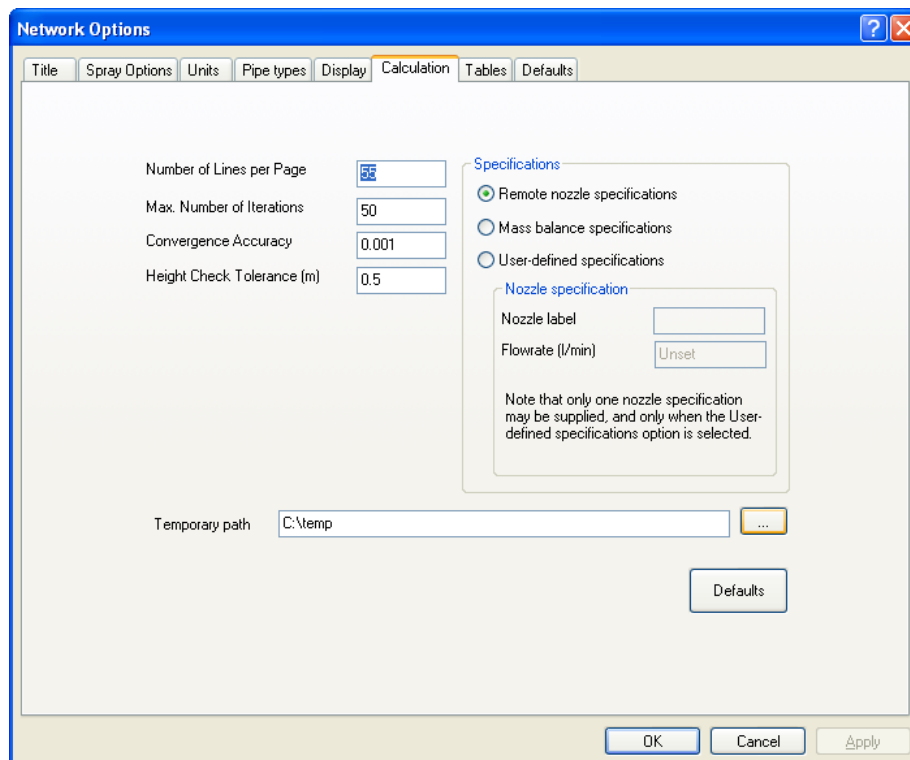
If we use the Remote Nozzle Specification option, we must give one more calculation

phase specification. In this case, we simply specify that node MAIN2/7 have a flow rate of 0; we are simply copying the design phase specification (See the Technical Manual for a full description of the Remote Nozzle Specification option and making specifications). Even if we are making no user-defined specifications on it, we must still remember to declare node MAIN1/1 as an input node.

8.4 Calculation and results

Mass balance and remote nozzle specification

These selection of mass balance or remote nozzle specification is made via the menu option **Options | Calculation**:



Results

Both simulations should run smoothly, and in neither one does the flow exceed the designed speeds or pressures of the network. If you run the simulation and look at the "Flow Through Nozzles" section of the output browser file you should observe that the specifications are indeed met by the simulation. Using the Mass Balance Option we ensure that the system delivers exactly 840 UK gal/min. However, the Remote Nozzle Specification only uses 6% more water and ensures that a more uniform protection is provided. The pumps would have to deliver 47.4 psi G in this case, rather than 43.3 psi G if the Mass Balance is used.

PIPENET has provided an assessment of the feasibility of installing the new risers, and other configurations could just as easily be tested. If they require the enlargement of some of the existing pipes because these exceed their designed velocities, PIPENET issues a

warning in the report file.

The Schematic

Part

9

9 The Schematic

9.1 Schematic Window

The Schematic window is the primary means of entering and viewing networks. It closely resembles the schematic window of earlier products but has a number of improvements.

When the window is first displayed it is presented with a light-grey background suitable for general viewing. The background colour may be changed to white or black. However, for coloured links and text, it will generally be found that a white background is unsuitable for viewing.

New elements are added by selecting the appropriate element tool from the tool palette and then placing and drawing the component using the mouse.

All labeled elements created via the schematic are automatically assigned a unique label. Labeled elements include nodes, link elements and attribute elements. Numeric labels are used (no tags) with each component type having its own set of unique labels.

The background colour and the font sizes used for labeling components can be changed using the Display Options dialog.

Schematic Underlay

A facility has been included whereby a graphic may be imported and displayed as a background to the main schematic. Display of this [underlay](#) is enabled and disabled via the [View menu](#). The underlay may be zoomed independently of the main network to achieve relative scaling and registration. Zooming the network, zooms the underlay by the same selected zoom factor. In normal use the procedure to use an underlay commences with a new network:

1. Import and display the underlay.
2. Select a suitable zoom size for the underlay.
3. Commence laying out the PIPENET components using the underlay as a guide.

9.2 Schematic Underlay

It is possible to import a graphic to underlay the main pipe network. The graphic to be imported must be a Windows enhanced metafile (file extension .EMF), a Windows metafile (file extension .WMF) or an AutoCAD .DXF file.

Currently the image is loaded on the first request to display the underlay (see [View menu](#)). Once loaded the underlay may be zoomed to establish the relative scale between the underlay and the network. Generally the procedure will be as follows:

1. Load the underlay.
2. Establish a suitable scaling for the underlay.
3. Start drawing the network over the underlay.

The display of the underlay can be turned on or off via the View menu - when turned off the underlay will track any changes to the scale of the network, retaining the correct relation.

Zoom of the underlay is independent of the network, whereas zooming the network, also zooms the underlay to maintain relative scales.

Currently the underlay can not be translated left-right or up-down, although the network can be moved using the [Area Tool](#).

9.3 Selection Tool

The first button is the selection tool, and is used to select a single component. Simply click the left mouse button whilst pointing at a component to select it. A selected component is shown highlighted in red and its attributes displayed in the [Properties Window](#). Right-clicking the mouse on a selected component will display a pop-up menu, the contents of which are dependent on the type of component selected, but generally, the pop-up menu will allow you to:

- Delete a component.
- Add [waypoints](#).
- Delete [waypoints](#).
- Insert a node in a pipe.
- Reverse the direction of a component, note reversing the direction of a pipe will negate the elevation change.
- Copy and paste the attributes of a component.

The selection tool can also be used for dragging nodes to a new position, and for dragging pipe components, such as orifice plates, along the length of a pipe. To drag a node or a component, click the left mouse button and whilst holding the button down move the mouse to the desired position and release the mouse button. Note, that if a node is dragged then all of the components to which it is connected also move with it.

Full [undo/redo](#) is available for all operations with this tool.

Adding a waypoint

If you left-click on a pipe, and with the mouse button held down move the mouse then a waypoint will be inserted at the selected point in the component link.

Selecting several components

To select multiple components select the first component in the normal way by placing the cursor on or near the component and left-clicking on the component. Subsequent selections are made in the same way but with the keyboard **Ctrl** key held down whilst making the selections. Another way to select multiple components is using the [Area tool](#).

Copy/Paste

To copy the attributes from one *source* component onto another *target* component of the

same type:

1. right-click on the source component and select the **Copy** option (alternatively use Ctrl-c)
2. right-click on the target component and select the **Paste** option (alternatively use Ctrl-v) - all attributes are copied from the source component to the target component.

If the source and target components are pipes, then prior to the copy all fittings are removed from the target pipe and replaced with the fittings from the source pipe.

If **Paste (Incl. layout)** is selected instead of Paste the process is very similar, except that any waypoints and components (for example orifice plates) are also copied from the source to the target.

Explode Node

This facility can be used to break all of the links meeting at a designated node. To explode a node simply right-click on the node and select the Explode node option

9.4 Pan and Zoom Tool

This tool can be used to zoom the network to a required size or to pan across the network.

- To zoom the network, hold down the left mouse button and drag the cursor until the network is at the required size. Dragging to the right will enlarge the network, to the left will minimize.
- To pan across the network, click the left mouse button whilst in the schematic window and move the mouse in the direction you wish to pan. Click the left mouse button again to cease panning. Whilst panning, if the cursor is moved to the edge of the window the network will scroll along until the end of the scroll bar is reached.

Pan and zoom can also be achieved [using the mouse wheel](#), if one is present.

9.5 Area Tool

The Area tool is used to select and manipulate a number of components at the same time. With the Area tool it is possible to:

- Move a group of components
- Select a group of components.
- Copy-paste a group of components.
- Delete a group of components.
- Mirror (left-right) a group of components, i.e. mirror the components about a vertical centre line.
- Invert (up-down) a group of components, i.e. flip the components about a horizontal centre line.

- Turn all nozzles, within the selected area, on or off.

To use the tool, click the left mouse button at the point which is to be the top-left of a rectangular area. Whilst holding the mouse button down move to the point which is to be the bottom-right-hand corner of the rectangle and release the mouse button. A dashed outline of the defined rectangle is drawn and all nodes and components which lie completely within the rectangle are selected.

Via the Edit menu, or by right-clicking, the selected items can now be moved, deleted, mirrored, inverted, copied and nozzles turned on/off. Full [undo/redo](#) is available for all operations with this tool.

Selected items

The items marked as selected are:

- Each node contained within the rectangle.
- Each link component contained within the rectangle whose input and output nodes are both within the rectangle, i.e. components crossing the boundary are not selected.

Moving the selected area

The rectangle and all selected items within the rectangle can be moved by selecting the left mouse button and whilst it is down dragging the rectangle to a new position. On releasing the mouse button the components will be drawn at the new position.

Copying selected items

Items selected within the rectangle can be copied to the clipboard, from whence they can be pasted onto the same network, or onto another network in a separate instance of the same module. Copy/paste are activated via the **Edit** menu options, the **Copy** and **Paste** buttons on the toolbar, or by right clicking within the defined rectangle to display a pop-up menu of options.

Note that if you are using copy/paste to copy from instance of a module to another instance of the same module, ensure that both instances are using the same unit systems.

9.6 Polygon Tool

The Polygon tool is used to select and manipulate a number of components at the same time. It is similar to the [Area tool](#) except that it allows components to be selected within a polygonal area. With the Polygon tool it is possible to:

- Move a group of components
- Select a group of components.
- Copy-paste a group of components.
- Delete a group of components.
- Mirror (left-right) a group of components, i.e. mirror the components about a vertical centre line.
- Invert (up-down) a group of components, i.e. flip the components about a horizontal centre line.
- Turn all nozzles, within the selected area, on or off.

Note that for the purposes of mirror and inversion operations the centre of the polygon is the centre of the enclosing rectangle.

To use the tool, proceed as follows:

1. Define the first point by holding the shift key down click the left mouse button.
2. Define the second point by moving the mouse to the location of the second point and whilst holding the shift key down, click the left mouse button;
3. Repeat step 2 for as many points as you require to define the polygonal area.
4. When you have defined all the points right click the mouse anywhere in the schematic (without the shift key being down) to close the defined polygon - the last defined point is connected to the first.

Via the Edit menu, or by right-clicking, the selected items can now be deleted, mirrored, inverted, copied and nozzles turned on/off. Full [undo/redo](#) is available for all operations with this tool.

Selected items

The items marked as selected are:

- Each node contained within the defined polygon.
- Each link component contained within the polygon whose input and output nodes are both within the polygon, i.e. components crossing the boundary are not selected.

Moving the selected area

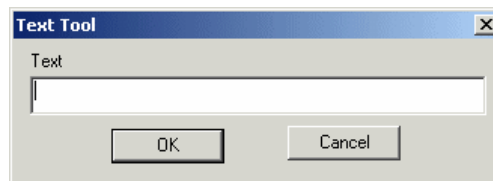
The polygon and all selected items within the polygon can be moved by selecting the left mouse button and whilst it is down dragging the polygon to a new position. On releasing the mouse button the components will be drawn at the new position.

Copying selected items

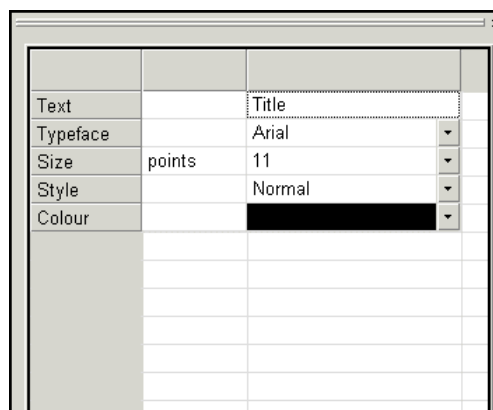
Items selected within the polygon can be copied to the clipboard, from whence they can be pasted onto the same network, or onto another network in a separate instance of the same module. Copy/paste are activated via the **Edit** menu options, the **Copy** and **Paste** buttons on the toolbar, or by right clicking within the defined rectangle to display a pop-up menu of options.. or the Copy and Paste buttons on the toolbar.

9.7 Text Tool

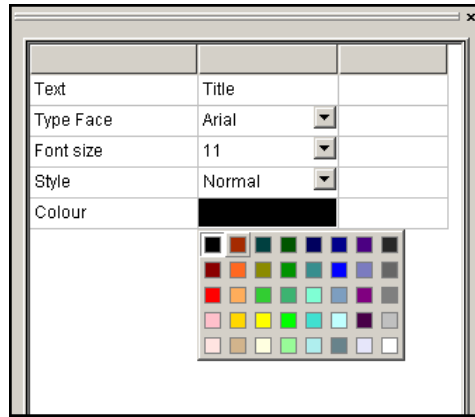
The Text Tool is used to place text on the schematic, for example, as titles and additional labeling information. To place a text item on the schematic click the left mouse at the approximate point at which the first character is to appear; the following dialog appears:



Enter the text which is to appear in the text field and then select **OK** to accept the text or **Cancel** to abort. Text options can be specified in the Properties Window and include typeface, size, style (normal, bold, italic or bold and italic) and colour.



To change the colour of the text click on the arrow button in the cell. Selecting the drop-down arrow in the Colour field will display a small selection of colours:



Moving and editing text

Once a text element has been added to the schematic it can be selected, edited and moved:

1. Click on a text element and its properties can be edited in the [Properties window](#).
2. Click and drag to move the text.
3. Right-click on a text element and select **Delete** from the pop-up menu to delete the text.

9.8 Link Component Tools

For all link elements, the procedure for creating a new link is as follows:

1. Select the appropriate element tool from the tool palette.
2. Place the cursor at the point where you want the input node to appear and left click.
3. If the selected point coincides with an existing node then that node becomes the input node; otherwise a new node is created and displayed at the selected point.
4. A line representing the link element is drawn and tracks mouse movements.
5. Place the cursor at the point where you want the output node to appear and left click.
6. As with the input point, if the selected output point coincides with an existing node then that node will become the output node, otherwise a new node is created.
7. If, in between defining the input node and the output node, you want to abort creation of the link then select the Escape key.

When the component is drawn it will be displayed in blue to indicate that this is new component for which the component defaults have been used. If any changes are made to the component's attributes then it will change to black.

Waypoints

For some components, specifically those for which both an output and an input node is displayed (excludes components such as spray nozzles and Transient caissons), additional,

intermediate points may be specified between the input and output nodes. These intermediate nodes, or waypoints, do not form part of the hydraulic network and merely exist to aid in the layout of the schematic. To add waypoints at the time a new component is added, proceed as follows:

1. Place the cursor at the point where you want the input node to appear and left click.
2. If the selected point coincides with an existing node then that node becomes the input node; otherwise a new node is created and displayed at the selected point.
3. A line representing the link element is drawn and tracks mouse movements.
4. To add an intermediate point hold the shift key down and left-click the mouse at the desired position. This may be repeated as many times as you like to create multiple-segment pipes.
5. If you left click without holding down the shift key then the output node is created.
6. If, after creating the pipe, you want to add additional waypoints or to move waypoints, this can be done using the [selection tool](#).
7. The creation of a pipe can be aborted any time between the creation of the input and the out node by pressing Escape.

If you have selected the display of direction and/or the presence of fittings on pipes then the associated symbol will be displayed on each segment of the pipe.

Undefined or invalid components

When a pipe, or in fact any link component, is first drawn in the schematic it is coloured blue to indicate that either the component has not had its attributes specified and/or it has an invalid combination of attributes, for example a zero length pipe. When one or more attributes are entered for the component its colour changes to black (or white if the background is black).

The colour of a component may revert to blue in either of the following situations:

1. If a check is performed using the check button on the calculation toolbar and the component is found to be invalid.
2. If the file is saved and re-opened, and on re-loading the component it is found to be invalid.

9.9 Pipe Component Tools

These tools are used for placing components on a pipe, for example orifice plates and equipment items. These components can only be added to existing pipes and cannot be created in isolation:

1. Place the cursor on the pipe.
2. Left click to add the attribute element to the pipe.
3. The component may subsequently be moved along the length of the pipe using the [Selection tool](#).
4. Attributes for the added component are displayed in the Properties Window as for

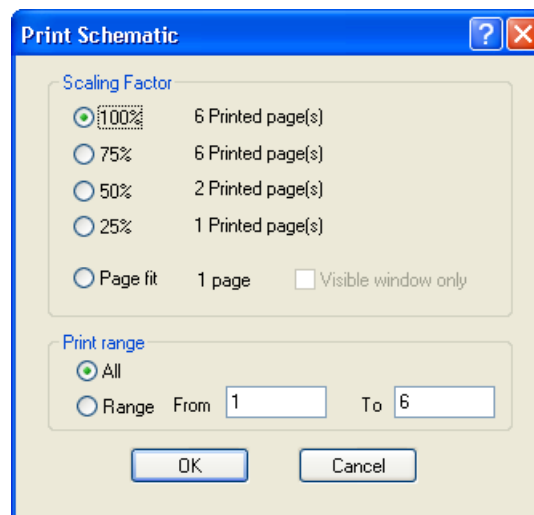
link components.

With any of the attribute element tools selected it is possible to move any attribute element along the length of a pipe using click and drag; see also the section on the [Selection Tool](#).

9.10 Schematic Printing

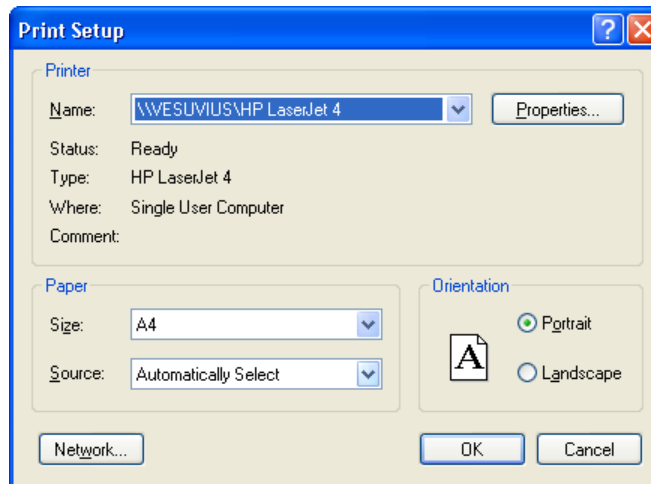
A schematic may be printed by selecting the **File | Print** option. The schematic may be printed to any supported Windows' printer on a single page or across multiple pages. Note however, that printing to a large plotter using a Windows' printer driver may be very slow since some Windows' drivers will work by rasterizing the schematic. For optimum drawing you should [export the schematic](#) for off-line plotting.

To print a schematic select the **File| Print** option, this will display a dialog box showing a range of print scales and the number of pages required to print at each scale, and offering the option to print all pages or a selected range of pages.



However many pages are required to display the schematic, the network will be displayed centred across all pages. Each printed page will show the network title, the date, and the page number in the form Page m of n. Page fit can be selected to print the complete schematic on one page. If the option **Visible window only** is selected, only that part of the network visible in the schematic window will be printed on a single page.

Printing will be to the currently selected printer with the currently selected page orientation. To change either or both of these: select the **File | Print Setup** menu option to display the standard Windows print setup dialog.



The appearance of the printed schematic can be previewed using the **File| Print Preview** option.

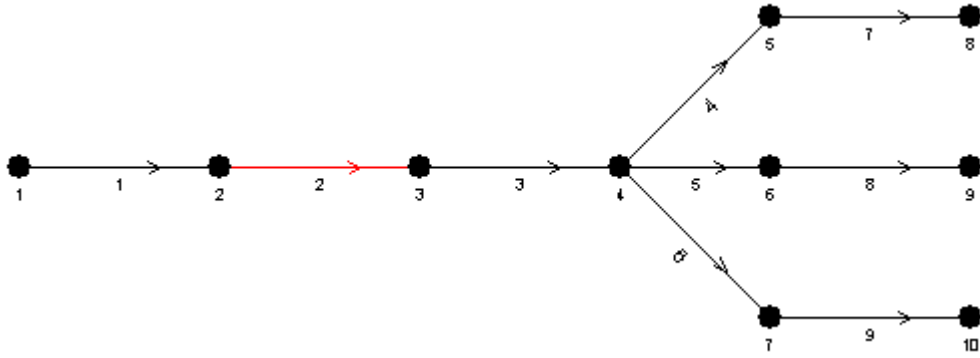
9.11 Exporting the Schematic

The schematic may be exported for use with other graphical or CAD programs. Currently the network may be exported as an [HP-GL2](#) file or as an [AutoCad™ DXF file](#).

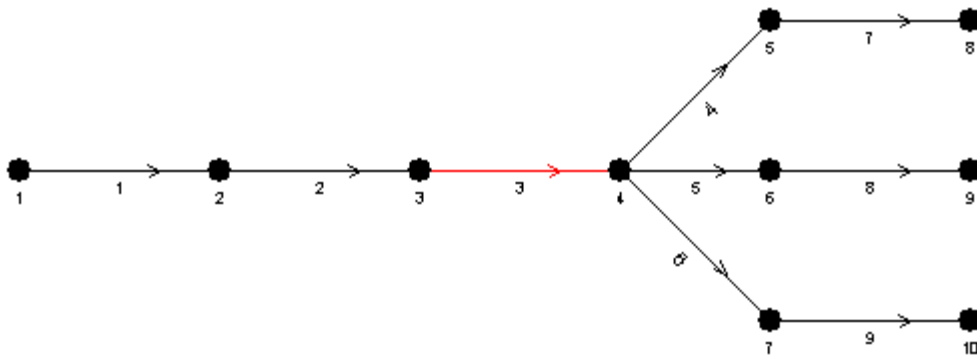
9.12 Moving around the network

To move around the network, displaying each component in turn can of course be achieved by using the mouse to select the component and then viewing its attributes in the [Properties Window](#). Alternatively, you can view all components of a given type via the [Tabular Window](#), clicking on a row will highlight the corresponding component in the schematic window, and vice-versa, selecting a component in the schematic window highlights the corresponding component in the [Tabular View](#). Yet another way is to use the two arrow buttons in the bottom right-hand corner of the Properties Window to select the next component of the same type (right-facing arrow) or the previous component of the same type (left-facing arrow).

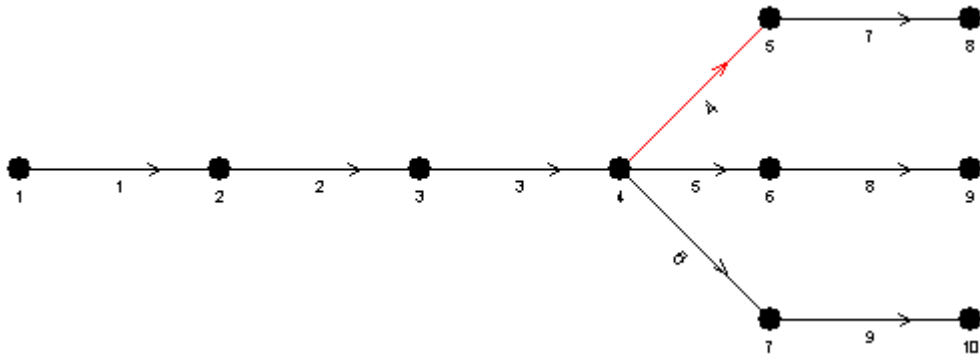
There is another way to move around the network, highlighting components in turn, using the four cursor keys on the keyboard. This is best illustrated by the following example. With pipe 2 the currently selected component and all component directions being from left to right, selecting the right cursor key moves to pipe 3 and selecting the left cursor key moves backwards to pipe 1.



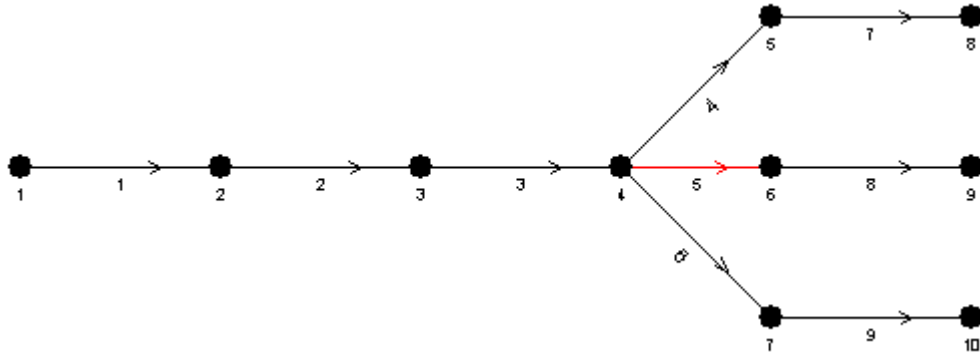
Now consider what happens when we are on pipe 3:



Selecting the right cursor key will move to uppermost component on the right, i.e. pipe 4:



To move to pipe 5 from pipe 4 simply select the down cursor key:



Selecting the down cursor key again will move to pipe 6 whilst selecting the up cursor key will move back to pipe 4.

If the component is reversed, i.e the input node is to the right of the output node then selecting the right cursor key will move to the link to the left. Similarly selecting the left cursor key will move to the link to the right of the current link. So selecting the right cursor key is interpreted as a move in the component direction and the left cursor key as a move in the reverse direction.

9.13 Use of the mouse

The left and right-hand mouse buttons are used as in many other Windows programs:-

Left mouse button

Used to select items and, if held down, to drag components around the network

Right mouse button

Used to display context dependent menus

If a mouse wheel is present then this can also be used for panning and zooming, the operations being similar to those found in programs such as AutoCad or Adobe Acrobat.

Simple vertical scroll

Move the mouse wheel to scroll the schematic up and down.

Zoom

Press the **Ctrl** key and move the mouse wheel up and down to zoom in and out of the schematic. The point of the graph directly under the mouse cursor will stay the same.

Panning

Click the mouse wheel, and whilst holding down the mouse wheel, move the mouse to pan the network.

The Tabular View

Part

10

10 The Tabular View

10.1 Tabular View

The Tabular View window and the [Properties Window](#) are used to enter and edit the attributes of components displayed on the schematic. The Tabular View window cannot be used for adding new components or deleting existing components. These operations must be performed via the schematic.

Each tabular view displays the information for one component type selected from the drop-down list at the top of the window:

	Label	Input node	Output node	Diameter		Length	Elevation	C-factor	Status	Type	Design group
				in	ft						
1	MAIN1/1	MAIN1/1	MAIN1/2	6	12.5	0	120	Normal	Copper/Nickel (90/10) 14 Bar g	Unset	
2	MAIN1/2	MAIN1/2	MAIN1/3	6	12.5	0	120	Normal	Copper/Nickel (90/10) 14 Bar g	Unset	
3	MAIN1/3	MAIN1/3	MAIN1/4	6	12.5	0	120	Normal	Copper/Nickel (90/10) 14 Bar g	Unset	
4	MAIN1/4	MAIN1/4	MAIN1/5	6	12.5	0	120	Normal	Copper/Nickel (90/10) 14 Bar g	Unset	
5	MAIN1/5	MAIN1/5	MAIN1/6	6	12.5	0	120	Normal	Copper/Nickel (90/10) 14 Bar g	Unset	
6	MAIN1/6	MAIN1/6	MAIN1/7	6	12.5	0	120	Normal	Copper/Nickel (90/10) 14 Bar g	Unset	
7	MAIN3/1	MAIN1/1	MAIN3/1	6	99.15	8.65	120	Normal	Copper/Nickel (90/10) 14 Bar g	Unset	
8	MAIN3/2	MAIN3/1	MAIN3/2	6	12.5	0	120	Normal	Copper/Nickel (90/10) 14 Bar g	Unset	
9	MAIN3/3	MAIN3/2	MAIN3/3	6	12.5	0	120	Normal	Copper/Nickel (90/10) 14 Bar g	Unset	
10	MAIN2/1	MAIN2/1	MAIN2/2	6	12.5	0	120	Normal	Copper/Nickel (90/10) 14 Bar g	Unset	
11	MAIN2/2	MAIN2/2	MAIN2/3	6	12.5	0	120	Normal	Copper/Nickel (90/10) 14 Bar g	Unset	
12	MAIN2/3	MAIN2/3	MAIN2/4	6	12.5	0	120	Normal	Copper/Nickel (90/10) 14 Bar g	Unset	
13	MAIN2/4	MAIN2/4	MAIN2/5	6	12.5	0	120	Normal	Copper/Nickel (90/10) 14 Bar g	Unset	
14	MAIN2/5	MAIN2/5	MAIN2/6	6	12.5	0	120	Normal	Copper/Nickel (90/10) 14 Bar g	Unset	
15	MAIN2/6	MAIN2/6	MAIN2/7	6	12.5	0	120	Normal	Copper/Nickel (90/10) 14 Bar g	Unset	
16	RISER1/1	MAIN1/1	APEX/1	2	178	15	120	Normal	ANSI B36.10 Schedule 40	Unset	
17	RISER4/1	MAIN1/4	APEX/4	2	178	15	120	Normal	ANSI B36.10 Schedule 40	Unset	
18	RISER5/1	MAIN1/5	APEX/5	2	178	15	120	Normal	ANSI B36.10 Schedule 40	Unset	
19	RISER6/1	MAIN1/6	APEX/6	2	178	15	120	Normal	ANSI B36.10 Schedule 40	Unset	
20	RISER7/1	MAIN1/7	APEX/7	2	178	15	120	Normal	ANSI B36.10 Schedule 40	Unset	

Sorting

Components are initially displayed in data entry order, however, rows may re-ordered by clicking in a column heading. For example, to sort pipes in ascending order of diameter, click on the heading for the pipe diameter column. To sort in descending order of diameter, click on the column heading a second time - clicking on a heading toggles between ascending and descending order.

Cell shading

Cells are normally displayed with a white background, however the following cell shadings may also be observed:

- Cells coloured light grey are read-only.
- Cells coloured yellow (currently only for pipe sizes) indicates that the displayed

size has been calculated during the design phase.

Selecting a row

Left-click on the leftmost cell of the row to select a row.

Editing a cell

The contents of editable cells (non-editable cells will be grayed out) are either of the direct data entry type or of the drop-down selection type. For example in the pipe tab, pipe bore and pipe length are both of the direct data entry type - simply click on the cell to edit or re-enter the value. The pipe status is selected from a drop down list and is one of Normal, Broken or Blocked.

Cell edits can be undone using the [Undo/Redo](#) facilities.

Printing

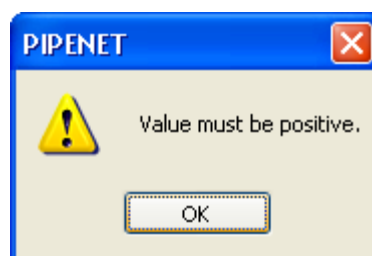
The current grid can be printed by selecting the Print button.

10.2 Validation

All attributes are validated as they are entered:

- Fields are validated to check that they are of the correct type, that is if a field must contain only numeric data then only entry of numeric digits and optional sign and decimal point are permitted.
- Simple range checking is carried out to ensure that numeric values are within range, for example pipe bores must always be positive, filter coefficients must always be negative, temperatures must be at or above absolute zero and so on.

If an invalid entry is made then a simple dialog is displayed indicating the fault, for example:



Clicking **OK** leaves the error highlighted, the value must be corrected before moving on to further editing.

10.3 Copying Cells

A cell or a rectangular group of cells can be copied using techniques similar to those used in Excel or 123. All paste operations can be [undone](#).

Copying a single cell

To copy a single cell simply right-click on the cell to display a popup menu and then select the **Copy** option. The copied cell can be pasted to another target cell by right-clicking on the target cell and selecting the Paste option. However, note that a cell in one column can only be copied to another cell in the same column, since it makes no sense to allow copying between columns which represent completely different types of attribute.

Paste does not work in read-only columns or component label columns.

Copying a cell or range of cells

Right click on the cell to be copied and select the Copy option. Now select the target group of cell as follows:

1. Left click the first target cell.
2. Whilst holding down the shift key left-click on the last target cell (in the same column).
3. Right click on any cell within the selected group to display the popup menu.
4. Select the **Paste** option.

Copying a single cell to multiple, non-contiguous cells in the same column

The previous operation will also work if the selected cells in the column are non-contiguous:

1. Left click the first target cell.
2. Whilst holding down the Ctrl key select any number of other cells in the column.
3. Right click on any cell within the selected group to display the popup menu.
4. Select the **Paste** option.

Paste-in-Column

Since the operation of reproducing a single value in a column is common, a shortcut is provided via the Paste-in-Column option in the popup menu. Simply point to the value to be repeated in the column and select **Paste-in-Column**.

Copying a contiguous group of cells from one row to the corresponding cells in another row

Select the source cells as follows:

1. Left click the first source cell.
2. Whilst holding down the shift key left-click on the last source cell (in the same row).
3. Right click on any cell within the selected group to display the popup menu.
4. Select the **Copy** option.

Select the target cells in the same manner only now select the Paste option in the popup menu. Note the source and target selections must start and end in the same columns.

Copying a rectangular group of cells from one area of the grid to another

Select the source area as follows:

1. left-click a cell at one corner (top-left for example) of the rectangular group.
2. whilst holding down the shift key left-click on the diagonally opposite corner cell to select the group.
3. Right click on any cell within the group to display the popup menu.
4. select the **Copy** option.

Select the target area in the same manner only now selecting the Paste option in the popup menu. Note the following:

- The source and target areas must have the same shape, i.e. they must be the same number of columns wide and the same number of rows high. Note however, the one exception to this in the next paragraph.
- The source and target areas must start and end on the same column.

Copying a contiguous group of cells from one row to the corresponding cells in several rows

A combination of the previous two copy operations provides the facility to copy cells from one row to the corresponding cells in a number of rows:

1. Select the cells from the source row as described in "Copying a contiguous group of cells from one row to the corresponding cells in another row".
2. Select **Copy** from the popup menu.
3. Select the target cells as described in "Copying a rectangular group of cells from one area of the grid to another".
4. Select **Paste** from the popup menu.
5. Repeating a value.

Copy cells to external programs

Cells can be copied in the ways described above and then pasted into an external program, typically a spreadsheet. All cells in the Tabular View can be selected by clicking in the top left-hand corner cell.

Specifications

Part

11

11 Specifications

11.1 Introduction to Specifications

In order to solve a network, boundary conditions must be provided in the form of flow or pressure specifications on input and output nodes or pressure specifications on internal nodes (an internal node is any node which is not an input or output node). These specifications must obey the rules described more formally in the following [Specification Rules](#) section.

Many of the aspects of specifications can however, be described by reference to a simple single pipe network.

With this simple example, an initial approach might be to provide equal flow specifications on both the input and output nodes. However, since the output flow must equal the input flow, one of these specifications is not required. If we provide two identical flow specifications then there is redundancy and there is no unique solution to the network. If instead, we provide two different flow specifications then the specifications would be inconsistent, and again there would be no solution.

With one flow specification provided at one node we know the flow at the other node, however, we do not know the pressure. In fact pressures cannot be determined without the specification of a reference pressure. So, for our simple network, it turns out that we must provide two specification, one of which must be a pressure specification. Thus there are three possibilities:

1. We provide a flow specification on the input and a pressure specification on the output
2. We provide a pressure specification on the input and a flow specification on the output,
3. We provide a pressure specification at both the input and output.

This can be generalized to larger networks with any number of input and output nodes to the simple statement that:

The number of specifications must be equal to the total number of input and output nodes, and at least one of the specifications must be a pressure specification. See [Specification Rules](#) for further details and the special considerations which apply to the Design Phase, nozzles and remote specifications.

Disjoint Network

A network is considered disjoint if it is in two or more unconnected parts, or sub-networks. The following is an example of a simple disjoint network, with two sub-networks A and B:



Since each sub-network is solved separately, the specifications in each sub-network must be valid. Thus in the above example there must be a total of four specifications, with sub-networks A and B each having at least one pressure specification.

It is obvious from this example that the network is disjoint, however, disjoint networks can also arise in a less obvious way from the use of [breaks and blocks in pipes](#). Consider the following simple three-pipe network, with the central pipe blocked



The network was initially setup with the pipe in the normal, unblocked state and the calculation ran satisfactorily with a flow specification provided at the input and a pressure specification provide at the output. When the blocked pipe was added the network refused to calculate - why? Simply, that the blocked pipe has split the network into two disjoint networks, one consisting of the single pipe A/1 and the other of the single pipe C/1. Whilst the network containing the pipe C/1 includes the original pressure specification, the A/1 network does not have a pressure specification. It should be noted that with a blocked pipe a zero flow specification is added to the node at each end of the block, hence there are correct number of specifications.

11.2 Specification Rules

Assumptions

1. Input and output nodes correspond to those points in the network where fluid enters or leaves the network.
2. Internal nodes are those nodes which are not inputs or outputs.
3. Sub-networks may be created by the presence of breaks and blocks.
4. If a node is at one end of a break then it is considered to have an attached pressure specification.
5. If a node is at one end of a block then it is considered to have an attached flow specification.
6. In the Design Phase an arbitrary pressure of 50 bar G is associated with one of the nodes, therefore a user-supplied pressure specifications is not used in this phase.

Design phase

1. There must be one (and only one) pressure specification, which may be on an input node, an output node or an internal node.

2. In a network with a total on n input and output nodes, all but one of these nodes must have a flow specification.

Calculation Phase

1. There must be at least one pressure specification.
2. Pressure specification may appear on input, output or internal nodes.
3. The total number of pressure and flow specifications must equal the total number of input and output nodes, but see the following:
4. A user supplied pressure specification is not required in the Analysis Phase if there are one or more nozzles present. The rule that the total number of specifications must equal the total number of input and output nodes still holds.
5. Selection of the Most Remote Nozzle option adds one flow specification to the Analysis Phase. This means that we must only provide $n - 1$ pressure or flow specifications, where n is the number of input or output nodes.

11.3 Breaks and Blocks

In terms of the solving of a network modelling breaks and blocks are modelled as follows:

Block

Each of the input and output nodes of the break is assumed to have an associated zero flow specification.

Break

Each of the input and output nodes of the break is assumed to have an associated pressure specification.

Each break or block may separate a single network into two sub-networks, and since specifications must be valid in each sub-network, problems can occur. The most likely problem to arise is that the presence of a blocked pipe breaks a valid network into two sub-networks with one of the sub-networks having no associated pressure specification.

11.4 User Interface

Viewing and editing specifications

The specifications associated with a node can be viewed in a number of ways:

1. by clicking on a node, the details of the specification appear in the [Properties window](#) where they may be edited. If no specifications have been attached to the node then the properties window appears as:

Label	DISTRIB/6	
Elevation	0.00000	m
Input/Output node	NO	
Design Spec.	NO	
Calculation Spec.	NO	
Results		
Pressure	n/a	

Properties Fittings

- by displaying the [tabular view window](#) and selecting the component type as Node to display all nodes with details of any specifications attached or Design phase specification or Calculation phase specification as required.

Adding specifications

Specifications are added by selecting the appropriate node and then setting the status of the node as an I/O node as appropriate, changing the Design specification drop-down to **YES** if you want to add a design specification and the Calculation Specification drop-down to **YES** if you want to add a calculation specification. Changing either of these will result in the display of additional attributes. For example, adding a calculation phase specification will change the display to:

Label	DISTRIB/6	
Elevation	0.00000	m
Input/Output node	NO	
Design Spec.	NO	
Calculation Spec.	YES	
Pressure	Unset	Bar G
Flow	Unset	m ³ /sec
Results		
Pressure	n/a	

Properties Fittings

where the pressure and/or flow can be provided.

Removing specifications

Specifications are removed simply by clicking on the node and changing the Design specification and/or Calculation specification options to **NO**.

Checking specifications

Specifications are checked during the performance of a check operation along with height checking and general consistency checking. This can be initiated by a user selecting the **Calculation | Check** menu option or more readily by selecting the Status tab in the Properties Window.

A number of messages, relating to specifications can appear in the status window. Most will be errors preventing a calculation from being performed. Specifications are checked separately for the Design and Analysis phases.

The errors and warnings are as follows:

Broken pipe encountered - warning

The presence of a broken pipe may separate a network into two sub-networks, where each sub-network is checked separately for consistency of specifications.

Blocked pipe encountered - warning

The presence of a blocked pipe may separate a network into two sub-networks, where each sub-network is checked separately for consistency of specifications. There is a potential danger here that one of the sub-networks may be left without a pressure specification, resulting in a network for which there is no solution.

Node has no inputs and is not an I/O node - warning

Taking into account component directions, a node has been found that has no inputs. It may have been the user's intention to associate a specification with the node.

Node has no outputs and is not an I/O node - warning

Taking into account component directions, a node has been found that has no outputs. It may have been the user's intention to associate a specification with the node.

No design pressure specification - error

One, and only one, design pressure must be provided for the design phase.

No design flow specification -error

At least one flow specification must be provided in each sub-network, together with a single pressure specification.

One I/O node with no flow specification is required for Design

For the Design phase at least one I/O node must be provided which has no flow specification. The I/O node may have a pressure specification.

No analysis pressure specification - error

At least one design pressure must be provided for each sub-network. See also blocked pipe warning above.

There must be at least two specifications - error

At least two specifications must be provided for each phase.

Network is over specified in analysis phase

More specifications than are necessary have been supplied, that is:

number of flow specifications + number of Pressure specifications > number

of I/O nodes.

Network is under specified in analysis phase

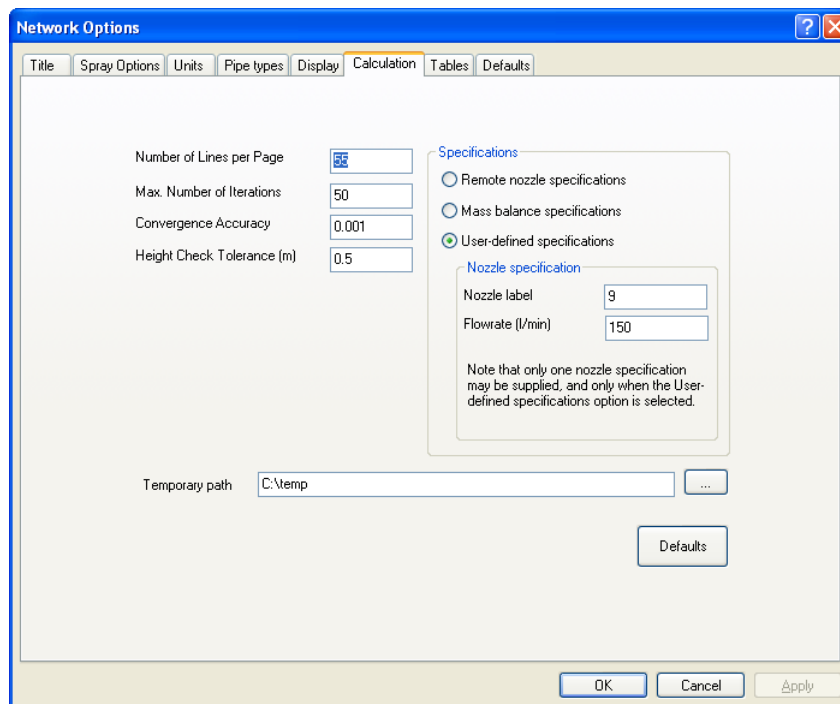
Insufficient specifications have been supplied, that is:

number of flow specifications + number of Pressure specifications < number of I/O nodes

11.5 Nozzle Specification

For deluge systems the first calculation is almost always the [most remote nozzle option](#). This assumes that all the nozzles have to deliver at least the required minimum flow rate. This may not always be essential or required. For example, there could be one nozzle which is at a much higher level than the other nozzles but is there only to deliver a nominal amount of water. There could be another nozzle somewhere in the system which is in a critical area. This particular nozzle may not be the most remote nozzle. It might be sufficient if this nozzle met the required minimum flow rate. The user can then select this nozzle and assign its flow rate. The "real" most remote nozzle would then deliver less than the required flow rate but that does not matter.

By definition one can only assign the flow rate at one nozzle and the flow rate must be given by the user. This is achieved using the **Options | Calculation** dialog but only when the **User defined specifications** option is selected:



In this example there is a specification on nozzle 9 with a flow rate of 150 l/min.

Note that if there are any overboard dump valves present in the network then the **Remote**

nozzle specifications and **Mass balance specifications** options will be greyed out. This is because overboard dump valves can only be used with the **User-defined specifications** option.

Status Checking

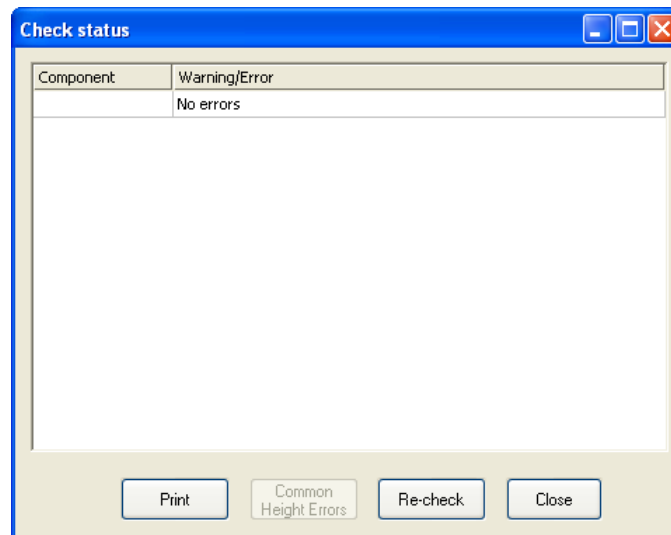
Part

12

12 Status Checking

12.1 Status checking

A facility is provided for checking the correctness of the currently defined network prior to attempting to perform a calculation. This will check for component errors, specification errors and height elevation inconsistencies. This facility is activated whenever a calculation is attempted or explicitly by selection of the **Calculation | Check** option, when the following window is displayed:



If there are no errors or warnings, the window contains a single status line as depicted above. If there are errors and/or warnings these are shown one to a line, with the first column showing a component label (where appropriate) and the second column a description of the error. All components found to be in error, for example a zero length pipe will be coloured blue.

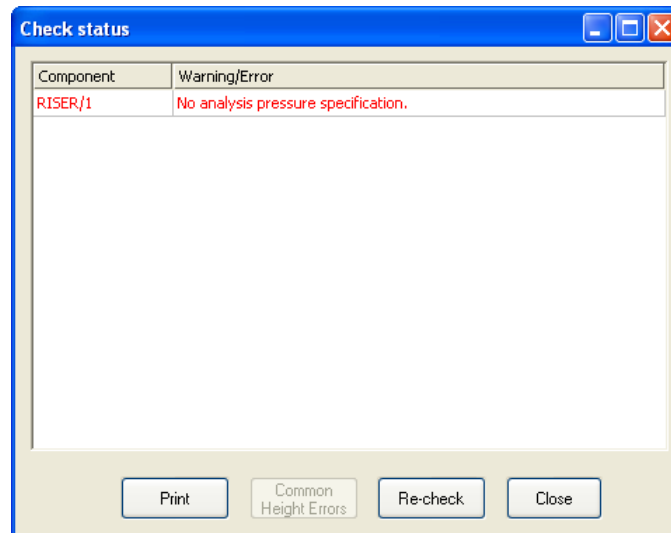
Warnings are shown in back text, and indicate possible problems with the network. Errors are shown in red text and indicate problems that may prevent a successful calculation.

If a component number is displayed in a cell then double-clicking anywhere in the row will highlight the component in the schematic window, scrolling the schematic window to display the component if it is not already visible.

For reference purposes a copy of the check results can be printed via the **Print** button.

12.2 Specification Checks

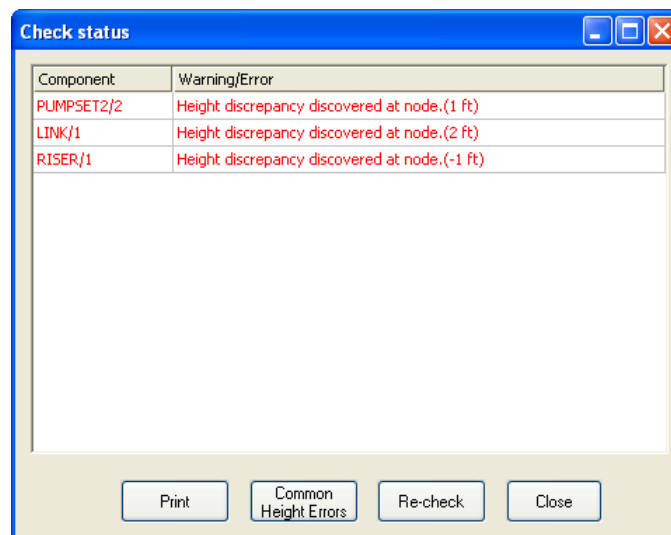
When a check is activated the number of specifications and their type is validated against the rules defined in [Specification Rules](#). The following is the check window displaying a specification error.



Double-clicking in the first column of a row containing an error will select a node in the sub-network containing the error

12.3 Height Checking

Height checking will be performed if pipe elevations are used, and essentially involves summing the rises and falls in every loop to check that the sum is zero (within the height check tolerance. For every loop in error one line will be displayed in the status window, with the component identifying one node in the loop. The error description will include the value of the error in user-defined length units. Clicking on the component cell will highlight all of the components in the loop, as well as creating a [path](#). The creation of a path means that a [graphical elevation profile](#) can be displayed.



If two or more height errors are found then selecting the **Common Height Errors** button will highlight all of the pipes that appear in two or more loops. This is not guaranteed to pinpoint the error, but it may help.

Colour Schemes

Part

13

13 Colour Schemes

13.1 Colour Schemes

A colour scheme is a simple set of rules used for the colouring of components on the schematic. At any time two colour schemes may be in effect, one for colouring nodes and one for colouring pipes/nozzles. The two colour schemes can be selected from an application specific set of colour schemes. There is a default colour scheme for both nodes and pipes/nozzles which provides the default colouring of:

- RED - item is selected
- BLUE - item is not completely defined or is invalid.

In all other situations a component is displayed in the default colour: black for white and grey backgrounds or white on a black background.

Each colour scheme can assign one of six colours to a component: RED, GREEN, BLUE, CYAN, MAGENTA and ORANGE. If a component falls outside of the rules for a colour scheme it will assign a default colour of black (on a white or grey background) or white (on a black background).

Two generic types of colour scheme are identified:

1. Simple schemes where components are coloured according to the value of a single attribute or result.
2. Complex schemes where components are coloured according to some logical combination of one, two or more attributes and results.

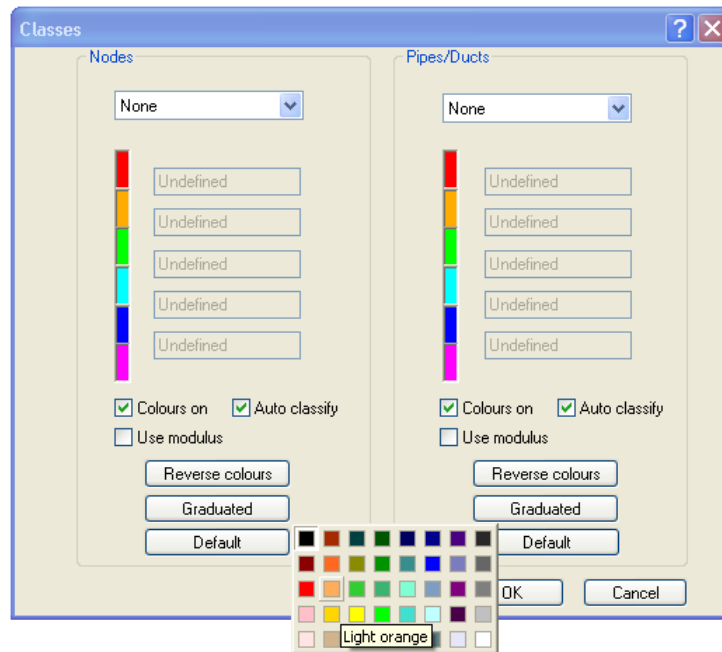
Simple Colour schemes

With simple colour schemes, the user selects a component attribute or result, for example, pipe length, pressure difference, node elevation, and then creates a scheme by associating the selected attribute or result with a set of intervals. The intervals are defined by five values v_1, v_2, v_3, v_4, v_5 , the intervals being:

$< v_1$	RED
$\geq v_1 \ \& \ < v_2$	GREEN
$\geq v_2 \ \& \ < v_3$	BLUE
$\geq v_3 \ \& \ < v_4$	CYAN
$\geq v_4 \ \& \ < v_5$	MAGENTA
$\geq v_5$	ORANGE

If intervals are not defined they will be provided automatically based on a suitable scaling of the known values for the attributes or results. Note for results the intervals are calculated on the completion of a calculation.

Colour schemes are displayed and edited via the Classes dialog.



This consists of two columns, one for nodes and the other for pipes/nozzles. At the head of each column is a combo box for selecting the colour scheme. Note these two combo boxes have a dual purpose in also selecting which attribute is displayed above a component in the schematic.

Below the combo box are five edit boxes used for defining the six intervals. The coloured boxes alongside the edit boxes define the colours for the six intervals. The edit boxes will be grayed out if the selected colouring scheme does not use intervals.

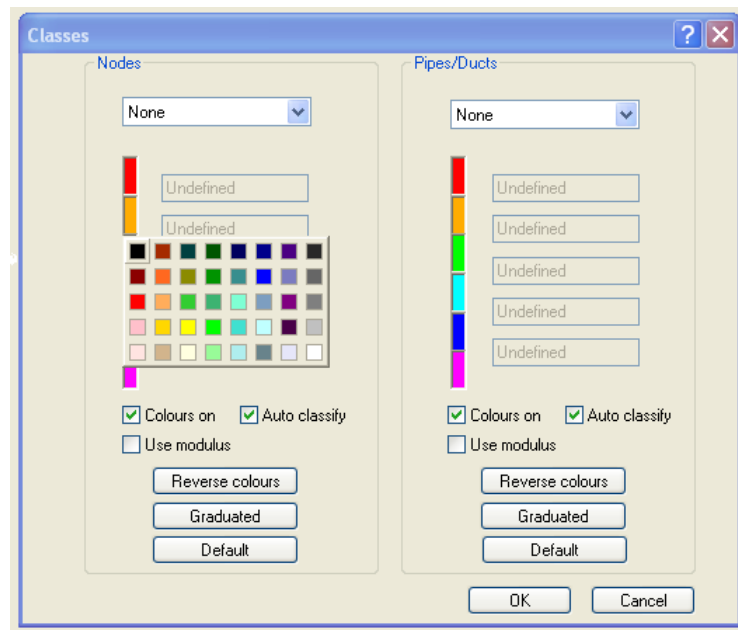
Immediately below each column of five edit boxes are two check boxes:

- **Colours On** - used to enable/display colours. If this box is unchecked then the default colouring scheme will be used. However, the selected attribute will still be displayed when the Display Attributes buttons are selected.
- **Auto classify** - used to select automatic calculation of ranges following completion of a calculation. If this box is checked then the five values defining the six intervals will be re-calculated each time a calculation is performed.
- **Use modulus** - if this is selected then classification will not take the sign of the attribute into consideration.

The final three buttons are used to select the way in which the intervals are coloured, the default being the six colours ordered as RED, GREEN, BLUE, CYAN, MAGENTA and ORANGE. The ordering of the colours may be reversed by selecting the Reverse colours button. As an alternative to the six colours a single colour may be selected to be represented in six shades. Selecting the Graduated button displays a pop-up window via which the colour can be selected.

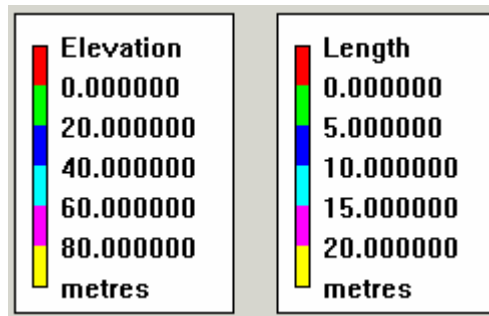
Finally, it is possible to change the default colours by left-clicking on one of the coloured boxes, when a colour selection pop-up appears. For example, in the following image the

user has left-clicked on the orange box.



Selecting a colour from the pop-up will result in the selected colour replacing the colour in the box.

The two combo boxes for selecting the colour scheme (and hence the attribute to be displayed on the schematic) are reproduced on the [Options Toolbar](#). The Options toolbar can also be used to display legends (one for nodes and one for pipes) on the schematic. For example:



The legends can be moved around by clicking and dragging with the mouse.

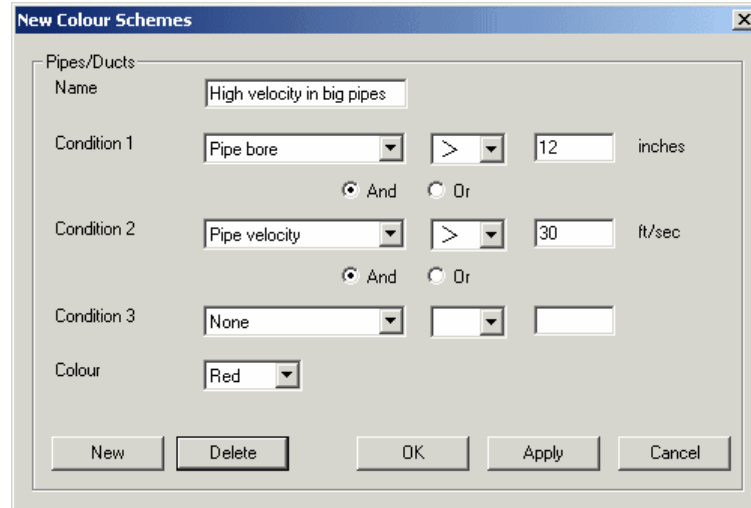
Values are displayed with the correct sign for directional components; negative if the flow is in the opposite direction to the component direction, or positive if it is in the same direction.

Note for pipes, results are shown on the Schematic or in the Properties Window with the correct sign; negative if the flow is in the opposite direction to the component direction, or positive if it is in the same direction. For colour coding the absolute value is used, for example, +10 m/s will be coded using the same colour as -10 m/s. This means that the

interval values specified in the dialog should be zero or positive.

Complex rules

Complex rules allow the user to define a colouring rule in a more flexible manner. This is best described by reference to the dialog used to enter a complex colouring rule:

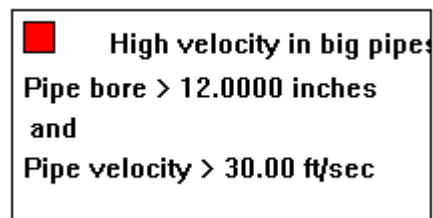


This shows the dialog for entering a complex rule for a pipe, that for a node has the same layout. It comprises:

1. A name for the rule.
2. Up to three conditions (only two are used in this example) consisting of an attribute or result name (selectable from a drop down list), a relational operator (again selectable from a drop-down list) and a value.
3. Logical And or Or operators relating the conditions.
4. The colour to be used for display.

The above example states that each pipe with a bore greater than 12 inches, where the calculated velocity in the pipe exceeds 30 ft/sec, is to be coloured red.

The legend window can be selected for display as with simple rules, for the above example it would appear thus:



13.2 Tagging

In addition to the default colour scheme, one additional tagging colour scheme is provided which is common to all application modules. This scheme can be used to colour components based purely on user selection. This operates as follows:

1. The user selects components in the normal way
2. The user then tags these selected items by selecting the menu option **Tools | Tag Selected Items**.
3. Steps 1 and 2 can be repeated, each Tag-Selected-Items operation adding to the set of tagged items.

Now if the tagged colour scheme is selected, tagged items will be displayed in CYAN (this colour may be changed via the **Options | Display Options** menu item.

Tagged items can be made untagged by selecting the menu option **Tools | Remove Tags**.

13.3 Background Colours

By default the schematic is displayed on a light grey background, this is probably the best choice if component colouring is used. Other background colours can be selected via the **Options | Display Options** menu item.

Regardless of the chosen background colour, the schematic is always printed on a white background.

Elevation Profile and Hydraulic Grade Line

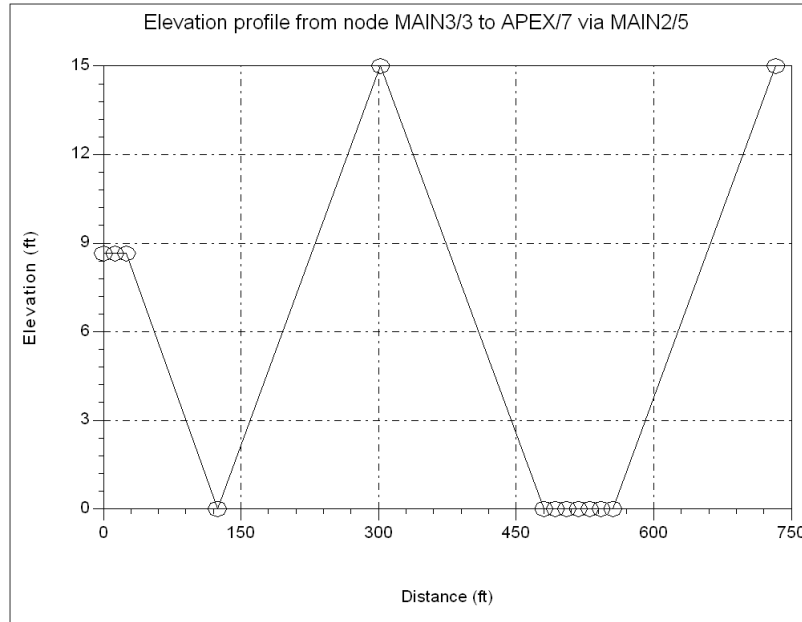
Part

14

14 Elevation Profile and Hydraulic Grade Line

14.1 Elevation Profile Window

This window can be selected for display via the Graphs tab in the Browse Window.



Before a profile can be plotted a path must be defined using the **Tools | Make Path** menu option. To use this, simply select two or more nodes and then select the **Tools | Make Path** menu option to select all components joining the selected nodes. To select more than one node at a time, simply hold down the **Ctrl** key down whilst selecting the nodes.

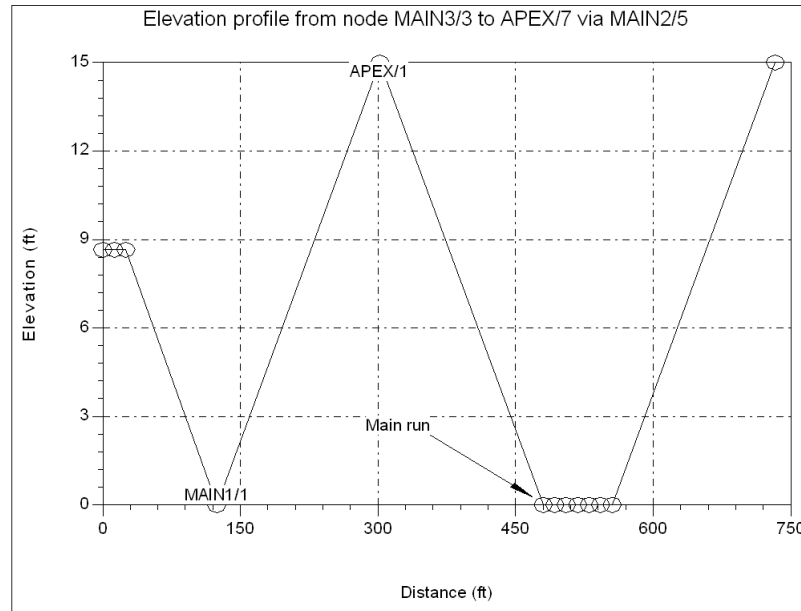
The path found is the shortest path, where shortest means the smallest number of nodes. Having created a path, the elevation profile will be displayed with the nodes plotted from the leftmost of the two nodes selected to the second, rightmost node. The vertical axis displays an elevation scale in the user-selected units and the horizontal axis the distance as measured from the starting node.

Right-clicking with the mouse displays a popup menu with the following options:

- Show values - selecting this option will display the value at a point in a bubble tool-tip.
- Label Point - if the mouse is position on or close to a node this will label the node.
- Add Text - add text annotation to plot
- Add Arrow - add an arrow to plot
- Add arrowed text - add arrowed text to plot
- Copy - copies the plot to the clipboard, from where it can be inserted in, for example, a Word document.
- Edit properties - selecting this option will display a tabbed dialog, via which it is

possible to edit the title, labels, styles and the axes. These properties can be saved as a template.

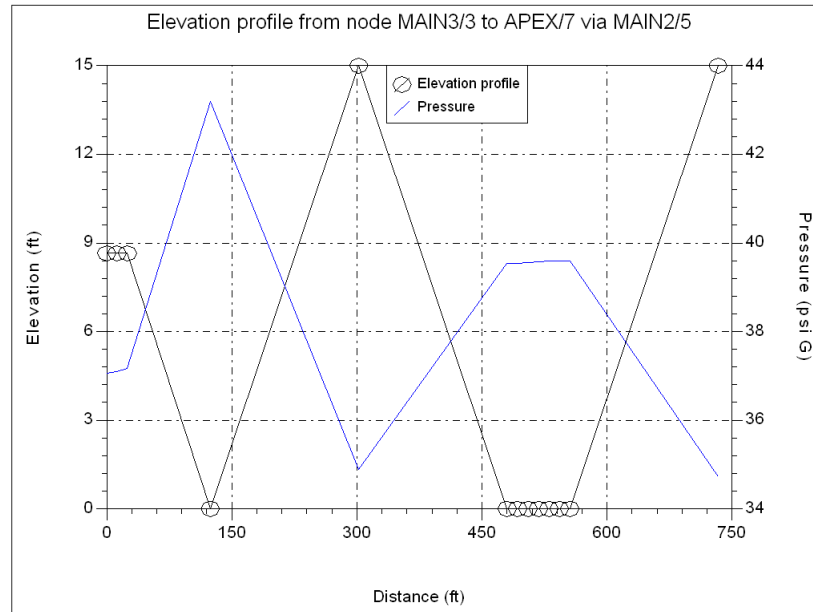
The following figure shows the same plot with some nodes labeled and an arrowed text item:



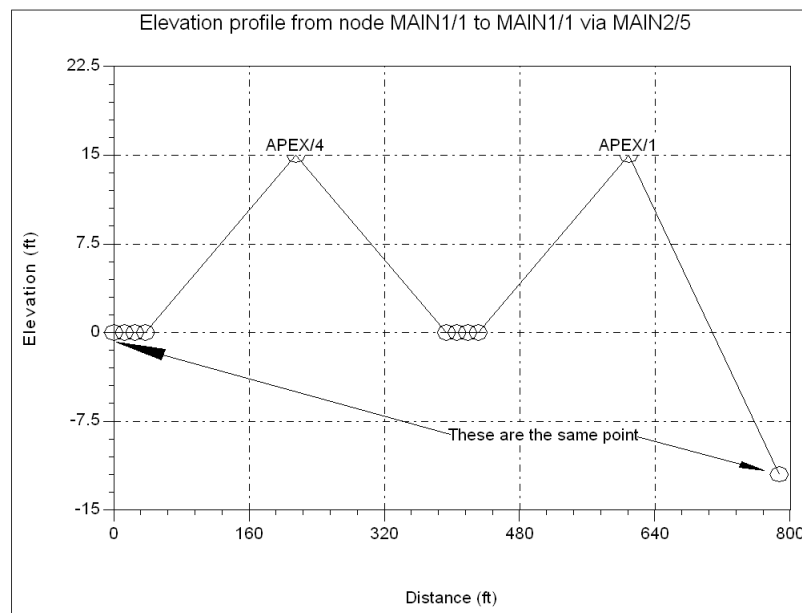
The Elevation profile has a number of uses, the major two being:

1. The display of a [Hydraulic Grade Line](#).
2. Locating [height check errors](#) reported in the [Status Window](#).

For the first of these, the following shows the elevation profile, after a calculation has been performed, with the hydraulic grade shown.



In the second of these, clicking on the line in the Status Window, displaying a height error, will highlight the loop in the network, as well as making a closed path. The following depicts the profile of a height check error (with labeled nodes) which clearly indicates the error. Note the first and nodes shown on the plot are the same physical node and hence their differing heights on the plot illustrates the scale of the error.



14.2 Hydraulic Grade Line

The Hydraulic Grade Line (GL) is the sum of the static head, and elevation head., that is:

$$HGL = SH + EH \text{ \{with units of length\}}$$

If the static pressure is known then we can also use the relation :

$$HGL = \frac{P}{\rho g} + Z$$

where P is the static pressure, Z is the elevation, and ρ the density.

The Hydraulic grade line can be displayed in the elevation profile window, following a calculation by selecting <defined later>.

Multiple Pipe Generation

Part

15

15 Multiple Pipe Generation

15.1 Multiple Pipe Generation

This facility is activated via the Tool menu and provides a quick and simple method of generating pipe runs, typically as used in pipeline applications. Here the user may start with a set of distance and elevation pairs, possibly in a spreadsheet form.

Selecting the tool via **Tools | Add Multiple Pipes** option produces the dialog:

Edit common pipe data:			Create [2] Pipes		
Tag	<NO TAG>		No.	Distance (m)	Elevation (m)
Diameter	Unset	mm	1	1	0
Length	1	m	2	2	0
Elevation	0	m			
C-factor	120				
Status	Normal				
Type	Schedule 80				
Design group	Unset				

Draw in profile

OK Cancel

The left-hand window contains the attributes to be set for each pipe created when the **OK** button is selected, the right-hand window the set of distance-elevation pairs. Above this right-hand window is the number of pipes to created, which can be changed using the up-down buttons to the right of the displayed value. The distance is incremented automatically by the length of the pipe specified in the left-hand window. Values in the right-hand window can be edited.

The check box at the bottom of the dialog indicates that the pipe run will be displayed in profile with pipes being drawn to scale according to their lengths. If this box is unchecked then the window appears thus:

0 ° angle of inclination

No.	Distance (m)	Elevation (m)
1	1	0
2	2	0
3	3	0
4	4	0

OK Cancel

Here the pipe run is displayed in plan at a specified angle of inclination, angles being measured anti-clockwise from the horizontal. Selecting the option **Use proportional lengths** draws the pipes to scale according to their lengths.

Data can be copied from a spreadsheet and pasted into the right-hand window, the number of rows being set automatically from the number of data pairs copied. This facility must be used with caution since the data from the spreadsheet must be arranged in the same column order as that used in the right-hand window, also the units must agree with those in the column headings. [Undo/redo](#) is available with this facility.

Autolayout and Nozzle Generation

Part

16

16 Autolayout and Nozzle Generation

16.1 Autolayout

Creating sub-networks using the Autolayout tool

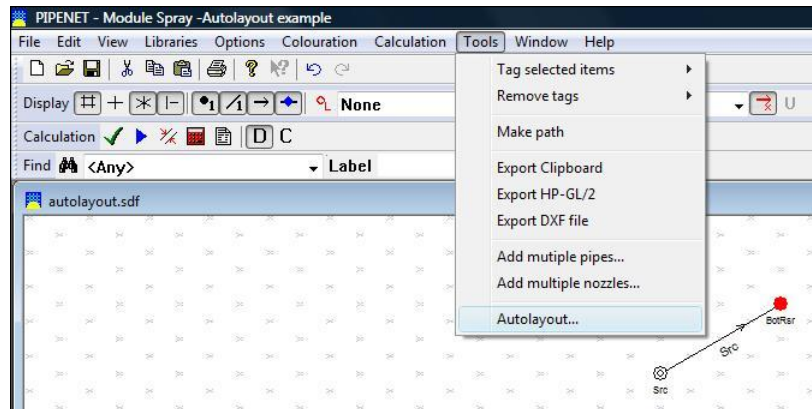
The Autolayout tool, which is available in the PIPENET spray/sprinkler module, can be used to automate the task of designing spray/sprinkler systems for vessels or for area protection.

Before using the tool

Before using the tool make sure that you have [defined the nozzles](#) you intend to use as library items and that you have provided, for each nozzle type, the spray divergence angle. Also ensure that you have at least one [pipe type](#) defined.

Launching the tool

Select Autolayout from the Tools menubar item:



If an existing node is selected at this point, the Autolayout tool will offer to join the new sub-network to that node.

First dialog - defining the geometry and requirements

When the tool is launched, the first of two dialogs will be displayed:

Geometry of protected object

Select the object type from the drop-down list, and enter dimensions for the object or area, currently the options are:-

- Vertical tank with a specified diameter and height
- Rectangular area with a specified length and width.

Spray requirements

Select the “**Compliant with**” radio button in order to opt for compliance with a given fire authority standard (and then choose the required standard from the drop-down list): select the “**User-defined**” radio button in order to enter the spray-density and nozzle-configuration requirements manually (and then enter the required values). For vessels, there are three requirements as shown above: for area protection there are only two – areal flow rate and maximum distance between nozzles.

Currently the defined authority option is limited to NFPA-15(7.4.2), that is, as defined in [\[NFPA15\]](#), sub-section 7.4.2 Vessels.

Schematic display

Select the required orientation and range pipe shape from the drop-down lists. The orientation is that at which the created sub-network will be drawn on the schematic display:-

- NE - north-east, i.e. from top left.
- SE - south-east, i.e. from bottom left.
- SW - south-west, i.e. from bottom right.

- NW - north-west, i.e. from top right.

There are two options for the range pipe shape – “**Straight**” and “**Open rectangle**” – and this option is only applicable for vessels.

Pipes

The topmost parameter in this section specifies either length of the pipe(s) which will join the riser to the range pipe(s) (for vessels), or the height of the riser (for area protection). Enter the required value, and select the pipe type and diameter from the drop-down lists. Leave the pipe diameter “Unset” in order to have the pipe sizes determined automatically following creation of the sub-network. If no pipe types have been defined, the pipes will be created with unset type and diameter.

Labeling

Enter tags for the pipes, nodes and nozzles which are to be created. If the “**Label components by range**” check-box is checked, the labels of the generated components will be numbered according to the ranges with which they are associated, based on the specified tags: for example, if the nozzle tag is “NOZZ” and this option is enabled, the third nozzle on the second range will be labeled “NOZZ2/3”.

When the required details have been entered in the first dialogue, click on the “**Next**” button and the second dialogue will be displayed.

Second dialog - optimizing the design

This dialog is intended to make it very easy to optimize the design and to draw comparisons between the different available nozzles, according to whether you want to minimize total flow, pressure, or number of nozzles.

Autolayout - step 2 of 2

Nozzle Type	Nozzles/range	Pressure	Flow/nozzle	Total nozzles	Total flow	Overlap
		psi A	US.gal/min		US.gal/min	ft
BETE-N3W	27	34.69595	15.37	81	1245.27	0.00103
BETE-N4	69	34.69595	22.44	207	4645.52	0.00654
BETE-N5	27	34.69595	30.75	81	2490.55	0.00103
HOSEREEL						
BETE-N2	11	162.56174	21.42	33	706.86	4.86479
Wall Drench-4	27	36.45161	18.85	81	1526.52	0.00103

Selected nozzle parameters

Spray cone angle (deg)

K-factor (US.gal/min.psi^{1/2})

Minimum/maximum pressure (psi A)

Minimum/maximum flowrate (US.gal/min)

Nozzle configuration

Nozzle-to-vessel distance (ft)

Minimum spray overlap (ft)

Number of sprinkler levels

Back Finish Cancel

The nozzle grid

At the top of the dialogue is a nozzle grid with an entry for each library nozzle. Each entry shows the pressure, flow rate and number of nozzles needed to fulfill the design requirements using nozzles of the given type. For vessels, the actual horizontal spray overlap which will be achieved in practice is also shown. The information in the grid will be recalculated should any of the design parameters be changed in the “Nozzle configuration” section at the bottom of the dialogue (which is only applicable for vessels). If a given type of nozzle is not capable of fulfilling the design requirements (for example, because the spray divergence angle has not been set or is too narrow to achieve the minimum overlap), its entry in the grid will be disabled. Note that the spray divergence angle is only relevant for vessels since the requirements for area protection are concerned with maximum nozzle spacings rather than minimum spray overlaps. To select a nozzle, simply click on the appropriate entry in the grid.

Selected nozzle parameters

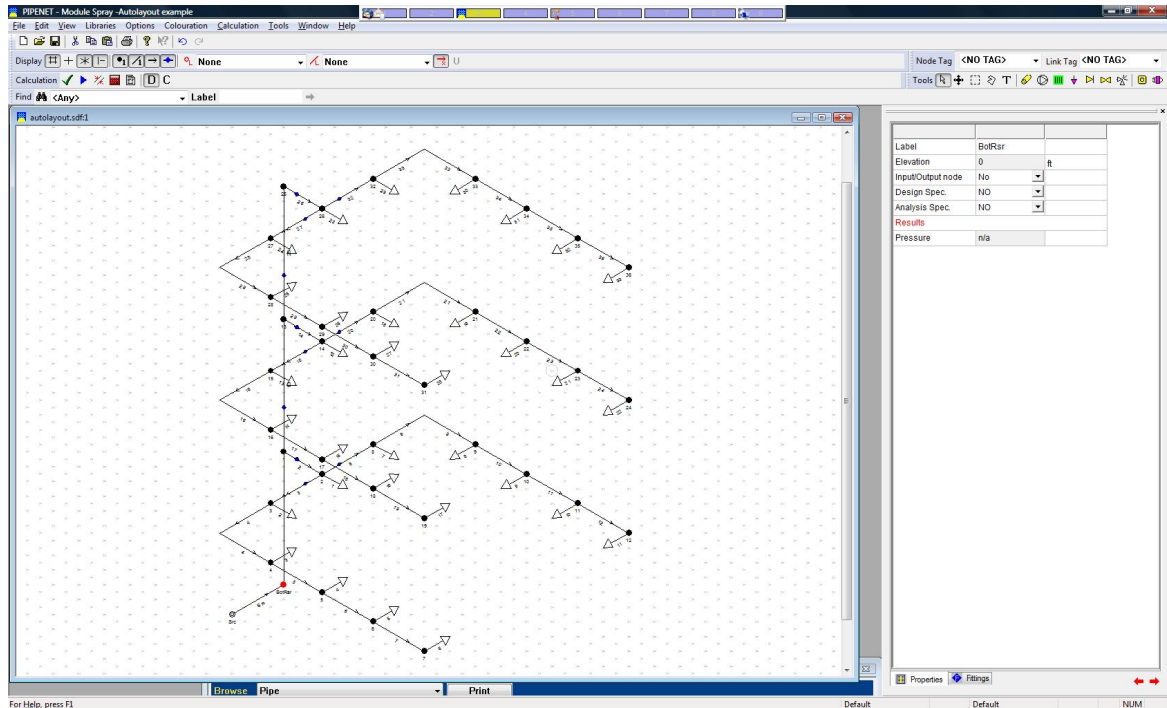
This section simply displays the parameters of whichever nozzle is selected in the grid above.

Nozzle configuration

This section allows some of the design parameters to be changed, within the constraints imposed by the requirements specified in the first dialogue. Following a change to any of these values, the information in the nozzle grid will be recalculated. Note that this section is only available for vessels: the parameters are not applicable to area protection.

Completion

Having adjusted the nozzle configuration values interactively and selected the nozzle type from the list, click on the “**Finish**” button and the sub-network will be drawn on the schematic display ready for further editing:



All the generated pipes have appropriate elevations and fittings, and pipe-sizing can be performed in the usual way if the pipe diameter was left at its default value of “Unset” in the first dialogue.

16.2 Nozzle generation

For the Spray/Sprinkler module a tool is available for generating either

- a run of n nozzles separated by pipes of equal length l , or
- generating n nozzles equally spaced along a distance of l .

Selecting the tool from the tools menu displays a dialog of the form:

The left-hand window contains the attributes of each pipe, the right-hand window the attributes of each nozzle. With the check box in the last row of the left-hand window unchecked, clicking **OK** will generate n nozzles (in the example above 10 nozzles are created) separated by pipes of the specified pipe length. If the check box is checked then the dialog changes to:

Here the length field is now grayed out in the left-hand window since the separation of the nozzles is determined by the number of nozzles and the total run length as specified in the last row of the left-hand window.

The pipe run can be drawn at an angle by providing an angle of inclination (bottom left-hand corner of dialog) measured in degrees counter-clockwise from the horizontal.

Part

17

17 Libraries

17.1 Libraries

Two types of library are used in PIPENET modules:

Local User Library

This library is associated with the data file and is opened when the data file is open. It can contain various items, depending on the module in use,:

- fittings
- pumps
- valves
- nozzles
- linings
- pipe schedules

There is only one local user library. It has the file extension .SLF and replaces all of the separate library files from previous modules. Whilst a data file is open entries can be added, deleted or edited using the Library Editor dialog.

A local user library can be shared by one or more data files, although if the library is changed in one network, it is changed for all users of the library.

Local and system libraries have the same format, the only difference is in their relationship to the data file.

System Library

Whereas the local user library is considered to be under the control of the user opening the data file, and is for all intents and purposes part of the data file, system libraries are external libraries, generally considered to be under the control of some central administrator.

System libraries can be referenced by a data file but are not normally edited whilst a data file is open, instead the user must edit system libraries in an [External System Library Editor](#).

The concept of separate System Libraries introduces extra flexibility in that now a user can have more than one source for schedules, fluids, etc. There also is the possibility of imposing central control over some (System) libraries whilst allowing users to have their own private library definitions.

A network data file referencing a single Local User Library essentially corresponds to the way in which previous products have worked. I.e., in previous products a network data file could only refer to a single PDF file, a single UFL library and so on. However, System libraries provide the facility for a network to gather library definitions from multiple files.

Opening Libraries

Libraries opened from the File menu are always considered external system libraries. Only the new format libraries, that is those with a .SLF extension, can be opened.

Importing libraries

New and old style libraries (i.e. .PDF, .PMP, etc.) can be imported via the Import Library option in the file menu. Imported libraries are merged into the single local user library.

Exporting Libraries

Libraries cannot be exported individually but only as a result of exporting the network as an old style .DAT file. This is achieved by selecting the **File | Export...** menu option and then providing the name to be given to the .DAT file. Any associated library files will be saved with the same file name but with the appropriate file extension.

Libraries - example 1 - new data and library file

In this example we assume that a new data file is being created and no library files exist.

1. Create a new project.
2. Define the library items using the Library Editor dialog.
3. Create the network.

Libraries - example 2 - new data file using old style library files

In this example we assume that a new data file is being created but it is required to import library files created under a previous version of the PIPENET.

1. Create a new project.
2. Go to the File menu and select the option Import library.
3. Use the file dialog to select the library to open, that is a .PDF, .PMP, etc. file.
4. Open the file - the file is imported into the local user library.
5. Repeat steps 2 and 3 for each old style library to be imported.
6. Create the network.

Libraries - example 3 - old data file using old style library files

Opening an old style .Dat file automatically imports any old style library files referenced by the .Dat file.

17.2 Spray Library Editor

The Library Editor comprises a number of pages, one for each type of library item. For the Spray/Sprinkler module these are:

Schedules - Define or edit pipe schedules.

Nozzles - Define or edit nozzles.

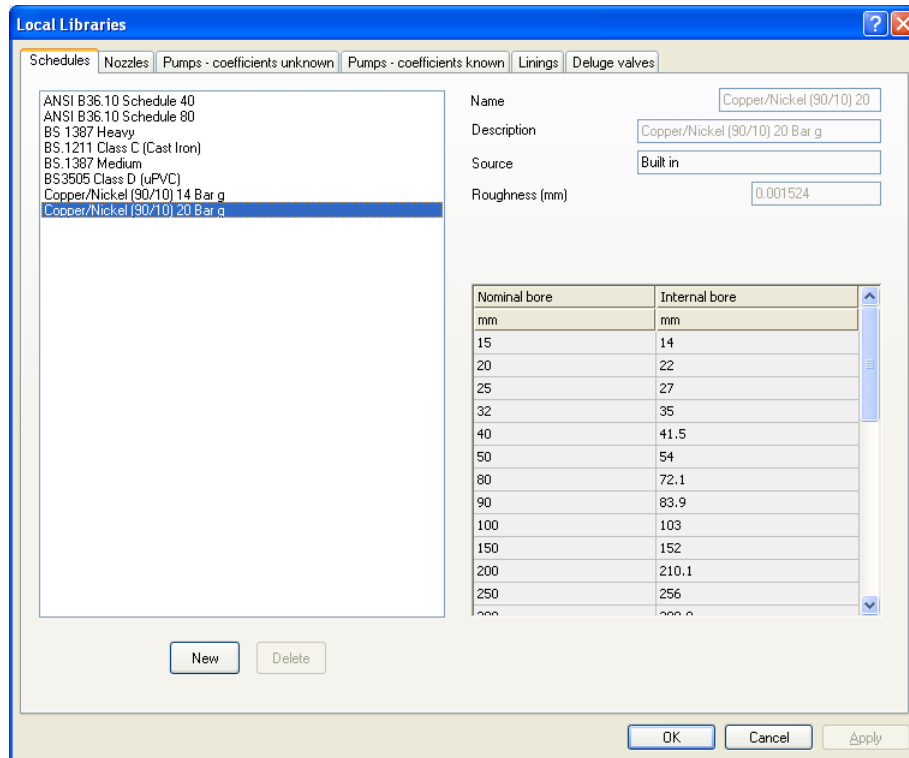
Pumps - coefficients unknown - Define or edit pumps with unknown coefficients.

Pumps - coefficients known - Define or edit pumps with known coefficients.

Linings - Define or edit pipe linings.

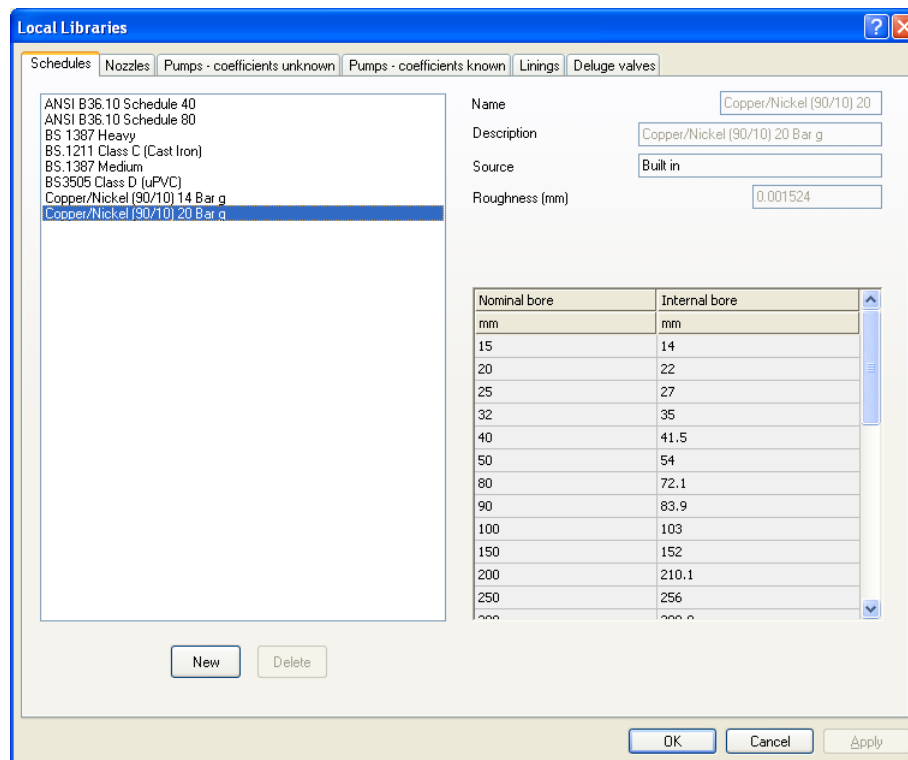
Deluge valves - Define or edit deluge valves.

Selecting any of these options displays a tabbed dialog with the appropriate library edit selected:



17.3 Pipe Schedules

This library page is used for creating and editing pipe schedules:



To the left is a list of available schedules, both built-in and user-defined, selecting any item in this list displays the properties of the schedule on the right.

At the top right are three fields common to all library editors:

1. The name of the schedule as it appears in pop-up menus, the length of this name is limited to 20 characters.
2. An optional longer description.
3. The source of the schedule, which may be one of the following:
 - Built-in schedule provided with the Spray/Sprinkler module. Built-in schedules cannot be edited.
 - Local user library - these items may be edited whilst a network is open.
 - System library - these items can only be edited when a network is not currently open.

Below these three fields is a field containing the roughness, and below that a grid showing the standard nominal sizes and the corresponding internal diameters. If the nominal diameter is "unset" then the corresponding nominal diameter is not included in the schedule.

Nominal diameters are greyed-out indicating that their value is fixed. Right-click within the grid area to display an option to copy the grid contents to the clipboard.

Roughness and diameters are displayed in the user specified units (see [Options - Units](#)).

Adding a new schedule

To add a new schedule select the **New** button in the bottom left-hand corner of the dialog. Enter the desired values (if the Description field is left empty it will by default be the same as the schedule name). When all of the data has been entered click the **Apply** button to accept the new schedule or the **Cancel** button to abort.

Editing an existing schedule

Select the schedule to be edited from the left-hand window, enter the desired changes and then select the **Apply** button to accept the changes.

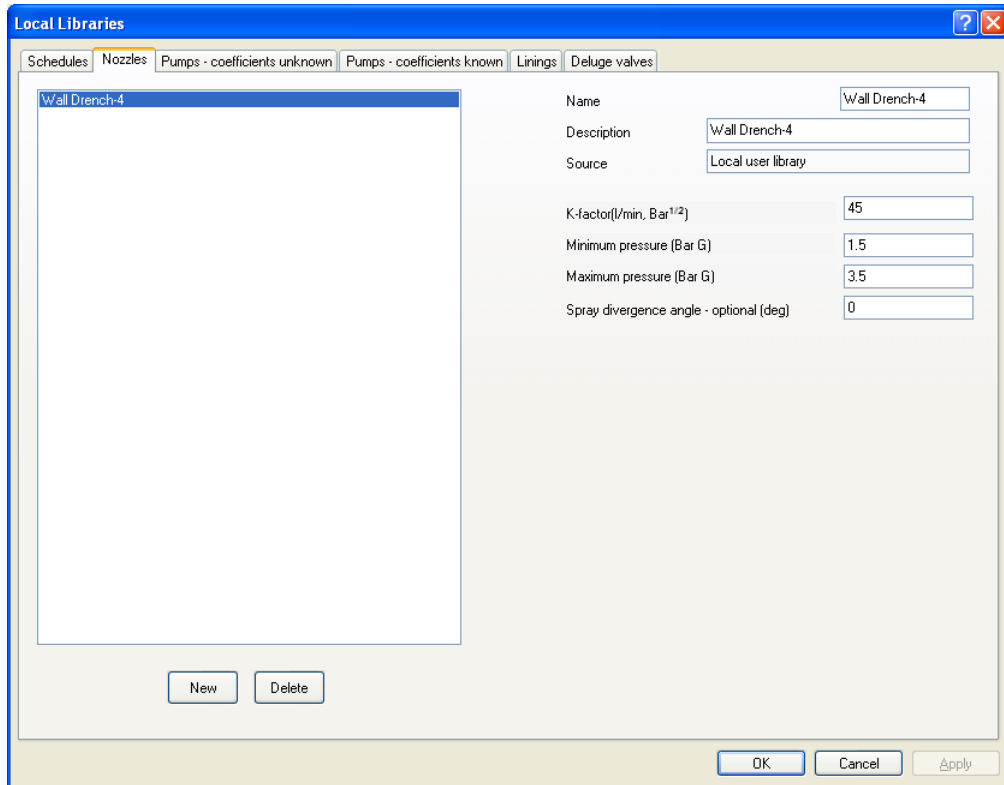
Deleting an existing schedule

Select the schedule to be deleted in the left-hand window and then the **Delete** button.

17.4 Nozzles

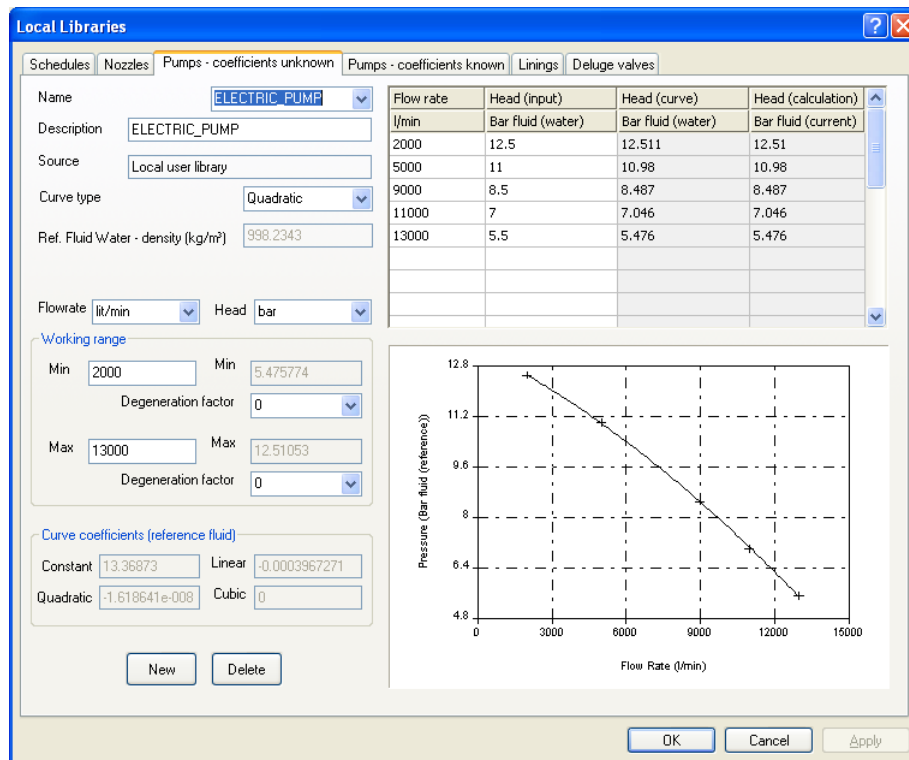
To view a library nozzle, select the required nozzle from the left-hand window and its properties are displayed on the right-hand side. To delete a nozzle, select the nozzle in the left-hand window and then select the **Delete** button. To add a new nozzle:

1. Select the **New** button.
2. Provide a unique name for the nozzle and an optional description.
3. Enter the values for K-factor, Minimum pressure and maximum pressure.
4. If [autolayout and nozzle generation](#) is required enter the spray divergence angle.
4. Select the **Apply** button to add the nozzle to the library.



17.5 Pumps - Coefficients Unknown

This dialog is used to define the characteristics of a pump when the pump coefficients are unknown:



Creating a new pump

To create a new pump:

1. Select the **New** button
2. Input pump name and an optional description
3. Select the desired curve type- Quadratic (default), Cubic or Smooth (cubic spline)
4. Select the desired flow rate and pressure units from the drop-down lists provided at the top left-hand corner of the dialog
5. Provide a minimum and maximum flow rate
6. Provide the two degeneration factors (defaults 0.0) for a Quadratic curve that modify the curve slope outside the working range
7. Provide a minimum of three points for the curve in the bottom left-hand corner window
8. Select **Apply** to add the pump to the library
9. The coefficients are calculated and displayed, along with the pump curve.

Note that the definition of the pump curve will only be accepted if:

- For a quadratic curve - at least three points are provided for a quadratic and the slope of the calculated curve is negative everywhere between the minimum and maximum values.
- For a cubic or smooth curve at least four points are provided.
- For flow rates between the specified minimum and maximum flows there must be no flow rate that gives no pressure change, that is the performance curve must not cross the horizontal axis.

Right-click within the grid area to display an option to copy the grid contents to the clipboard; right-click within the graph area to copy the graph to the clipboard.

The reference density for the curve is water at normal conditions with an assumed density of 998.2343 kgm⁻³. For units of head such as m the fourth column is the same as the third column. For units of pressure such as psi the fourth column is based on the known density of the fluid, if the density is not known the density of water will be assumed. Note that currently the reference density cannot be changed.

Editing an existing pump

1. Select the pump from the drop-down list presented in the pump name box.
2. Make any changes required to the pump parameters.
3. Select **Apply** to commit the changes.

Deleting a pump

1. Select the pump from the drop-down list presented in the pump name box.
2. Select the **Delete** button.

17.6 Pumps - Coefficients Known

This dialog is used to define the characteristics of a pump when the pump coefficients are known:

The screenshot shows the 'Local Libraries' dialog box with the 'Pumps - coefficients known' tab selected. The dialog contains the following elements:

- Name:** TestKnown
- Description:** TestKnown
- Source:** Local user library
- Ref. Fluid Water - density (kg/m³):** 998.2343
- Flowrate:** ft³/min
- Head:** m
- Working range:**
 - Min: 40
 - Max: 300
- Curve coefficients (reference fluid):**
 - Constant: 18.94
 - Linear: 0.007469
 - Quadratic: -0.00013581
- Table:**

Flow rate	Head (input)	Head (curve)	Head (calculation)
ft ³ /min	m	m	m
40		19.021	19.021
53		18.954	18.954
66		18.841	18.841
79		18.682	18.682
92		18.478	18.478
105		18.227	18.227
118		17.93	17.93
131		17.588	17.588
144		17.100	17.100
- Graph:** A line graph showing Head (m fluid) on the y-axis (ranging from 7.5 to 20) versus Flow Rate (ft³/min) on the x-axis (ranging from 0 to 375). The curve is a downward-sloping parabola.
- Buttons:** New, Delete, OK, Cancel, Apply.

Creating a new pump

To create a new pump:

1. Select the **New** button
2. Input pump name and an optional description
3. Select the desired flow rate and pressure units from the drop-down lists provided at the top left-hand corner of the dialog
4. Provide a minimum and maximum flow rate
5. Provide the coefficients
6. Select **Apply** to add the pump to the library
7. The pump curve is displayed.

Note that the definition of the pump curve will only be accepted if the slope of the calculated curve is negative everywhere between the minimum and maximum values.

Right-click within the grid area to display an option to copy the grid contents to the clipboard; right-click within the graph area to copy the graph to the clipboard.

The reference density for the curve is water at normal conditions with an assumed density of 998.2343 kgm⁻³. For units of head such as m the fourth column is the same as the third column. For units of pressure such as psi the fourth column is based on the known density

of the fluid, if the density is not known the density of water will be assumed. Note that currently the reference density cannot be changed.

Editing an existing pump

1. Select the pump from the drop-down list presented in the pump name box.
2. Make any changes required to the pump parameters.
3. Select **Apply** to commit the changes.

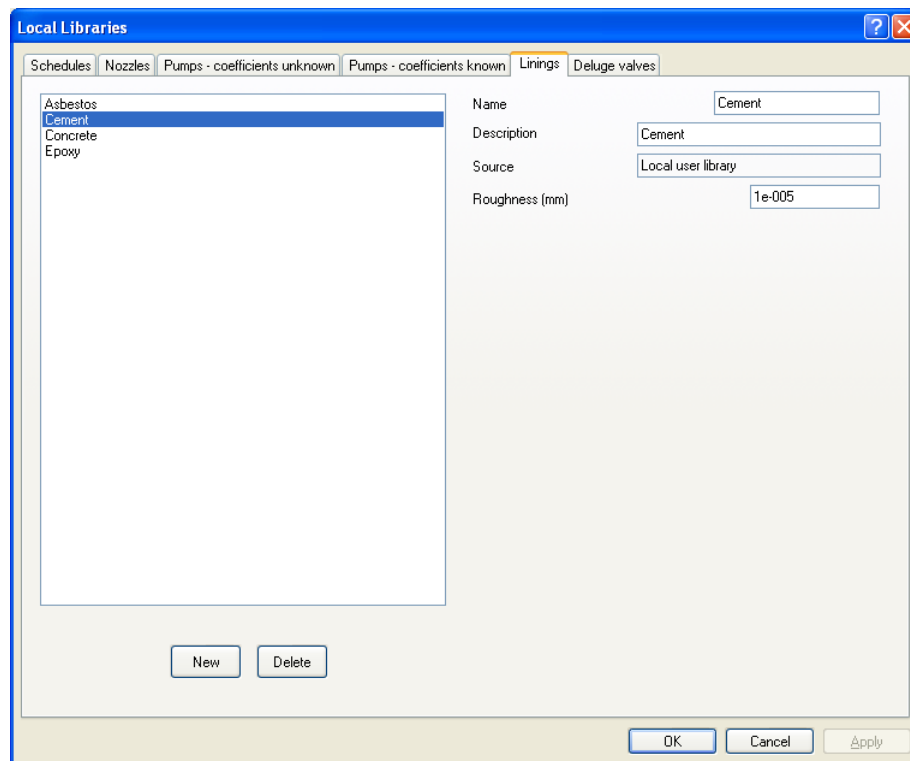
Deleting a pump

1. Select the pump from the drop-down list presented in the pump name box.
2. Select the **Delete** button.

17.7 Linings

To view a library lining, select the required lining from the left-hand window and its properties are displayed on the right-hand side. To delete a lining select the lining in the left-hand window and then select the **Delete** button. To add a new lining:

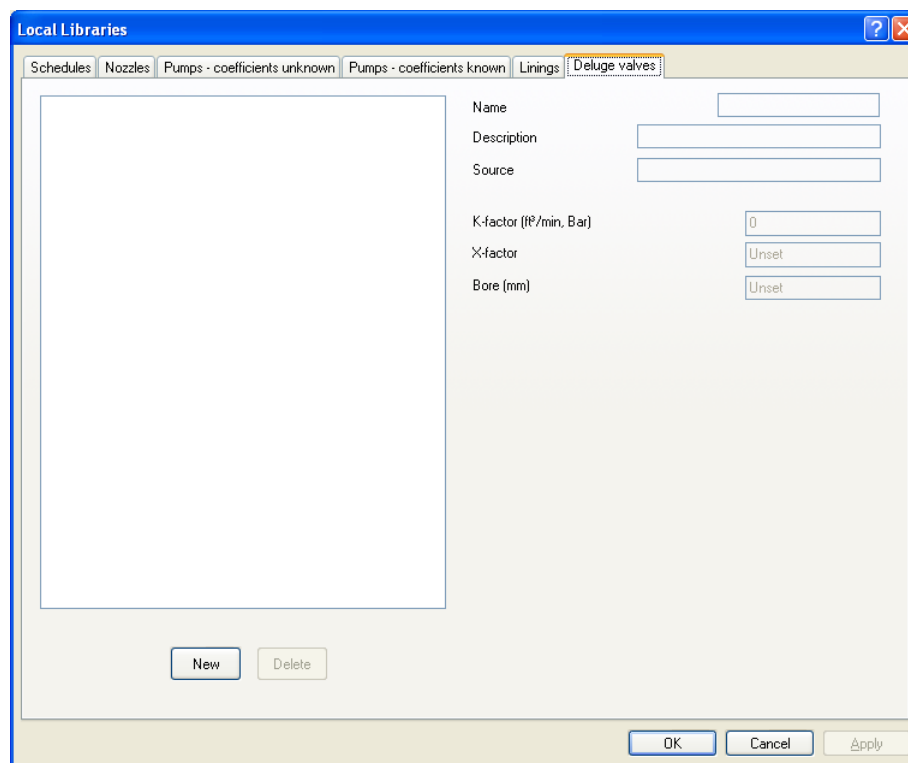
1. Select the **New** button.
2. Provide a unique name for the lining and an optional description.
3. Enter the value for roughness.
4. Select the **Apply** button to add the lining to the library.



17.8 Deluge Valves

To view a library deluge valve, select the required deluge valve from the left-hand window and its properties are displayed on the right-hand side. To delete a deluge valve select the deluge valve in the left-hand window and then select the **Delete** button. To add a new deluge valve:

1. Select the **New** button.
2. Provide a unique name for the deluge valve and an optional description.
3. Enter the values for the K-factor, X-factor and bore (see the [modelling](#) section for further details).
4. Select the **Apply** button to add the deluge valve to the library.



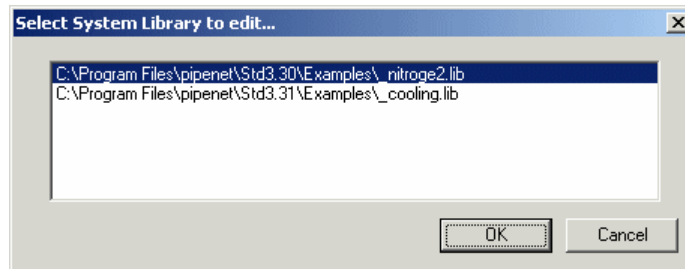
17.9 Editing system libraries

Editing System Libraries is essentially the same process as for editing the Local User Library, using the same dialogs. There are however, some important observations:

- System Libraries can be edited whilst a network referencing the file is open, but this is not generally recommended.
- System Libraries are designed to be shared by a number of users and any edits may affect other users.
- System Libraries will typically be administered by a central controller who would control access to the libraries via read/write permissions.
- Whilst there is only one User Library referenced by a network there can be many referenced System Libraries.

Selecting this option will either:

- Open a library file directly if one and only one system file is referenced by the network, or
- Display a list of referenced library files, any one of which may be selected for editing.



Specifying Options

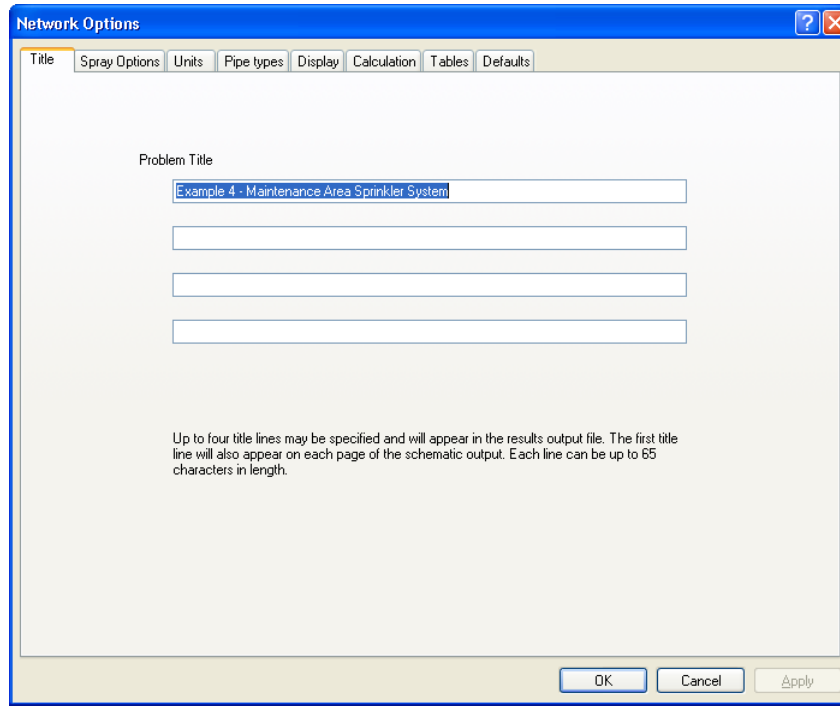
Part

18

18 Specifying Options

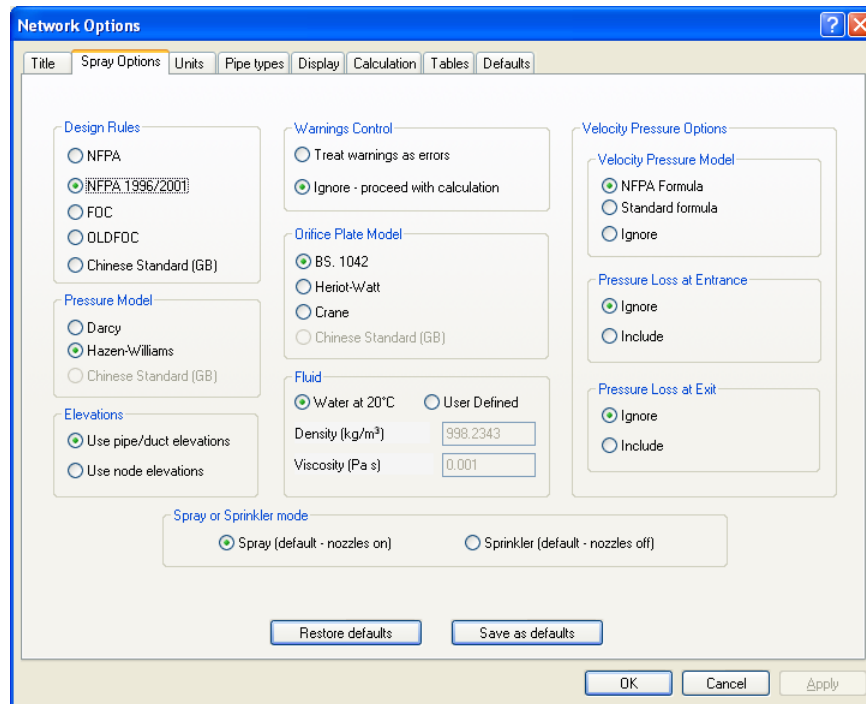
18.1 Title

Use this dialog to enter a title for the PIPENET problem. A title may consist of up to four lines of text, each line no more than 65 characters in length. The first line of the title will be displayed on each page of a printed schematic.



18.2 Spray Options

The options page provides control over a number of modelling and calculation options for the Spray/Sprinkler module:



Design Rules

Select one of the available options:

- NFPA
- NFPA 96/2001 - note that NFPA96 refers to the NFPA edition from 1996. This edition made some changes to the way calculations are made. The equivalent lengths of fittings before 1996 were applicable only to Schedule 40 carbon steel pipes. In 1996 NFPA introduced a table of correction factors for other pipe schedules based on the inner diameter of pipes. This changes the equivalent length of fittings for pipes other than schedule 40.
- FOC
- OLDFOC
- Chinese Standard (GB) - Note if this option is selected then the corresponding Chinese options for the Pressure Model and Orifice Plate model are automatically selected. The support for the Chinese Standard (GB) is based on the [\[CHINGB\]](#)

Pressure Model

Select from the available options, referring to the Modelling section - Pressure Models for further details.

Elevations

Specify means by which node heights are derived.

Use pipe elevations

Each pipe is assigned a change in elevation (rise) from its input to its output. A reference node is selected and assigned a reference height. The height of each node is calculated with reference to the reference node. This option can result in height inconsistencies if a network contains one or more loops. In a loop the sum of the elevation changes must sum to zero. However, if a rise has been incorrectly entered the sum will not be zero and a height check error will be reported. Height check errors can be difficult to locate in large loops.

Use node elevations

The elevation of each node is directly entered as an attribute of the node - height check errors cannot occur with this method.

Warnings Control

The default is to treat warnings as errors. Occasionally, it may be appropriate to allow a calculation to continue even if there are warnings.

Orifice plate model

Select one of the available options:

- BS1042
- Heriot-Watt
- Crane.
- Chinese Standard (GB) - Note, this option is automatically selected when the Chinese Standard option is selected for the Design Rules. It cannot be de-selected other than by changing the Design Rules.

Fluid

Select water at 20°C or select user-defined and provide the values for density and viscosity.

Spray or Sprinkler mode

Select either the Spray option in which all newly created nozzles will be created with the nozzle open, or the Sprinkler option in which case all newly created nozzles are turned off.

Velocity Pressure Options

This includes three options:-

- [Velocity Pressure model](#) - use the NFPA model, the Standard model, or ignore velocity pressure considerations. The NFPA model is that given in [NFPA15] and the Standard model is that given in most textbooks
- **Include pressure loss at entrance** - whether or not the pressure loss should be included for the pipes located at the system entrance
- **Include pressure loss at exit** - whether or not the pressure loss should be included for the pipes located at the system exit

The selection of the velocity pressure options only affects the appearance of the [NFPA submittal report](#).

18.3 Units

A wide selection of unit systems are provided, including:

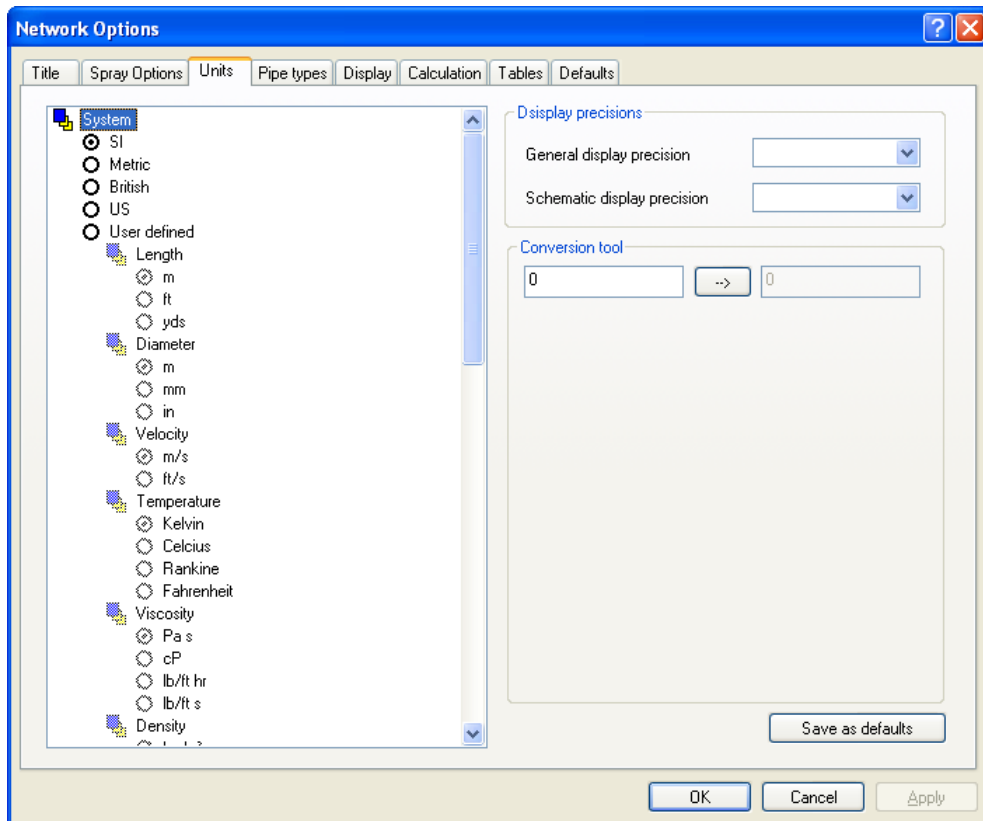
- SI
- Metric
- US
- Imperial
- User Defined

Each of the first four provides a fixed, consistent set of options. For example the unit of diameter (of pipes) and length in the SI system is fixed at metres and the unit of temperature is fixed at Kelvin. The **User Defined** option allows the user to specify the unit to be used for each measure independently. We could, for example, have length measured in metres but diameter measured in inches.

In general it is recommended that the very first operation in the process of creating a new network should be the selection of the appropriate units. Whilst units can be changed later it can lead to complications and confusion to do so. In particular switching between mass and volumetric flow can create problems when the fluid density is unknown (dependent on fluid model being used) and a warning may be issued in these situations.

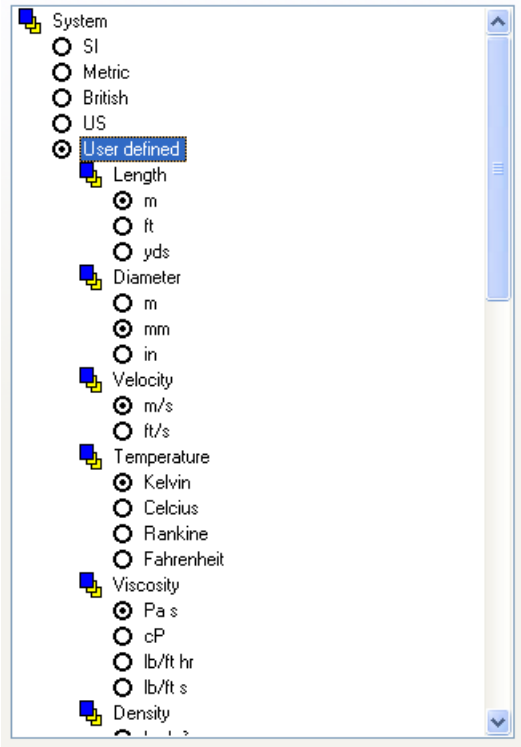
The Units dialog

Selecting the menu option **Options | Units** displays:



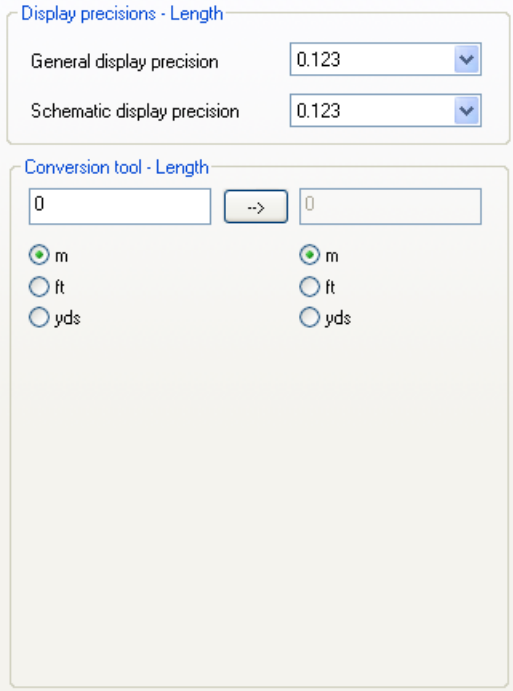
The left-hand side is a scrollable window via which for each *quantity* (length, diameter, velocity, etc) a *unit* (metres, feet, etc.) may be selected. For users of previous versions of PIPENET this window is functionally identical to the Units dialog in those versions; the same set of options is available. The units options window can be scrolled vertically to display more options.

The example above illustrates that SI units are in use and units cannot be changed individually. Selecting the **User-defined** option will result in the following display and all unit options are now enabled:



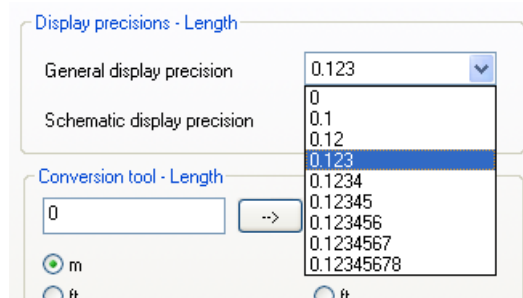
Display Precisions

At the upper right of the dialog is an area where the display precisions can be set individually for general display (in the Property windows, dialogs and the data window) and in the Schematic window. To use this facility select the name of the quantity (Length, Diameter, etc.) in the left-hand window and the display changes to:



Here **Length** has been selected. The values displayed for the precision [**0.123**] indicate that three decimal places will be used for the display of length in the selected unit of length (metres in this example).

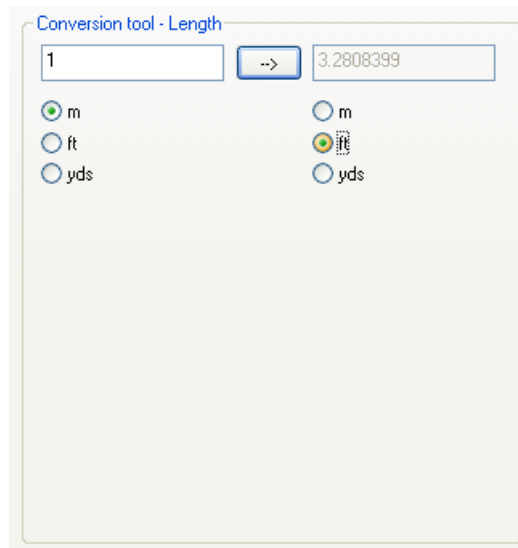
Selecting the arrow to the right of each field produces a drop-down showing that the number of decimal places can be selected to between zero and eight.:



When a network is saved the preferred precisions are saved and they are reloaded when the file is re-opened. Selecting the **Save As Defaults** button will save the current settings in the registry and these will be the defaults used when new networks are created.

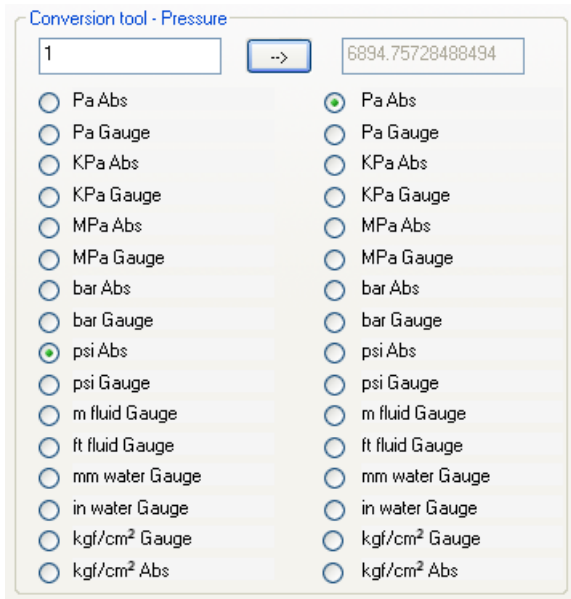
Unit Conversion Tool

At the lower right of the dialog is a unit conversion tool, again this area is only active if a measure is selected

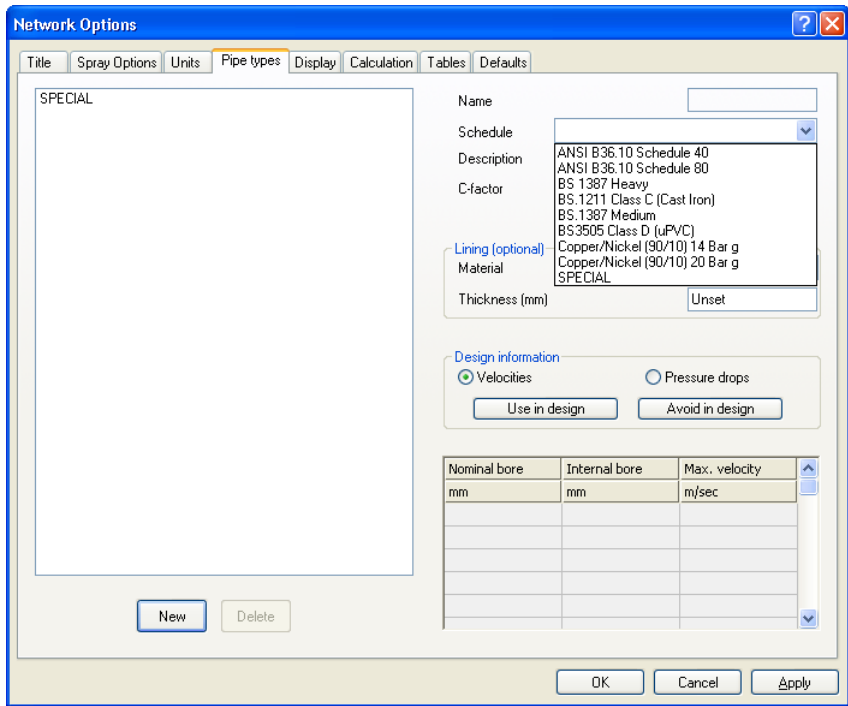


To use the tool simply enter a value in the left-hand field, select the unit that the value represents from the left-hand column (*from* unit) and the unit to which to convert to from the right-hand column (*to* unit) and the result is displayed in the right-hand field. This example illustrates that 1 metre is converted to 3.2808399 ft. The example below illustrates the use of the tool to convert a pressure,

specifically 1.0 psi Abs converts to 6894.75... Pa Abs.



18.4 PipeTypes



In the spray module all pipes in a network must be of a specified type. A pipe type will be associated with a pipe schedule, and this schedule must already exist before the pipe type can be created. An optional lining definition may be provided by specifying the material and the lining thickness.

If no pipe type is defined then the pipe drawing tool on the toolbar is disabled and the

status message **No Pipe Type** appears in the status bar at the bottom of the screen. This status will disappear when a pipe type is defined and the pipe button will be enabled:



Creating a new pipe type

To create a new pipe type:

1. Select the **New** button.
2. Select the associated schedule from the drop-down list provided - a name and description is provided automatically.
3. Provide velocities or pressure drops for all those bores which are to be provided by the schedule, bores being marked as available or unavailable by selecting the bore in the bottom right-hand corner window and selecting the **Use in design** or **Avoid in design** button as appropriate (default is all valid bores are marked as available). Bores for which **Avoid in Design** have been selected are shown with a red background in the velocity/pressure cell.
4. Select **Apply** to complete the creation of the pipe type.

Note that if a pipe is defined as using a pipe type, then instead of explicitly entering the pipe diameter, the diameter is selected from a combo box listing the available sizes together with the **unset** value.

Editing an existing pipe type

1. Select the pipe type from the top left-hand window.
2. make any changes required to the pipe type parameters.
3. select **Apply** to commit the changes.

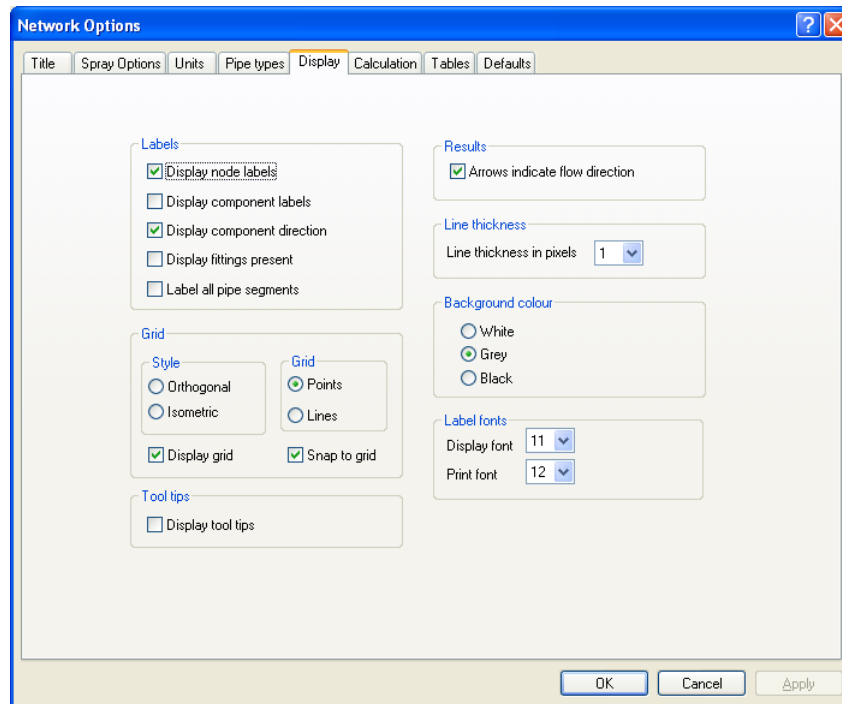
Deleting a pipe type

1. Select the pipe type from the top left-hand window.
2. Select the **Delete** button.

A pipe type cannot be deleted if it is in use.

18.5 Schematic display options

All schematic related display options are displayed on the Display tab. Selecting this option will display a dialog box:



Label Options

The label options allow nodes to be labeled, links to be labeled, component direction to be indicated and the presence of any fittings on a pipe to be shown.

For pipes an additional option is available indicating whether or not all segments of a multi-segment pipe (that is one containing way points) are to be labeled, or only the central segment. Labeling only the central segment may improve the appearance of printed schematics.

An arrowhead pointing from the input towards the output indicates component direction. As with the standard PIPENET definition, component direction does not necessarily correspond to the direction of flow.

The presence of one or more fittings on a pipe is indicated by a blue diamond symbol centred along the length of the pipe.



Results Options

Results options - selecting the option "Arrows indicate flow direction" will display an arrow on each pipe indicating the direction of flow instead of the component direction. Which results are displayed, pressure, flow rate, velocity, etc. is selected via the classes tab of the Properties Window or the [Options toolbar](#).

Line Thickness

Specify the thickness of the lines used for drawing as being between one (default) and

eight pixels.

Grid Options

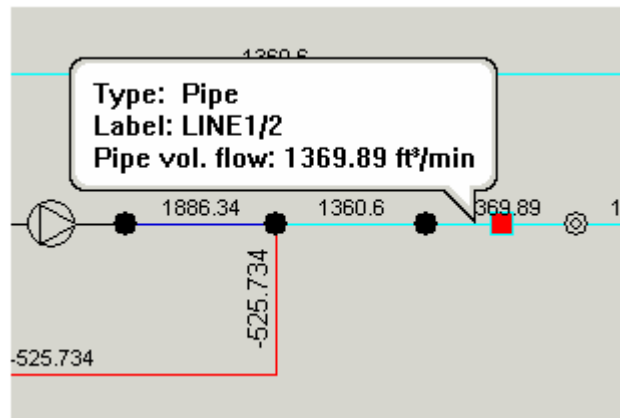
Via the Grid options you can select the display of a grid, its style and whether all nodes and waypoints are confined to lay on grid intersections. The grid can be orthogonal (vertical and horizontal grid lines) or isometric (one vertical axis, one axis at 30 degrees to the horizontal and a third at 150 degrees to the horizontal).

Colours and Fonts

This options are used to select the background colour and the font size for node and link labels.

Tool Tips

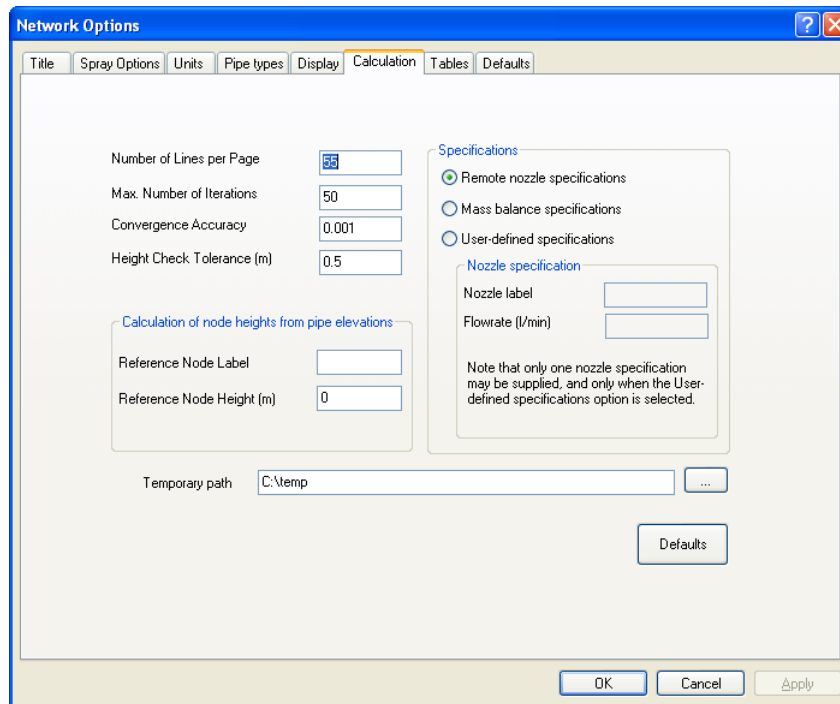
Selecting this option will cause tool tips to be displayed when the mouse cursor is on or near a component. the tool tip will display the component type, its label and the current parameter selected from the Options toolbar. The information is displayed for a few seconds and then will disappear. For example moving the mouse to a pipe will display:



Here the tool tip is displaying the calculated volumetric flow.

18.6 Calculation options

Calculation related options are accessed via the **Options | Display Options** menu option. Selecting this option will display a dialog box:



This page contains a number of calculation related option

Number of lines per page

This is the number of lines per page used in the traditional output file produced by the calculator.

Maximum number of iterations

The default value of 50 should be sufficient for almost all circumstances. If you see the error message "Failed to Converge" then increasing this value may produce results.

Convergence Accuracy

This parameter defines the convergence accuracy used to determine when two iterations are sufficiently close to assume a solution has been reached.

Height Check Tolerance

If pipe elevations are used this is the acceptable error in resolving node heights in loops.

Specification

Select the type of specification required - note the Nozzle label and flow rate fields will only be enabled for User-defined specifications, and only used to supply a single [Nozzle specification](#).

Temporary Path

Enter the path to be used for the storage of all intermediate temporary files required by the calculator.

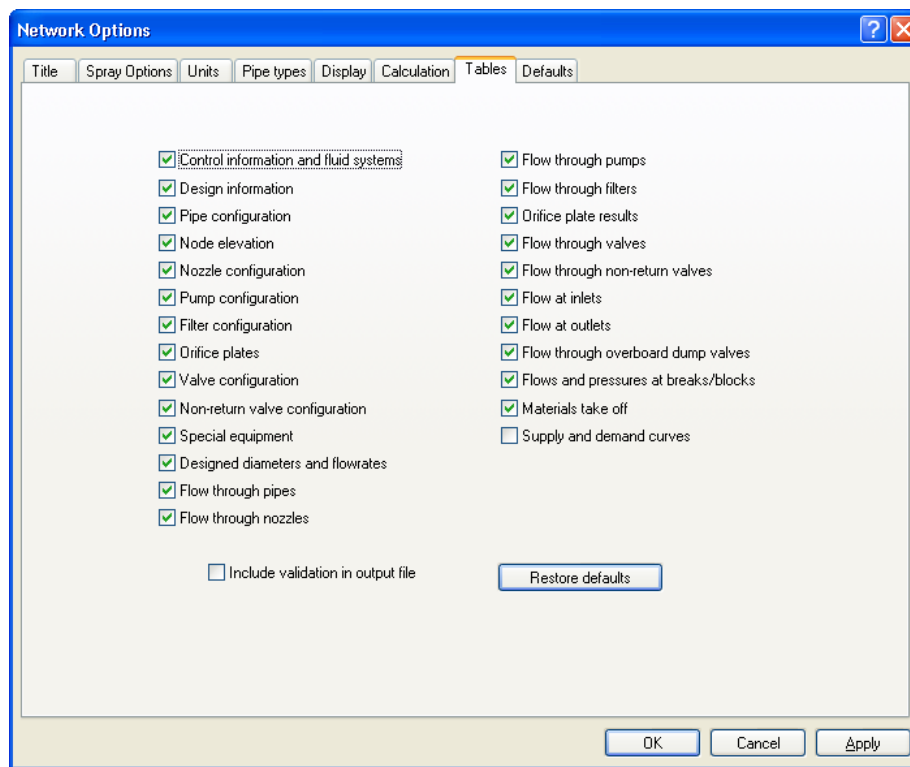
Calculation of node heights from pipe elevations

If pipe elevations are being used (see [Spray Options](#) dialog) then whenever a check or calculate operation is performed the node heights will be calculated from the pipe elevations. Here a reference node, from which all other node heights are calculated, and a reference height can be provided. Note the calculated node heights are only for display in the schematic window they have no effect on the hydraulic calculations. If a reference node is not provided then an arbitrary node will be used and assigned a zero elevation. If node elevations are being used then the two edit fields are greyed out.

The specification of a reference height and the calculation of node heights from pipe elevations simply provides additional useful feedback to the user, the information has no effect on the calculated results.

18.7 Spray Tables

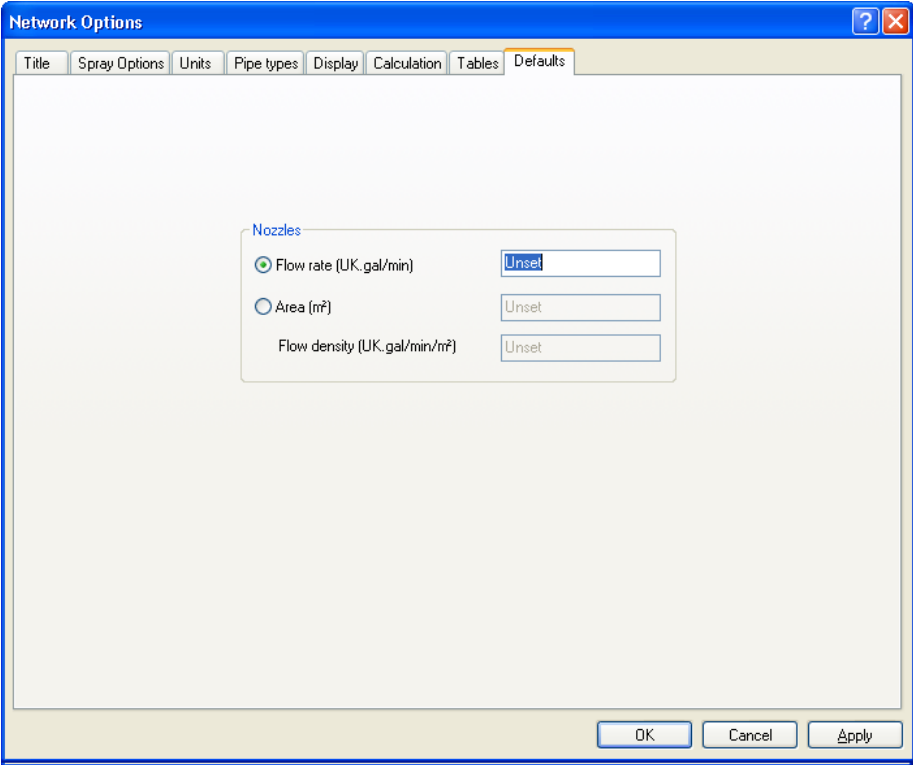
This dialog allows the user to select the tables to appear in the calculation output for the Spray/Sprinkler module.



If the include validation box (lower left-hand corner) is selected, a listing of the original input file will be included at the beginning of the output file. This will display any errors or warnings arising from the input file.

18.8 Defaults

Via this dialog the user can specify the defaults to be applied when creating new nozzles:



Modelling

Part

19

19 Modelling

19.1 Remote Nozzle Specification

This very useful feature allows the user to specify that all nozzles must be discharging at or above their required minimum rate. PIPENET achieves this by identifying the most remote (see below) nozzle in the network and making a flow rate specification on it of its minimum required discharge rate. This counts as one calculation phase flow rate specification. Thus, when using this option the user must supply another $(N_{i_o} - 1)$ calculation phase specifications so that total no. of specifications = no. of I/O nodes (ie $N_f + N_p = N_{i_o}$).

The 'Most Remote' nozzle is the nozzle in the network which is hardest to supply at its minimum rate. Thus, if the most remote nozzle is being supplied at this minimum rate, all other nozzles will be supplying above their minimum rate. Some of the factors which influence the remoteness of a nozzle are:

- The elevation of the nozzle.
- The K-factor of the nozzle.
- The minimum flow rate required through the nozzle .

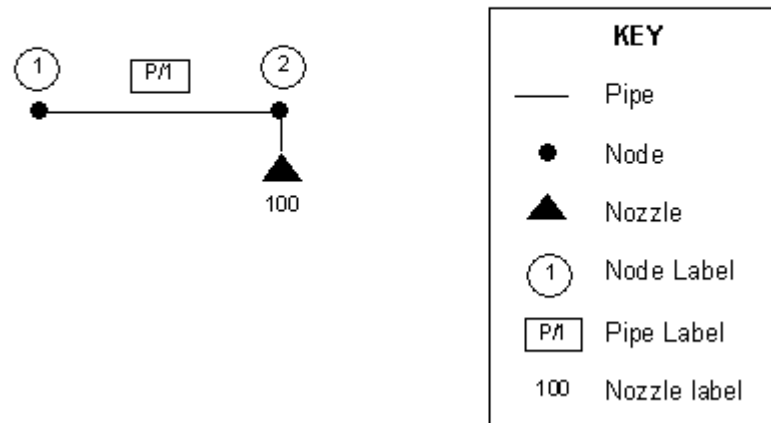
19.2 Mass Balance Specifications

If this option is selected then PIPENET will automatically make all the calculation phase specifications, and the user need only supply the design phase specifications. PIPENET will make a calculation phase flow rate specification on every ionode in the network. The flow rates specified will be those used in the design phase. Thus in effect this option sets the calculation phase specifications to the same as the design phase specifications.

19.3 Design Facility

PIPENET's Design Facility helps the user to design safe networks, which meet given supply demands whilst ensuring that the fluid velocity in each pipe does not exceed a given maximum value (known as the design velocity of the pipe), or that the pressure drop per unit length of pipe does not exceed a given maximum value (the design pressure drop).

As a simple example of this type of calculation, consider once again the example network:



Suppose that the nozzle is required to supply water at a rate of 600 lit/min, but that for safety reasons the velocity of fluid in the pipe must not exceed 4 m/s. (i.e. the design velocity is 4 m/s). Then we have:

Flow rate through pipe = Fluid velocity x Pipe cross-sectional area

and

Fluid velocity \leq Pipe Design velocity.

Using 600 lit/min = 10 lit/s = 0.01 m³/s:

$$0.01 \leq 4.0 \times (0.25 \times \pi \times d^2)$$

or

$$d^2 \geq 0.01/\pi = 0.003183$$

$$d \geq 0.056411 \text{ m} = 56.4 \text{ mm..}$$

Thus in this case the diameter of the pipe should be larger than 56.4 mm. The actual diameter of the pipe used will depend on what sizes of pipe are available. This is determined by the pipe schedule being used.

Given the flow rates required in and out of the network, PIPENET will calculate the flow rates through every pipe in the network. PIPENET will then perform a calculation similar to the one given above, and select a suitable diameter for each pipe.

19.4 Linings

Any of the available pipe schedules (built-in or user-defined) may also be lined in accordance with the user's requirements. This is done by specifying the lining to be used for the Pipe Type, together with the thickness of the lining. Of course, if no lining is to be used then no such data is supplied.

Three types of lining are built into the program, and a further 3 types may be defined by the user in the Private Data File. The 3 built-in linings are:

- Concrete
- Asbestos
- Epoxy

In order to define a lining it is simply necessary to specify the name and roughness of the lining material via the [Library](#) option..

19.5 Pipe Modelling

Modelling Equation

Pipes are assumed to be of uniform circular cross-section.

$$P = P_{\text{fric}} + P_{\text{elev}} + P_{\text{plat}}$$

where:

P_{fric} = Pressure loss due to friction and fittings.

P_{elev} = Pressure loss due to elevation change.

P_{plat} = Pressure loss due to any orifice plate fitted.

Pipe Friction Loss (including fittings)

P_{fric} is found using one of the two methods: the [Coulson-Richardson equation method](#) or the [Hazen-Williams equation method](#).

Data Required

From the library:

The pipe schedule data (if a user-defined schedule used).

In the input file:

1. Definition of the Pipe Type (Schedule, lining, [C-factor](#) and design velocities).
2. Pipe label.
3. Input and output nodes.
4. Nominal Diameter.
5. Pipe length.
6. Increase in elevation from inlet to outlet.
7. A list of fittings on the pipe.

Notes

1. The pipe diameter may or may not be supplied by the user. If the diameter is not set by the user then a suitable size will be found by [PIPENET's Design Facility](#).
2. Increase in elevation may be given directly or by supplying the elevations of the input and output nodes.

Chinese Standard (GB)

With Chinese Standard (GB) options, the pipe pressure loss due to friction and fittings is calculated as follows:

$$P_{fric} = 0.0000107 \frac{u^2}{(D - 0.001)^{1.3}} (L + L_e)$$

where:

- P_{fric} is the pipe pressure loss due to friction and fittings, unit is M pa.
 u is the fluid velocity, unit is m/s
 D is the internal diameter of the pipe, unit is m.
 L is the pipe length, unit is m.
 L_e is the equivalent length of any pipe fittings, unit is m.

19.6 Hazen Williams

The Hazen-Williams equation is an empirical formula which gives an explicit expression for the frictional pressure loss.

SI units

The frictional losses are given by:

$$P_{fric} = \frac{6.05 \times 10^5 (L + L_e) Q^{1.85}}{C^{1.85} D^{4.87}}$$

where:

- P_{fric} is the friction loss in bar
 L is the pipe length in metres
 L_e is the equivalent length of any pipe fittings, in metres
 Q is the (volumetric) fluid flow rate in l/min
 D is the pipe diameter in mm
 C is the Hazen-Williams constant (or [C-factor](#)) for the pipe

Imperial units

The frictional losses are given by:

$$P_{fric} = \frac{4.52(L + L_e)Q^{1.85}}{C^{1.85} D^{4.87}}$$

where:

- P_{fric} is the friction loss in psi
 L is the pipe length in feet
 L_e is the equivalent length of any pipe fittings, in feet
 Q is the (volumetric) fluid flow rate in gallons per minute
 D is the pipe diameter in inches
 C is the Hazen-Williams constant (or [C-factor](#)) for the pipe

19.7 Coulson Richardson Equation Model

The Coulson-Richardson equation is a theoretical equation which gives the pressure in pipes, ignoring frictional effects. By comparing the theoretical results obtained using the Coulson-Richardson equation with those obtained in experiments the pressure drop due to friction effects can be found. Based on the work of the French engineer Henri Darcy(1803-58) the following equations are obtained:

$$P_{fric} = \frac{2f(L + L_e)\rho u^2}{D}$$

where:

- D is the internal diameter of the pipe.
 L is the pipe length.
 L_e is the equivalent length of any pipe fittings.
 f is the Fanning friction factor.
 u is the fluid velocity.
 ρ is the fluid density.

The Fanning friction factor depends on Reynold's number ($Re = uD \rho/\mu$ where μ is the fluid viscosity) and the relative roughness of the pipe (pipe roughness/pipe diameter). The standard values for f can be obtained from a graphical representation known as the **Moody diagram**. This is represented in PIPENET by the following empirical formulae (where r is the surface roughness of the pipe):

Laminar flow ($Re < 2000$):

$$f = 16/Re$$

Transitional flow ($2000 < Re < 3000$):

f is found by interpolating between laminar value for $Re = 2000$ and turbulent value at $Re = 3000$.

Turbulent flow ($Re > 3000$):

$$1/\sqrt{f} = -1.768\ln(0.27r/D + 1.252/Re\sqrt{f})$$

Bernoulli. Equation

Note that the Coulson-Richardson method is the same model as used in earlier products, where it was referred to by the more generic name of the Bernoulli equation.

19.8 Pipe C-factor

There are two possible methods for calculating pressure drop in pipes - the [Coulson-Richardson](#) method and the [Hazen-Williams](#) method.

The Coulson-Richardson method uses pipe roughness in its calculations, which is taken from the pipe schedule used for the Pipe Type. In this case there is no need to specify a C-factor for the pipe.

The Hazen-Williams method uses the pipe C-factor in its calculations, and so in this case the C-factor must be given.

It should be noted that for maximum flexibility the roughness or C-factor value of the Pipe Type can be overridden by the user when entering the individual pipe data. However, by default the roughness or C-factor of each pipe will be determined by the pipe's pipe type.

19.9 Pipe hydrostatic pressure loss

The pressure drop caused by the difference in elevation of the two ends of a pipe P_{elev} is given by:

$$P_{elev} = \rho g Z$$

where:

ρ is the fluid density.

Z is the change in elevation in the pipe.

g is the acceleration due to gravity.

19.10 Velocity Pressure

The velocity pressure is a measure of the energy required to keep the water in a pipe in motion. The PIPENET Spray/Sprinkler module provides a user option to include or ignore the velocity pressure. In the pressure calculation, all the pressures at the nodes are total pressure; velocity pressure is only variable for pipes.

When the velocity pressure is included, at any nozzle along a pipe, except the end nozzle, the pressure acting to cause flow from the nozzle is equal to the total pressure minus the velocity pressure on the upstream side; at the end nozzles, the pressure acting to cause flow from the nozzle is equal to the total pressure.

Options are also available to include pressure loss at entrance and pressure loss at exit. The pressure loss in the pipe caused by the entrance/exit effect is calculated according to the formulae in [\[CRANE\]](#). The entrance pressure drop formula is the most common Flush type with $r/d=0$.

Modelling

Normal pressure formula:

$$P_n = P_t - P_v$$

where

P_n is the normal pressure in Pa

P_t is the total pressure in Pa.

P_v is the velocity pressure in Pa

NFPA velocity pressure formula:

$$P_v = \frac{0.001123 Q^2}{D^4}$$

where

P_v is the velocity pressure in psi

Q is the (volumetric) fluid flow rate in gpm.

D is the pipe diameter in inches.

Standard velocity pressure formula:

$$P_v = \frac{1}{2} \rho v^2$$

where

P_v is the velocity pressure in Pa

ρ is the fluid density in kg/m³

v is the velocity in m/s

Entrance pressure drop formula:

$$\Delta P_{En} = 0.25 \rho v^2$$

where

ΔP_{En} is the frictional loss, in Pa, due to the entrance effect

ρ is the fluid density in kg/m³.

v is the velocity in m/s

Exit pressure drop formula:

$$\Delta P_{Ex} = 0.5 \rho v^2$$

where

ΔP_{Ex} is the frictional loss, in Pa, due to the exit effect

ρ is the fluid density in kg/m³.

v is the velocity in m/s

19.11 Equipment items

An equipment item is a fitting on a pipe which is not one of the nine standard fittings built into the Spray/Sprinkler module. It is defined by supplying:

- The label of the pipe to which it is fitted.
- A descriptor of up to 20 characters.
- An equivalent length.

19.12 Filters

Modelling Equation

$$P = A|Q| + BQ$$

where:

P is the pressure increase from inlet to outlet.

Q is the (volumetric) flow rate through the filter.

A is a coefficient less than or equal to zero.

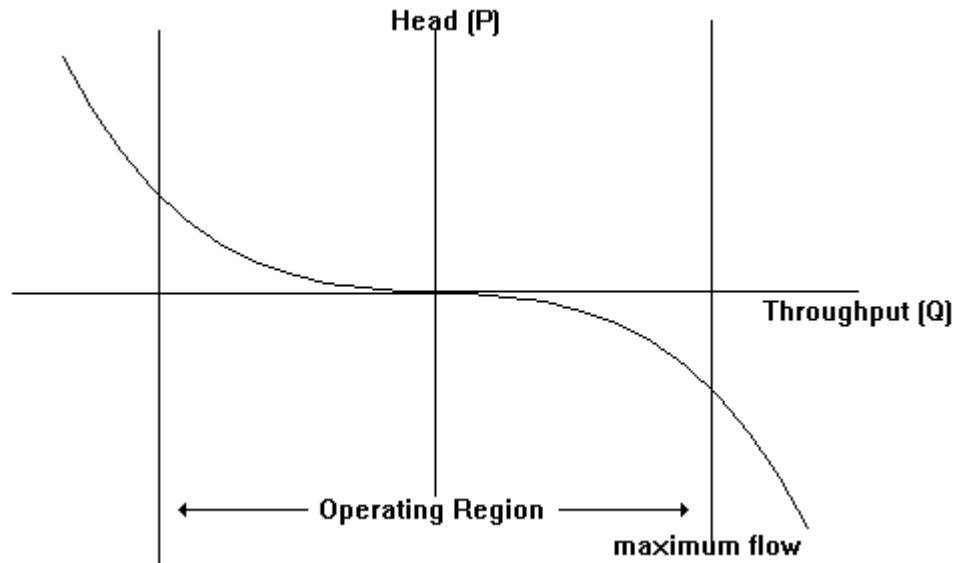
B is a coefficient less than zero.

This is operative for values of Q whose modulus is less than a given maximum flow Q_{max} . Note that filters are reversible (Q may be negative), and that as:

$$A \leq 0 \text{ and } B < 0$$

the pressure drops in the direction of the flow.

Typical Performance Curve



19.13 Deluge Valves

Defining a Deluge Valve Type

Before a deluge valve can be added to the network it must first be defined in the private data file via the **Libraries | Deluge Valve Type** menu option. Selecting this option will display a list of defined valve types.

A descriptor, manufacturer supplied K factor and X factor, and bore must be supplied.

Adding the valve to the network

Provide a label for the valve and identify the input and output nodes. The deluge valve type is selected from the list of types provided in the dialog.

Modelling Equation

$$P = Q^X/K$$

where:

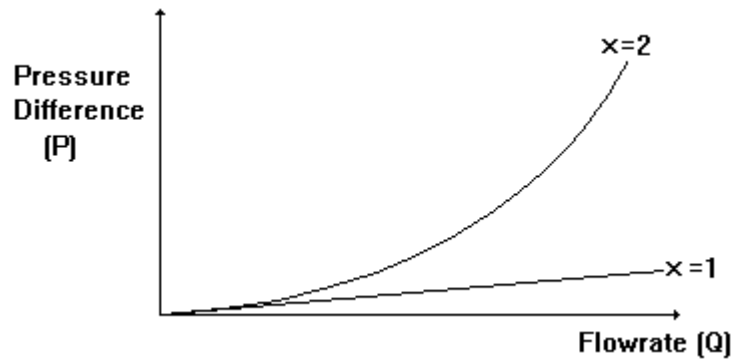
P is the pressure drop across the valve.

Q is the (volumetric) flow rate through the valve.

K is a constant for the valve.

X is a constant for the valve (typical values are 1 or 2, see curves below).

Typical Performance Curves



Data Required

In the library

1. valve descriptor to identify the type of valve (usually the manufacturer's valve name is used).
2. The valve K factor.
3. The valve X factor.

19.14 Non-return valve

Non-return valves allow unrestricted flow of fluid in a positive direction, and prevent all flow in a reverse direction. **Positive flow** is taken to mean from the valve's input node to its output node, in which case there will be no pressure drop across the component. Caution should be exercised not to position a non-return valve such that it would isolate a portion of the network. If this was to happen, the calculator could report an error: "network cannot be solved"

Note that even if the solution to the problem has the valve open, PIPENET can still generate this message. Occasionally, a non-zero "leakage flow" may be reported through a closed valve. This arises from rounding errors in the calculation which are smaller than the requested convergence accuracy, and is therefore usually negligible.

User Input Parameters

None.

Modelling Equations

Open valve:

$$P_1 = P_2.$$

Fully closed valve:

$$Q_1 = 0.$$

Continuity Equation:

$$Q_1 = Q_2.$$

where:

P_1 is the inlet pressure.

P_2 is the outlet pressure.

Q_1 is the inlet flow rate.

Q_2 is the outlet flow rate.

19.15 Overboard Dump Valves

Overboard dump valves (pressure safety valves) operate with a trigger pressure **PS**. When the pressure upstream of the valve is below **PS** the valve remains shut, and it opens when the pressure rises above **PS**. Once the valve is open, the upstream pressure will stabilize at **PS**. PIPENET models overboard dump valves as a special kind of outlet, with no need to model the network downstream of the point at which the trigger pressure will be measured.

Note that if there are any overboard dump valves present in the network then the **Remote nozzle specifications** and **Mass balance specifications** options will be greyed out in the [Calculations options dialog](#). This is because overboard dump valves can only be used with the **User-defined specifications** option. Note also, that if the **Remote nozzle specifications** or **Mass balance specifications** option has been selected the overboard dump valve icon on the toolbar will be greyed out (non-selectable).

19.16 Elastomeric Valve

The Elastomeric valve enables the user to achieve the required input pressure, output pressure, pressure drop or flow rate without the need to input the valve characteristic data. The elastomeric valve is representative of a valve type which fulfils the above roles; however, the model described below can be used to model any valve, including elastomeric valves, with the specified characteristics.

Note:

- i) The use of unreasonable target values might make the network unsolvable.
- ii) A warning message will be given in the calculation report when the output pressure is higher than the input pressure or the flow rate is negative.

User input parameters

- Type - Input pressure, Output pressure, Pressure drop or Flow rate
- Target value

Modelling equations

Design phase

$$P_1 = P_2$$

Analysis phase

Input pressure type - aim to control the input pressure of the valve to be the given target value:

$$P_1 = P_1^*$$

Output pressure type - aim to control the output pressure of the valve to be the given target value:

$$P_2 = P_2^*$$

Pressure drop type - aim to control the pressure drop of the valve to be the given target value:

$$P_1 - P_2 = \Delta P^*$$

Flow rate type - aim to control the flow rate of the valve to be the given target value:

$$Q = Q^*$$

where:-

- P_1 - input pressure
- P_2 - output pressure
- P_1^* - target input pressure
- P_2^* - target output pressure
- Q - flow rate
- Q^* - target flow rate

U

19.17 Orifice Plates

Orifice plates may be inserted into any pipes in the network, and produce an additional pressure drop across the pipe. In order to define an orifice plate the user must specify:

1. A label for the plate.
2. The label of the pipe to which the plate is attached.
3. Either the diameter of the orifice plate Or the pressure drop required across the plate.

Modelling - Pressure drop due to an Orifice Plate

Orifice plates may be inserted in any pipe, and are modelled using one of three models as selected by the user:

- Plates with flange tappings in accordance with BS1042, taking into account pressure recovery downstream. The restrictions of BS1042 are applied so plates may only be used in pipes with diameters in the range 2 – 14 inches (50.8 – 355.6 mm). Furthermore the ratio of the orifice diameter to the pipe diameter must be in the range 0.1 – 0.748 for pipes over 4 inches in diameter.
- The "Heriot-Watt Orange Book" orifice plate model [\[CRANE\]](#). This model derives from curves for laminar flow given by Miller in "Internal Flow Systems" and for turbulent flow given by the ESDU in "Flow of liquids. Pressure losses across orifice plates, perforated plates and thick orifice plates." The ratio of the orifice diameter to the pipe diameter should be less than 0.8.
- Sharp-edged orifice plates as described in Crane T.P. 410M. The ratio of the orifice diameter to the pipe diameter should be in the range 0.2 – 0.75. In addition, the Reynolds number of flow in the pipe should be greater than 100, as the accuracy of the model decreases at lower values.

The pressure drop due to an Orifice Plate, P_{plate} , may be found in one of two ways: either it may be specified directly by the user (in which case PIPENET will calculate the orifice diameter necessary to produce this pressure drop), or it can be calculated by PIPENET from the orifice diameter given by the user. Users should exercise caution when using this facility as the calculation of pressure drop from plate diameter is unstable in that a small change in orifice diameter may result in a very large change in the pressure drop.

Chinese Standard (GB)

With Chinese Standard (GB) options, the pressure drop due to an orifice plate is calculated as follows:

$$P_{plat} = \xi \frac{u^2}{2g}$$

Where:

P_{plat} is the pressure loss due to the orifice plate, unit is 10^{-2} M pa.

u is the fluid velocity, unit is m/s

g is the acceleration due to gravity

ξ is the friction factor of the orifice plate, which can be calculated using the following equation:

$$\xi = \left[\begin{array}{c} 1.75 \frac{D^2}{D_o^2} \cdot \frac{1.1 - \frac{D_o^2}{D^2}}{1.175 - \frac{D_o^2}{D^2}} - 1 \end{array} \right]$$

Where:

D is the internal diameter of the pipe, unit is m.
 D_o is the diameter of the orifice plate, unit is m

19.18 Pumps

A pump provides a pressure increase which depends on the pump speed and performance curve. The pump performance curve is entered as a library.

User Input Parameters

Pump Type - selected from a list of defined library pumps (either with unknown coefficients or known coefficients).

Quadratic Modelling Equations

The pressure increase produced by the pump is:

$$P = A Q^2 + B s Q + C s^2$$

where:

s is the pump setting.

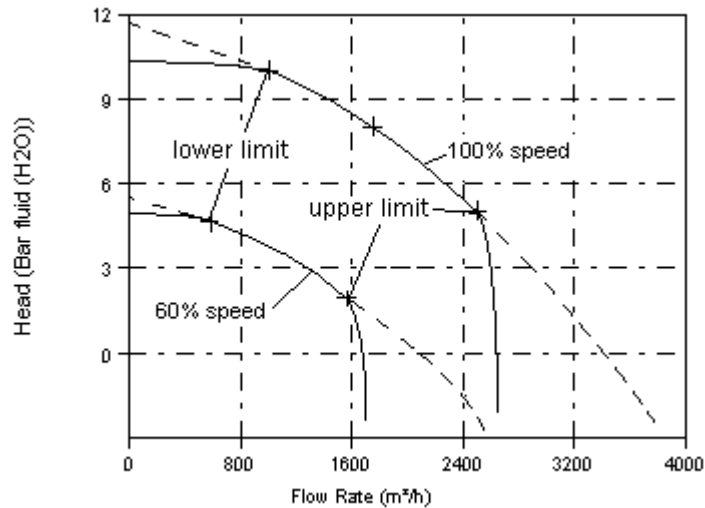
Q is the flow rate through the pump.

A, B, C are pump performance coefficients.

The pump performance coefficients A, B and C are entered directly, or calculated by PIPENET for a specified pump curve, in [pump libraries](#). The user also specifies an operating range for flow rates through the pump. It is recommended that the zero flow rate point is given when defining a pump curve.

The pump setting, s , must be between 0.0 and 1.0. A setting $s = 0.0$ represents a shut down pump, while $s = 1.0$ represents a pump at full speed.

The quadratic curve will, in general only apply to the specified working range of the pump, however degeneration factors can be used to specify the behaviour of the curve below the specified working range and/or above the specified working range.



Within the work region – Q_{LL} to Q_{UL}

PIPENET calculates a binomial correlation to fit the input data, see the solid line in the above figure:

$$P = AQ^2 + BsQ + Cs^2$$

Below the minimum flow rate Q_{LL}

We introduce a degeneration factor n , which is in the range 0 to 10. The larger the value of n , the flatter of the pump curve is. When the degeneration exponent n is zero, i.e. no degeneration the equation becomes the normal quadratic.

$$\Delta P = 2^{-n} \alpha (Q - Q_{LL}) + \beta$$

where:

$$0 \leq n \leq 10$$

$$\alpha = 2AQ_{LL} + Bs$$

$$\beta = AQ_{LL}^2 + BsQ_{LL} + Cs^2$$

$$Q_{LL} = Q_{min} s$$

Above the maximum flow rate Q_{UL}

We introduce a degeneration factor m , which is in the range 0 to 10. The larger the value of m the quicker of the pump curve degenerates..

$$\Delta P = AQ^2 + BsQ + Cs^2 - (2^m - 1)|A|(Q - Q_{UL})^2$$

where:

$$0 \leq m \leq 10$$
$$Q_{UL} = Q_{max} s$$

Notes

If the user knows the performance coefficients for a pump and does not want to use a pump library, then the pump can be defined as a non-library pump by giving the values of A , B , C , Q_{min} , Q_{max} .

In order for the calculator to function correctly, it is necessary to ensure that there is only one flow rate corresponding to each pressure gain, and so the following restrictions are applied:

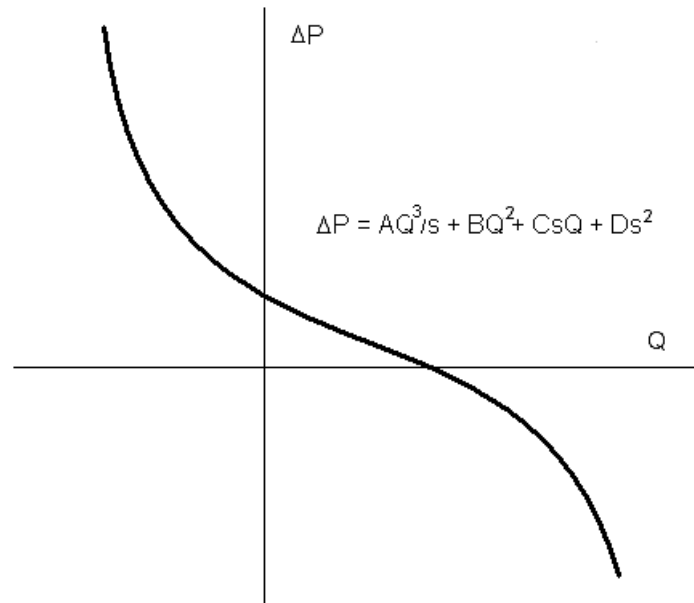
- For flow rates between Q_{min} and Q_{max} , the slope of the performance curve must be negative or zero.
- For flow rates between Q_{min} and Q_{max} , there must be no flow rate that gives no pressure change. (That is, the performance curve must not cross the horizontal axis.)

For flow rates outside the range Q_{min} to Q_{max} , PIPENET extrapolates the performance curve using the tangent to the curve at the minimum or maximum flow rate, and issues an appropriate warning message.

PIPENET will calculate the power required by a pump based on specified efficiency. The power calculation will assume that the pump is 100% efficient if the efficiency is not specified.

Cubic Modelling Equations

A simple quadratic curve can deviate considerably from the supplied pump data, especially when the flow rate exceeds the pump capacity, or is less than the allowed minimum flow rate. The cubic curve can provide better results.

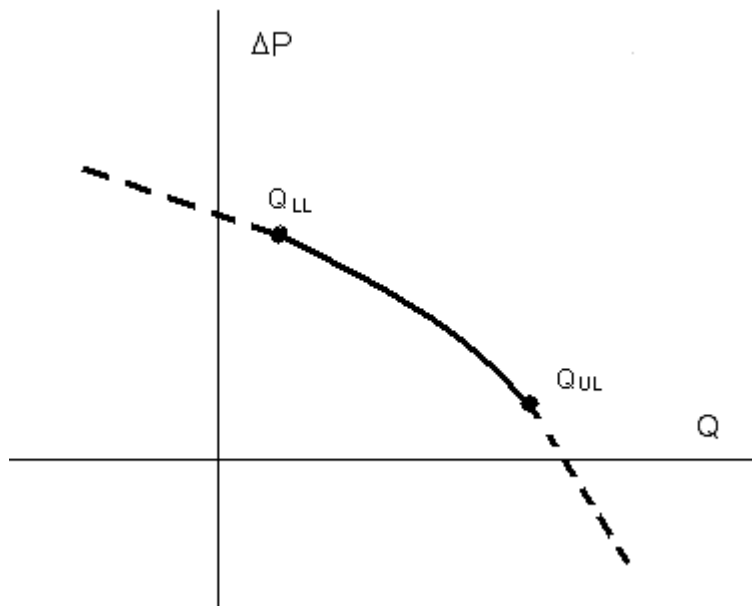


There is a potential problem with the cubic curve in that it may lead to multiple solutions. Therefore, the above function must satisfy the following condition:

$$B^2 - 3AC \leq 0$$

Smooth Cubic Spline Modelling Equations

The smooth curve uses cubic spline functions to fit a known pump curve. The obtained curve is not only a smooth curve but also can closely match all input data.



Within the specified working range Q_{LL} to Q_{UL} in the region of $[Q_i, Q_{i+1}]$

$$\Delta P = A_i Q^3 / s + B_i Q^2 + C_i s Q + D_i s^2$$

Below the lower limit, with $Q < Q_{LL}$ and along the tangent direction at the point $[Q_{LL}, \Delta P_{LL}]$

$$\Delta P = \alpha(Q - Q_{LL}) + \beta$$

where:

$$Q_{LL} = Q_{min} s$$

$$\alpha = 3A_1 Q_{LL}^2 / s + 2B_1 Q_{LL} + C_1 s$$

$$\beta = A_1 Q_{LL}^3 / s + B_1 Q_{LL}^2 + C_1 s Q_{LL} + D_1 s^2$$

Above the upper limit, with $Q > Q_{UL}$ and along the tangent direction at the point $[Q_{UL}, \Delta P_{UL}]$

$$\Delta P = \alpha(Q - Q_{UL}) + \beta$$

where:

$$Q_{UL} = Q_{max} s$$

$$\alpha = 3A_n Q_{UL}^2 / s + 2B_n Q_{UL} + C_n s$$

$$\beta = A_n Q_{UL}^3 / s + B_n Q_{UL}^2 + C_n s Q_{UL} + D_n s^2$$

NPSH

Net Positive Suction Head (NPSH) is an analysis of the energy conditions on the suction side of the pump to determine if the liquid will vaporize at the lowest pressure point in the pump. NPSH is measured in the units of length.

There are two components that must be considered for NPSH:

- **NPSHR** is defined as Net Positive Suction Head Required. NPSHR is a function of the design of the pump, and is determined by the pump manufacturers via testing.
- **NPSHA** is defined as Net Positive Suction Head Available, which can be calculated as follows:

$$NPSHA = (P_i - P_{vap}) / \rho g$$

$$= \frac{P_{atm}}{\rho g} - H_s - \frac{P_{vap}}{\rho g}$$

Where,

P_i is the (absolute static) pressure at pump inlet;

P_{vap} is the vapour pressure;

ρ is the liquid density;

g is the gravitational acceleration;

P_{atm} is the atmospheric pressure;

H_s is the suction head which includes not only the suction lift but also the sum of the losses in the inlet pipe and the velocity head.

If $NPSHA < NPSHR$, then **cavitation** will happen.

The **cavitation parameter** is a dimensionless ratio used to relate the conditions which inhibit cavitation to the conditions which cause cavitation. There are several common forms of the cavitation parameter. In PIPENET, the cavitation parameter uses the definition:

$$\text{cavitation parameter} = NPSHA / H$$

where H is the total head of the pump.

The calculation of NPSHA and cavitation parameter is based on the vapour pressure of water at 60 F (15.6 C) at 1767.745 Pa A, which was calculated using Water-steam property facility of the PIPENET Transient module. This information will be given as a note below the table in the report. This vapour pressure value is only used to calculate NPSHA and cavitation. It is not used for other purpose, such as a warning of cavitation.

19.19 Spray Nozzles

User Input Parameters

User supplied descriptor and user supplied K factor, minimum and maximum operating pressures.

Modelling

$$Q = K\sqrt{P}$$

where:

Q is the (volumetric) flow rate through the nozzle.

P is the drop in pressure across the nozzle.

K is a constant for the nozzle.

This equation holds for values of P between given minimum and maximum values P_{\min} and P_{\max} . This is known as the working pressure range of the nozzle.

Note that the value of K depends on the units used for P and Q. Generally the units used for P and Q when quoting the K value of a nozzle are bar and litres/min respectively.

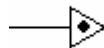
Data Required

In the library:

1. A nozzle descriptor to identify the type of nozzle (usually the manufacturer's nozzle name is used).
2. The nozzle K-factor.
3. P_{\min} and P_{\max} .

In the input:

1. Nozzle status (ON or OFF) - note that if a nozzle is off this is indicated in the schematic by a presence of a small solid circle inside the nozzle symbol:



2. Nozzle label.
2. Input node.
4. Nozzle name in the library.
5. The flow rate required through the nozzle.

Notes

1. Nozzles can be defined without including them in a library by specifying the values of K, P_{\min} and P_{\max} . When there are several nozzles of the same type it is easier to define the type in a library as described above.
2. The flow rate required through the nozzle is used by PIPENET's [Design Facility](#).

19.20 Fittings

All fittings are modelled as equivalent lengths according to either the [FOC](#) or [NFPA](#) rules, summarized in the Reference data section.

The range of fittings provided is that defined by [\[NFPA\]](#). If you wish to add some device or fitting not included in the range of available fittings then you may be able to consider the use of an [Equipment item](#).

Crosses and Tees

For the placement of crosses and tees the NFPA guidelines [\[NFPA\]](#) pages 13-213 to 13-216 provides the following information

- 22.4.4.7 (3) Calculate the loss for a tee or a cross where flow direction change occurs based on the equivalent pipe length of the piping segment in which the fitting is included.
- 22.4.4.7 (4) The tee at the top of a riser nipper shall be included in the branch line, the tee at the base of a riser nipple shall be included in the riser nipple, and the tee or cross at a cross main-feed main junction shall be included in the cross main.
- 22.4.4.7 (5) Do not include fitting loss for straight-through flow in a tee or cross.

It is possible, however, that sometimes a user may wish to define the fittings in the different way, according to their own good engineering practices.

NFPA Submittal Report

Part

20

20 NFPA Submittal Report

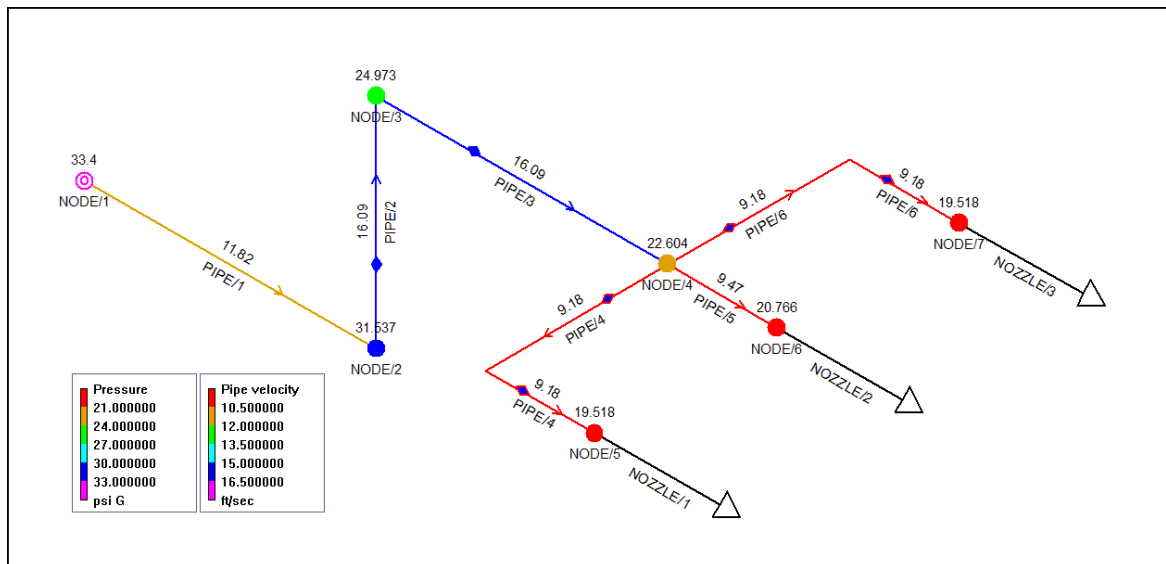
20.1 NFPA Submittal Report

Hydraulic calculation details can be written to a file in tabular form in order to assist in the preparation of a fire protection submittal of the kind required by the NFPA. This is done by selecting **Calculation | NFPA submittal report**, which will open a dialog allowing the output file to be selected; the output is written in rich text format (.RTF file). Note that this option is only enabled once a calculation has been performed.

The output file contains three tables. The first table “Summary values” contains various items which can be copied on to the submittal summary sheet; the second and third tables – “Node analysis” and “Pipe information” – are designed to comply with the format specified in the 2007 Edition of NFPA-13, and are suitable for inclusion in the submittal report. Values are shown in the user's currently selected units. Node heights are shown relative to an arbitrary datum node unless both i) pipe/duct elevations are in use (see **Elevations** on the **Options | Module options** dialogue) and ii) a reference node and height have been specified (see **Calculation of node heights from pipe elevations** on the **Options | Calculation** dialog).

Velocity pressure information can optionally displayed for each pipe in the Notes column. Selection of this option is via the [Spray options dialog](#).

Using the following network as an example:



The following tables are generated:

Title	Example 3-sprinkler system
Calculation date	1-Oct-2007 15:54
Calculator	PIPENET Vision Spray calculator, version 4.20
Friction loss formula	Hazen-Williams
Design standard	NFPA 1996/2001
Convergence accuracy	1.000000E-03
Elevation check tolerance (ft)	1.640
Total number of sprinkler heads	3
Number of sprinkler heads on	3
Total sprinkler discharge (US.gal/min)	75.000
Total non-sprinkler output flow (US.gal/min)	0.000
Total input flow (US.gal/min)	75.000
Highest fluid velocity (ft/sec)	16.088
Pressure at input nodes	See NODE ANALYSIS table

Summary Table

Node tag	Elevation (ft)	Node Type	Pressure (psi G)	Discharge (US.gal/min)	Notes
NODE/1	0.00	Input	33.400	0.000	
NODE/2	0.00		31.537	0.000	
NODE/3	6.50		24.973	0.000	
NODE/4	6.50		22.604	0.000	
NOZZLE/1	6.50	Sprinkler	19.518	24.741	
NOZZLE/2	6.50	Sprinkler	20.766	25.519	
NOZZLE/3	6.50	Sprinkler	19.518	24.741	

Node Table

Node 1	Elev 1 (ft)	K factor	Flow added (q) (US.gal/min)	Nominal ID (inches)	Fittings - quantity, type, and equivalent length (ft)	L (ft)	C factor	Pt (psi G)	Notes
						F (ft)		Pe (psi)	
Node 2	Elev 2 (ft)	Pipe label	Total flow (Q) (US.gal/min)	Actual ID (in)		T (ft)	Pf per ft (psi)	Pf (psi)	
NODE/1	0.00		75.000	1.5		10.000	120.0	16.841	
						0.000		0.000	
NODE/2	0.00	PIPE/1	75.000	1.610		10.000	0.186	1.864	
NODE/2	0.00			1.25	1xE=3.00	6.499	120.0	10.277	
						3.000		-2.813	
NODE/3	6.50	PIPE/2	75.000	1.380		9.499	0.395	3.751	
NODE/3	6.50			1.25	1xE=3.00	3.000	120.0	7.909	
						3.000		0.000	
NODE/4	6.50	PIPE/3	75.000	1.380		6.000	0.395	2.369	
NODE/4	6.50	5.60		1	1xE=2.00	8.999	120.0	4.822	
					1xT=5.00	7.001		0.000	
NOZZLE/1	6.50	PIPE/4	24.741	1.049		16.001	0.193	3.086	
NODE/4	6.50	5.60		1		8.999	120.0	6.070	
						0.000		0.000	
NOZZLE/2	6.50	PIPE/5	25.519	1.049		8.999	0.204	1.839	
NODE/4	6.50	5.60		1	1xE=2.00	8.999	120.0	4.822	
					1xT=5.00	7.001		0.000	
NOZZLE/3	6.50	PIPE/6	24.741	1.049		16.001	0.193	3.086	

Pipe Information

Exporting the Schematic

Part

21

21 Exporting the Schematic

21.1 HP-GL/2 Output

Via the **Tools | Export HP-GL/2** menu option the schematic can be output as a Hewlett-Packard HP-GL/2 file, with a .hpgl file extension. This type of file can be read by a number of graphics programs and is commonly used as an offline plotting format. The specifications for the HP-GL/2 format are provided by [[HPGL2](#)].

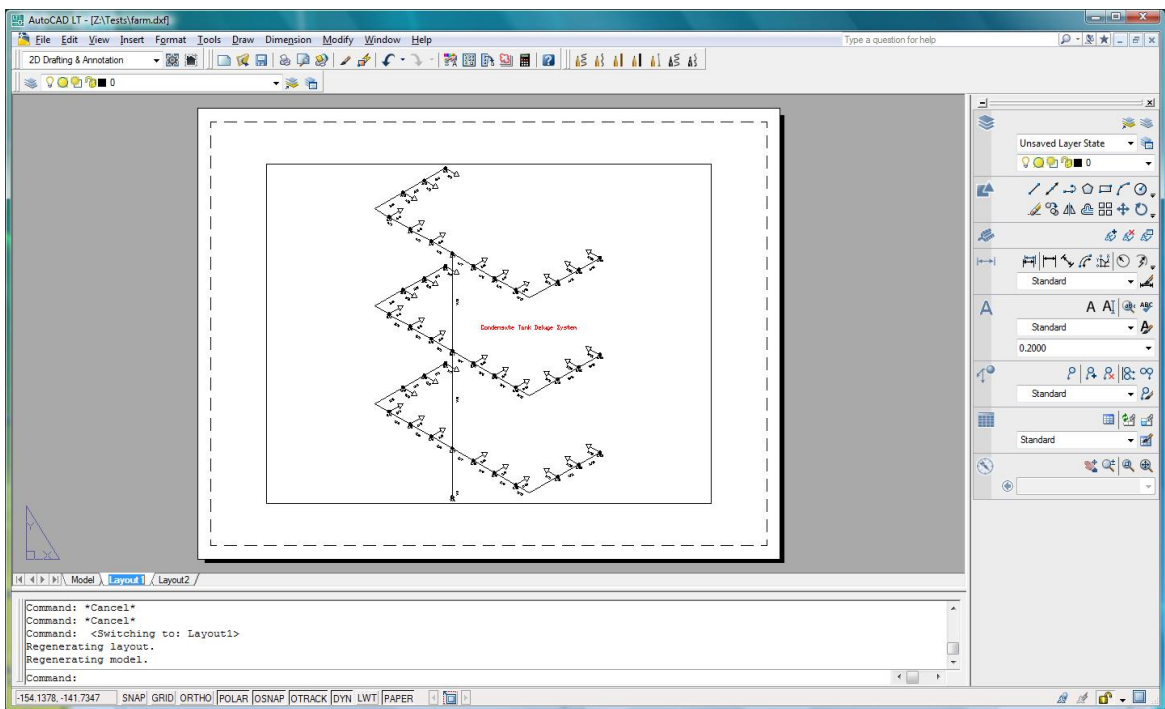
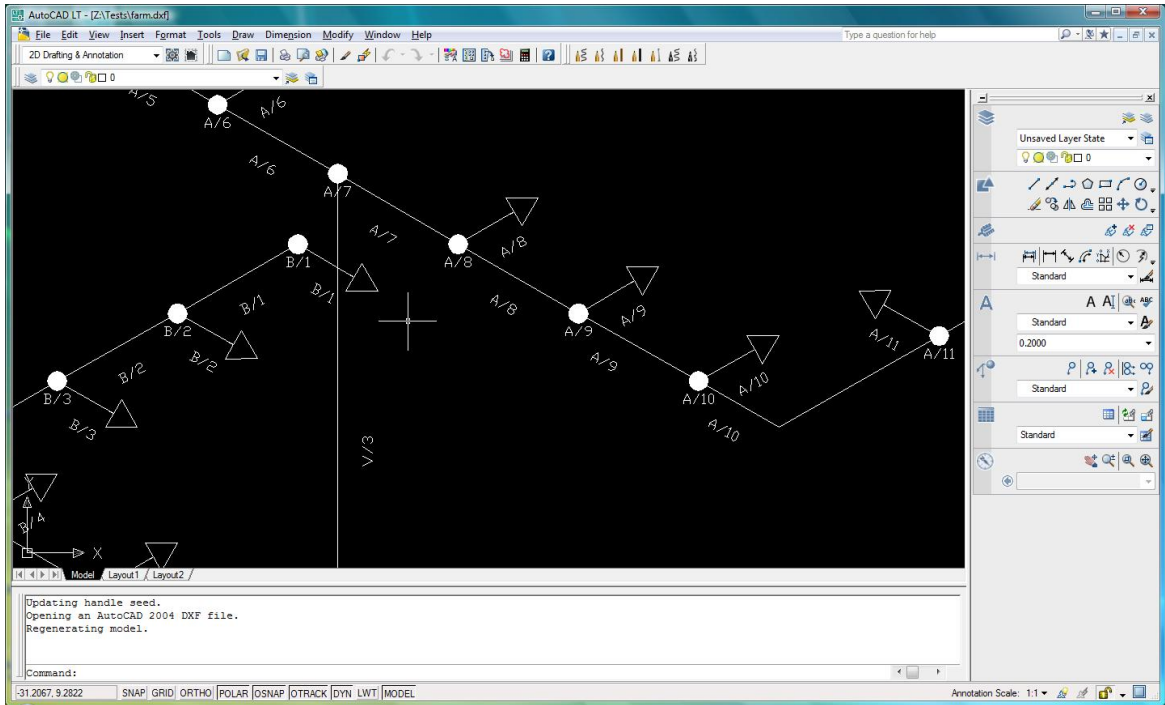
21.2 DXF Output

Via the **Tools | Export DXF file** menu option the schematic can be output as an AutoCAD DXF ASCII file. The format of the DXF files produced is according to the Autodesk DXF Reference March 2006 [[ACAD06](#)]. The DXF format is a complex format and one which can change with releases of AutoCAD. The output produced by PIPENET has been verified by loading the files into AutoCAD 2008, but is also known to be compatible with a number of other CAD programs, for example QCAD (low cost CAD program from RibbonSoft.) and progCAD 2007 (ProgeSOFT).

The DXF file specifies the following layers:

- **SCHEMATIC** - the main part of the schematic including all nodes and components
- **NLABELS** - node labels
- **CLABELS** - component labels
- **INDICATORS** - fittings symbol (blue lozenge) and directional arrows.
- **TEXT** - text items other than node and component labels.

The following screenshots show model and paper layouts:



Part

22

22 Errors

22.1 Errors

Errors can be detected at various points in the opening of an existing network file, or in the creation of a new network:

- **Basic errors** in the operating environment, the most common error being the absence of a valid licence to run a PIPENET module, or the absence or incorrect operation of the security key drivers.
- **Input errors** detected either in reading the main data file or an associated library file
- **Specific component errors** detected in the GUI when a [check operation](#) is performed, for example pipe has zero length – these errors should be self-explanatory;
- **Global errors** detected in the graphical user interface (GUI) when a [check operation](#) is performed
- **Numerical errors** that can only be checked after the calculation starts and are therefore reported in the calculator output.

22.2 Basic errors

These are usually related to the absence of a valid licence to run the program or incorrect operation of the security key drivers, but other options are also possible:

- Attempts to run the software on earlier versions of Windows operating system may fail – the software is only validated for Windows 2000 and XP.
- A working PIPENET system may suddenly stop working if a new operating system is installed or a new Windows service pack is installed. Generally Sunrise Systems will discover these errors in advance and be able to provide suitable fixes, new security key drivers, etc.
- No valid licence – separate licences are available for the three PIPENET modules Transient, Spray/Sprinkler and Standard. Attempting to run a module for which you have no licence will produce an error. To determine which licences you have see the [Help | Key status menu](#) option . If in doubt, contact Sunrise Systems for details of your current licences.
- Licence has expired – with the issue of PIPENET VISION 1.10 expiry date checking was added to HASP security keys. If an old key is being used, or if the licence for the module you are attempting to run has expired, then a message will be displayed informing you of the situation. To see the expiry dates see the [Help | Key status menu](#) option, if no expiry dates are shown then you have an old key that needs to be updated. Keys can be updated remotely by Sunrise Systems providing (usually via email) a small encrypted key-update file – see the [Updating a key licence](#) section for further details.
- Drivers for security keys not installed correctly – see the installation instructions supplied with the software for information on how to check the status of the drivers and if necessary re-install them. Periodically the supplier of the security

key drivers will have to issue new versions of the drivers when a new operating system or service patch is released. When this occurs Sunrise Systems will make the drivers available on our website or via email.

22.3 Input errors

Input errors can occur when reading either old .DAT files or library files, or more rarely when reading the newer .SDF data files and SLF library files. These errors should not occur, since PIPENET wrote the files and they should be correct. However, with the older .DAT files and library files some users would edit the file directly using a simple text editor, and this could result in errors. With the newer file formats (which can still be opened in a simple text editor) it is not expected that users will directly edit the files.

One error that may occur with both the old and new types of file is that one of the library files referenced by the main data file may be missing. In the earlier version of PIPENET this could occur if the .DAT file and the associated library files were moved to a new location, since the .DAT file would contain references to the library files in their original location. With PIPENET VISION, if a library file cannot be found in its original location a search will be made in the directory where the main data file (.SDF file) is located.

22.4 Global errors

These are errors detected when a [check is performed](#), and which relate either to the network as a whole, or to some section of the network. For the Spray/Sprinkler module the following errors are possible, classified as errors, warnings or information:

At least one pipe type must be defined – error

All pipes in a Spray/Sprinkler module must belong to a pipe type, therefore at least one pipe type must have been defined.

A library deluge valve must be defined – error

User defined deluge valves are not allowed. There is a deluge valve in the network but there is no library deluge valve defined.

Blocked pipe found – warning

A blocked pipe has been found – this is a warning message only but is noted since blocked pipes can have an expected affect on a calculation. If this item is double-clicked in the [Check dialog](#) the pipe with the block will be highlighted.

Broken pipe found – warning

A broken pipe has been found – this is a warning message only but is noted since blocked pipes can have an expected affect on a calculation. If this item is double-clicked in the [Check dialog](#) the pipe with the break will be highlighted.

Height discrepancy discovered at node – error

Pipe elevations are being used, and in checking a loop an error has been discovered, that is the sum of all elevations in the loop does not sum to zero. Double-clicking on this item will highlight the loop in error in the schematic, for

further information see the [height checking section](#)..

I/O node must be on a single pipe when using variable fluid properties and volumetric flow rate - error

An input or output node must only be connected to one pipe when using variable fluid properties and volumetric flow rate.

Most remote nozzle option specified but no nozzles in network - error

The Most remote nozzle specification option can only be used if there are one or more nozzles present in the network.

Network is empty – error

Network contains no components and there is therefore nothing to calculate.

Network is over specified in analysis phase - error

There are too many specifications in the analysis phase, that is the number of specifications and number of input/output nodes do not meet the specification requirements. This message is not produced if the Spray/Sprinkler module is being used with the [Mass Balance option](#).

Network is under specified in analysis phase – error

There are insufficient specifications in the analysis phase, that is the number of specifications and number of input/output nodes do not meet the [specification requirements](#). This message is not produced if the Spray/Sprinkler module is being used with the [Mass Balance option](#).

Network is under specified in design phase - error

There are insufficient specifications in the design phase, that is the number of specifications and number of input/output nodes do not meet the [specification requirements](#).

Network must have at least one inlet - warning

The network must have at least one inlet.

Network must have outlets or nozzles – error

The network must have one or more outlets.

No analysis phase pressure specification – error

At least one pressure specification must be provided for the analysis phase – see [specifications requirements](#). This message is not produced if the Spray/Sprinkler module is being used with the [Mass Balance option](#).

Non-pipe component has unequal input and output elevations - error

If node elevations are being used then the input and output nodes of all non pipe components, for example pumps and valves, must have the same elevation.

This node may need to declared as an input node - warning

This warning message will appear for each node that has a single pipe that uses the

node as an input, but there is no pipe that uses the node as an output. That is, there appears to be flow out of the node but there is no flow into the node.

This node may need to be declared as an output node - warning

This warning message will appear for each node that has a single pipe that uses the node as an output, but there is no pipe that uses the node as an input. That is, there appears to be flow into the node but there is no flow out of the node.

No open nozzle found – warning

One or more nozzles have been defined and none of them are open.

One I/O node with no flow spec is required for Design - warning

If a design phase is being used then there must be at least one input/output node with no flow specification – see [specifications requirements](#)

Pipe type <pipe-type-name> has a lining that is too thick – error

The pipe named pipe type has a lining with a thickness that is greater than or equal to half the smallest used bore.

Reversed flow at inlet- warning

There is a reversed flow at an input node. This will occur if (and taking specifications into account) the total flow into the network is greater than the flow out of the network and the node has been specified as an input node with the value for the flow specification being unset.

Reversed flow at outlet- warning

There is a reversed flow at an output node. This will occur if (and taking specifications into account) the total flow out of the network is greater than the flow into the network, and the node has been specified as an output node with the value for the flow specification being unset.

There must be at least two specifications - error

There must be at least two specifications provided for the analysis phase – see [specifications requirements](#). This message is not produced if the Spray/Sprinkler module is being used with the [Mass Balance option](#).

22.5 Specific component errors

These are errors detected during a [check operation](#) and relate to a single specific component. These errors are self-explanatory and usually relate to an incorrect or missing parameter. Examples include:

- Elevation change exceeds length (pipes)
- Minimum pressure must be less than the maximum pressure (nozzles)
- Coefficients must be less than zero (filters)
- Dump valve - Trigger pressure must be set and > 0.0 (overboard dump valves)

- Flow density must be given (nozzles)
- Pump efficiency must be given (pumps)
- Orifice plate - pressure drop must be given (orifice plates)

Note that many of these errors are of the general form *<attribute name> must be given*, as in the last three examples above, indicating that a value for the attribute has not been provided.

Double-clicking on the error in the [Check status](#) dialog will highlight the component in the schematic.

22.6 Numerical errors

There is a general category of errors that cannot be detected in the GUI because they can only be checked by the calculator, either in setting up the calculation, or during the calculation itself. For example:

- calculation fails to converge – this may be because the calculation cannot be solved as defined, or it may just require a few more iterations than are specified in the [calculation options](#). A calculation can also fail to converge if the network is incorrectly specified in some way.

Also note that when you request a calculation a check is performed first and any errors or warnings noted in the check status dialog.

Part

23

23 Reference Data

23.1 Bibliography

[CRANE] Technical paper No 410M "Flow of Fluids Through Valves, Fittings and pipe"; Crane Co, 1988

[NFPA13] NFPA 13, "Standard for the Installation of Sprinkler Systems"; NFPA 2007

[NFPA15] NFPA 15, "Standard for Water Spray Fixed Systems for Fire Protection"; NFPA 2007

[CHINGB] Code of Design for Sprinkler Systems GB 50084-2001; National Bureau of Quality and Technology Supervision, P.R. China and Ministry of Construction, P.R. China; April 5 2001.

[HPGL2] "The HP-GL/2 and HPRTL Reference Guide", Third edition, Addison Wesley, 1997

[ACAD06] "DXF Reference", March 2006, Autodesk

23.2 Properties of water

The physical properties of water are built into PIPENET. A selection of the properties for the temperature range 5 - 90°C are shown below.

Temp °C	Viscosity			Density	
	Pa s	Cp	lb/ft.hr	kg/m ³	lb/ft ³
5	0.001506	1.506	3.643	999.9	62.42
10	0.001300	1.300	3.145	999.7	62.41
15	0.001135	1.135	2.746	999.1	62.37
20	0.001002	1.002	2.424	998.2	62.32
25	0.000891	0.891	2.156	997.0	62.24
30	0.000797	0.797	1.928	995.6	62.15
35	0.000718	0.718	1.737	994.0	62.05
40	0.000651	0.651	1.575	992.2	61.94
50	0.000544	0.544	1.316	988.1	61.69
60	0.000463	0.463	1.120	983.3	61.38
70	0.000400	0.400	0.968	977.5	61.02
80	0.000351	0.351	0.849	971.8	60.67
90	0.000311	0.311	0.752	965.3	60.2

23.3 Surface Roughness

The following table shows values of surface roughness for typical materials. These values have been obtained from a number of sources and are not endorsed by Sunrise Systems Limited.

Material	Roughness	
	(mm)	(inch)
Asbestos Cement	0.015 – 0.06	5.91×10^{-4} to 2.36×10^{-3}
Brass	0.001 to 0.003	3.94×10^{-5} to 1.18×10^{-4}
Cement – smoothed	0.3	1.18×10^{-2}
Concrete - ordinary	0.6 - 1.5	2.36×10^{-2} to 5.91×10^{-2}
Concrete – precast	0.06 – 0.6	2.36×10^{-3} to 2.36×10^{-2}
Concrete – coarse	0.3 – 5.0	1.18×10^{-2} to 1.97×10^{-1}
Copper	0.001 to 0.003	3.94×10^{-5} to 1.18×10^{-4}
Glass Reinforced Plastic (GRP)	0.03 – 0.06	1.18×10^{-3} to 2.36×10^{-3}
Lead	0.001 to 0.003	3.94×10^{-5} to 1.18×10^{-4}
PVC and plastic pipe	0.0015 – 0.007	5.91×10^{-5} to 1.18×10^{-4}
Cast iron		
New - unlined	0.25 to 0.8	9.84×10^{-3} to 3.15×10^{-2}
Worn	0.8 to 1.5	3.15×10^{-2} to 5.91×10^{-2}
Rusty	1.5 to 2.5	5.91×10^{-2} to 9.84×10^{-2}
Asphalted	0.12	4.72×10^{-3}
Steel		
Stainless	0.015	5.91×10^{-4}

Material	Roughness	
	(mm)	(inch)
Stainless – fine drawn	0.0015	5.91×10^{-5}
Commercial pipe	0.045 to 0.09	1.77×10^{-3} to 3.54×10^{-3}
Stretched	0.015	5.91×10^{-4}
Riveted	1.0 to 9.00	3.94×10^{-2} to 3.54×10^{-1}
Welded	0.045	1.77×10^{-3}
Galvanized	0.15	5.91×10^{-2}
Rusted	0.1 – 1.0	3.94×10^{-3} to 3.94×10^{-2}

When a range of values is provided the lower value is for new pipes and the upper value for old pipes. When a single value is quoted this is for new pipes. Unfortunately there is no single source for the above data, and the data has been compiled from a number of sources, including:

1. The Engineering Toolbox (www.engineeringtoolbox.com)
2. Binder, R.C. (1973), Fluid Mechanics, Prentice-Hall, Inc. (Englewood Cliffs, NJ).
3. Tables for Hydraulic design of Pipes, Sewers and Channels
4. Coulson and Richardson, Chemical Engineering, Pergamon

23.4 Hazen-Williams Coefficients

The following table provides guidance and suggested values for C-factors. These values have been obtained from a number of sources and are not endorsed by Sunrise Systems Limited.

Material	C-factor	
	Possible values	Suggested value (NFPA13)
Asbestos Cement	140 - 150	140
Brass	120 - 150	
Black steel (dry systems)	100	100

Material	C-factor	
	Possible values	Suggested value (NFPA13)
Black steel (wet systems)	120	120
Cast iron		
New unlined	120 - 130	100
10 years old	105 - 75	
15 years old	100 - 60	
20 years old	95 - 55	
30 years old	85 - 45	
50 years old	75 - 40	
Bitumen-lined	140	
Cement-lined	140	140
Concrete	85 - 150	140
Copper	120 - 150	150
Fibre glass pipe	150 - 160	
Fire hose (rubber)	135	
Galvanized steel	120	120
Lead	130 - 150	
Polyethylene	150	
PVC and plastic pipe	150	150
Stainless steel	150	150
Steel new and unlined	140 - 150	
Steel, welded and seamless	100	
Vitrified clays	110	

When a range of values is provided the smoother value (higher C-factor) is for new pipes

and the lower value for old pipes. When a single value is quoted this is for new pipes. Unfortunately there is no single source for the above data, and the data has been compiled from a number of sources, including:

1. NFPA13, 2007 edition
2. The Engineering Toolbox (www.engineeringtoolbox.com)
3. Binder, R.C. (1973), Fluid Mechanics, Prentice-Hall, Inc. (Englewood Cliffs, NJ).
4. Harry E. Hickey (1980), Hydraulics for Fire Protection, NFPA publication

23.5 Physical Constants

All values are SI.

Atmospheric Pressure	101325 Pascal
Acceleration due to gravity	9.80665 ms ⁻²
0° C	273.15 Kelvin

23.6 FOC Rules

Equivalent Lengths (metres) of Fittings for FOC Rules Pipe Sizes 15mm to 40mm

Code	Fitting Type	mm	15.0	20.0	25.0	32.0	40.0
		in.	0.5	0.75	1.0	1.25	1.5
1	45° Screwed Elbow		0.34	0.34	0.4	0.55	0.66
2	90° Screwed Elbow (Standard)		0.63	0.63	0.77	1.04	1.22
3	90° Welded Elbow (Long Radius)		0.3	0.3	0.36	0.49	0.56
4	Screwed Tee or Cross (Flow turned 90°)		1.25	1.25	1.54	2.13	2.44
5	Gate Valve (Straightway)		0.38	0.38	0.38	0.38	0.38
6	Alarm or Non-Return Valve (Swinging)		2.42	2.42	2.42	2.42	2.42
7	Alarm or Non-Return Valve (Mushroom)		12.08	12.08	12.08	12.08	12.08
8	Globe Valve (Straightway)		16.43	16.43	16.43	16.43	16.43
9	Butterfly Valve		2.19	2.19	2.19	2.19	2.19

Notes

1. Data taken from 29th Edition FOC Rules, Table 6124.
2. The above values are used only for a Hazen-Williams [C Factor](#) of 120. For other values of [C factor](#), the values are scaled in accordance with Table 6124 of the

FOC Rules.

3. Equivalent lengths of 15mm fittings are assumed the same as the corresponding 20mm fitting.
4. Equivalent lengths of valves below 50mm are assumed the same as the corresponding 50mm valve.

Pipe Sizes 50mm to 100mm

Code	Fitting Type	mm	50.0	65.0	80.0	90.0	100.0
		in.	2.0	2.5	3.0	3.5	4.0
	45° Screwed Elbow		0.76	1.02	1.27	1.44	1.61
	90° Screwed Elbow (Standard)		1.46	1.89	2.37	2.71	3.04
	90° Welded Elbow (Long Radius)		0.69	0.88	1.1	1.26	1.43
	Screwed Tee or Cross (Flow turned 90°)		2.91	3.81	4.75	5.42	6.1
	Gate Valve (Straightway)		0.38	0.51	0.63	0.72	0.81
	Alarm or Non-Return Valve (Swinging)		2.42	3.18	3.94	4.5	5.07
	Alarm or Non-Return Valve (Mushroom)		12.08	18.91	19.71	22.54	25.36
	Globe Valve (Straightway)		16.43	21.64	26.8	30.64	34.48
	Butterfly Valve		2.19	2.86	3.55	4.06	4.56

Notes

1. Data taken from 29th Edition FOC Rules, Table 6124
2. The above values are used only for a Hazen-Williams **C Factor** of 120. For other values of **C factor**, the values are scaled in accordance with Table 6124 of the FOC Rules.
3. Equivalent lengths of 90mm fittings are the arithmetic means of the corresponding 80mm and 100mm fittings.

Pipe Sizes 125mm to 900mm

Code	Fitting Type	mm	125	150	200	250	300 - 1200
		in	5.0	6.0	8.0	10.0	12.0 - 48.0
1	45° Screwed Elbow		1.96	2.3	3.05	3.89	3.89
2	90° Screwed Elbow (Standard)		3.67	4.3	5.67	7.42	7.42
3	90° Welded Elbow (Long Radius)		1.72	2.0	2.64	3.35	3.35
4	Screwed Tee or Cross (Flow turned 90°)		7.36	8.61	11.34	14.85	14.85
5	Gate Valve (Straightway)		0.97	1.13	1.5	1.97	1.97
6	Alarm or Non-Return Valve (Swinging)		6.12	7.17	9.4	12.3	12.3
7	Alarm or Non-Return Valve (Mushroom)		30.62	35.88	47.27	61.85	61.85
8	Globe Valve (Straightway)		41.64	48.79	64.29	84.11	84.11
9	Butterfly Valve		5.47	6.38	8.62	9.9	9.9

Notes

1. Data taken from 29th Edition FOC Rules, Table 6124.
2. The above values are used only for a Hazen-Williams **C Factor** of 120. For other values of **C factor**, the values are scaled in accordance with Table 6124 of the FOC Rules.
3. All fittings above 250mm (10in) are assumed to have the same equivalent length as the corresponding 250mm fitting.

23.7 NFPA Rules

Equivalent Lengths (metres) of Fittings for NFPA Rules - Pipe Sizes 15.0mm to 40mm

Code	Fitting Type	Nominal Pipe Size					
		mm	15.0	20.0	25.0	32.0	40.0
		in	0.5	0.75	1.0	1.25	1.5
1	Standard 45° Elbow		0.3048	0.3048	0.3048	0.3048	0.6096
2	Standard 90° Elbow		0.6096	0.6096	0.6096	0.9144	1.2192
3	Long radius Elbow		0.3048	0.3048	0.6096	0.6096	0.6096
4	Tee or Cross (Flow turned 90°)		1.2192	1.2192	1.524	1.8288	2.4384
5	Gate Valve		0.3048	0.3048	0.3048	0.3048	0.3048
6	Swing Check Valve		1.2192	1.2192	1.524	2.1336	2.7432
7	Non-return Valve		4.3	4.3	4.3	4.3	4.3
8	Ball Valve		1.52	1.52	1.52	1.52	1.37
9	Butterfly Valve		1.8288	1.8288	1.8288	1.8288	1.8288

Notes

1. Equivalent lengths are taken from NFPA 15 (1990), except for fitting types 7 and 8 which are manufacturers' data.
2. The above values are used only with a Hazen-Williams [C factor](#) of 120. For other values of [C factor](#) the equivalent lengths are scaled in accordance with NFPA 15 Table A-7-2 (g).
3. Equivalent lengths of 15mm fittings are assumed the same as the corresponding 20mm fittings.
4. Equivalent lengths of gate and butterfly valves below 50mm are assumed the same as the 50mm values.

Pipe Sizes 50mm to 100mm

Code	Fitting Type	Nominal Pipe Size					
		mm	50.0	65.0	80.0	90.0	100.0
		in	2.0	2.5	3.0	3.5	4.0
1	Standard 45° Elbow		0.6096	0.9144	0.9144	0.9144	1.2192
2	Standard 90° Elbow		1.524	1.8288	2.1336	2.4384	3.048
3	Long radius Elbow		0.9144	1.2192	1.524	1.524	1.8288
4	Tee or Cross (Flow turned 90°)		3.048	3.6576	4.572	5.1816	6.096
5	Gate Valve		0.3048	0.3048	0.3048	0.3048	0.6096
6	Swing Check Valve		3.3528	4.2672	4.8768	5.7912	6.7056
7	Non-return Valve		4.3	5.2	6.1	6.9	7.7
8	Ball Valve		2.74	2.65	2.59	3.0	3.5
9	Butterfly Valve		1.8288	2.1336	3.048	3.6576	3.6576

Notes

1. Equivalent lengths are taken from NFPA 15 (1990), except for fitting types 7 and 8 which are manufacturers' data.
2. The above values are used only with a Hazen-Williams **C factor** of 120. For other values of **C factor** the equivalent lengths are scaled in accordance with NFPA 5 Table A-7-2 (g).
3. The equivalent length of a 90mm butterfly valve is assumed the same as the 100 mm valve.

Pipe Sizes 125mm to 300mm

Code	Fitting Type	Nominal Pipe Size					
		mm	125	150	200	250	300
		in	5.0	6.0	8.0	10.0	12.0
1	Standard 45° Elbow		1.5240	2.1336	2.7432	3.3528	3.9624
2	Standard 90° Elbow		3.6576	4.2672	5.4864	6.7056	8.2296
3	Long radius Elbow		2.4384	2.7432	3.9624	4.8768	5.4864
4	Tee or Cross (Flow turned 90°)		7.6200	9.1440	10.668	15.240	18.288
5	Gate Valve		0.6096	0.9144	1.2192	1.5240	1.8288
6	Swing Check Valve		8.2296	9.7536	13.716	16.764	19.812
7	Non-return Valve		10.000	12.200	15.300	19.800	23.000
8	Ball Valve		5.1000	6.7100	10.970	12.800	15.240
9	Butterfly Valve		2.7432	3.0480	3.6576	5.7912	6.4008

Notes

1. Equivalent lengths are taken from NFPA 15 (1990), except for fitting types 7 and 8 which are manufacturers' data.
2. The above values are used only with a Hazen-Williams [C factor](#) of 120. For other values of [C factor](#) the equivalent lengths are scaled in accordance with NFPA 15 Table A-7-2(g)

Pipe Sizes 350mm to 600mm

Code	Fitting Type	Nominal Pipe Size					
		mm	350	400	450	500	600
		in	14.0	16.0	18.0	20.0	24.0
1	Standard 45° Elbow		4.5720	5.1816	5.7912	6.4008	7.6200
2	Standard 90° Elbow		7.9248	9.1440	10.3632	11.5824	13.716
3	Long radius Elbow		6.4008	7.3152	8.2296	9.1440	10.668
4	Tee or Cross (Flow turned 90°)		20.4216	23.7744	27.1272	30.1752	36.576
5	Gate Valve		2.1336	2.4384	2.7432	3.0480	3.6576
6	Swing Check Valve		38.100	44.196	50.292	60.960	67.056
7	Non-return Valve		23.000	23.000	23.000	23.000	23.000
8	Ball Valve		15.240	15.240	15.240	15.240	15.240
9	Butterfly Valve		7.3152	7.9248	9.1440	10.668	13.4112

Notes

1. The above values are used only with a Hazen-Williams [C factor](#) of 120. For other values of [C factor](#) the equivalent lengths are scaled in accordance with NFPA 15 Table A-7-2 (g)
2. Equivalent lengths for these sizes are taken from Factory Mutual data sheet 2.89 except for fittings types 7 and 8, which are manufacturer's data for 300 mm valves.

Pipe Sizes 700mm to 900mm

Code	Fitting Type	Nominal Pipe Size					
		mm	700	750	800	850	900
		in	28.0	30.0	32.0	34.0	36.0
1	Standard 45° Elbow		9.0082	9.6516	10.2951	10.9385	11.5819
2	Standard 90° Elbow		15.7566	16.8821	18.0075	19.1330	20.2585
3	Long radius Elbow		12.6684	13.5733	14.4782	15.3831	16.2879
4	Tee or Cross (Flow turned 90°)		42.2380	45.2550	48.2720	51.2890	54.3060
5	Gate Valve		4.2061	4.5066	4.8070	5.1074	5.4079
6	Swing Check Valve		81.3194	87.1280	92.9365	98.7451	104.5536
7	Non-return Valve		23.000	23.000	23.000	23.000	23.000
8	Ball Valve		15.240	15.240	15.240	15.240	15.240
9	Butterfly Valve		13.4112	13.4112	13.4112	13.4112	13.4112

Notes

1. The above values are used only with a Hazen-Williams [C factor](#) of 120. For other values of [C factor](#) the equivalent lengths are scaled in accordance with NFPA 15 Table A-7-2 (g).
2. Equivalent lengths for pipe sizes above 600mm have been obtained by extrapolation from equivalent lengths for the smaller pipe sizes.

Pipe Sizes 950mm to 1200mm

Code	Fitting Type	Nominal Pipe Size						
		mm	950	1000	1050	1100	1150	1200
		in	38.0	40.0	42.0	44.0	46.0	48.0
1	Standard 45° Elbow		12.225	12.869	13.512	14.156	14.799	15.442
2	Standard 90° Elbow		21.384	22.510	23.635	24.7610	25.886	27.012
3	Long radius Elbow		17.193	18.098	19.002	19.907	20.812	21.717
4	Tee or Cross (Flow turned 90°)		57.323	60.340	63.357	66.374	69.391	72.408
5	Gate Valve		5.7084	6.0089	6.3094	6.6099	6.9104	7.2109
6	Swing Check Valve		110.362	116.171	121.979	127.788	133.596	139.405
7	Non-return Valve		23.000	23.000	23.000	23.000	23.000	23.000
8	Ball Valve		15.240	15.240	15.240	15.240	15.240	15.240
9	Butterfly Valve		13.4112	13.4112	13.4112	13.4112	13.4112	13.4112

Notes

1. The above values are used only with a Hazen-Williams [C factor](#) of 120. For other values of [C factor](#) the equivalent lengths are scaled in accordance with NFPA 15 Table A-7-2 (g).
2. Equivalent lengths for pipe sizes above 600mm have been obtained by extrapolation from equivalent lengths for the smaller pipe sizes.

23.8 Chinese Standard (GB)

Equivalent Lengths (metres) of Fittings

With Chinese Standard (GB) options, the equivalent lengths (metres) of fittings are same as the table for NFPA rules, but without the C-factor limitations.

23.9 Schedule - BS 3505 Class D (uPVC)

Surface roughness $1.524 \times 10^{-3} \text{mm}$ ($6.0 \times 10^{-5} \text{in}$)

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
15	0.5				
20	0.75				
25	1.0				
32	1.25	37.65	1.481	42.25	1.663
40	1.5	42.95	1.692	48.25	1.9
50	2.0	53.85	2.12	60.35	2.376
65	2.5	67.15	2.644	75.35	2.967
80	3.0	79.2	3.118	88.9	3.5
90	3.5				
100	4.0	101.7	4.004	114.3	4.5
125	5.0	124.9	4.918	140.2	5.5201
150	6.0	149.75	5.895	168.25	6.624
200	8.0	197.5	7.775	219.1	8.6260
250	10.0	246.1	9.689	273.0	10.748
300	12.0	291.95	11.495	323.85	12.75
350	14.0	320.4	12.6145	355.5	13.996
400	16.0	366.5	14.4285	406.4	16.0
450	18.0	412.2	16.2275	457.2	18.0

Notes

1. Internal and Outside diameters are arithmetic means of values given in BS.3505 - (1968).
2. BS.3505 does not include metric nominal sizes; these have been retained for users' convenience.

23.10 Schedule - BS 1387 Heavy

Surface roughness 0.04572mm (0.0018 ins)

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
15	0.5	14.9	0.5875	21.4	0.8435
20	0.75	20.4	0.8035	26.9	1.0595
25	1.0	25.7	1.011	33.8	1.331
32	1.25	34.4	1.352	42.5	1.672
40	1.5	40.3	1.584	48.4	1.904
50	2.0	51.3	2.022	60.3	2.374
65	2.5	67.0	2.6395	76.0	2.9915
80	3.0	79.1	3.1125	88.8	3.4965
90	3.5				
100	4.0	103.3	4.0675	114.1	4.4915
125	5.0	128.85	5.0725	139.65	5.4965
150	6.0	154.3	6.075	165.1	6.499
200	8.0	200.0	8.0	200.0	8.0
250	10.0	250.0	10.0	250.0	10.0
300	12.0	300.0	12.0	300.0	12.0

Notes

1. Internal and outside diameters are arithmetic means of values given in BS.1387 - (1967).
2. Nominal sizes have been used for internal and outside diameters for pipes in the range 200–300mm (8–12 inch) for users' convenience.
3. This data is not part of BS.1387.

23.11 Schedule - BS 1387 Medium

Surface roughness 0.04572mm (0.0018 ins)

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
15	0.5	16.1	0.6355	21.4	0.8435
20	0.75	21.6	0.8515	26.9	1.0595
25	1.0	27.3	1.075	33.8	1.331
32	1.25	36.0	1.416	42.5	1.672
40	1.5	41.9	1.648	48.4	1.904
50	2.0	53.0	2.086	60.3	2.374
65	2.5	68.7	2.7035	76.0	2.9915
80	3.0	80.7	3.1765	88.8	3.4965
90	3.5				
100	4.0	105.1	4.1395	114.1	4.4915
125	5.0	129.95	5.1125	139.65	5.4965
150	6.0	155.4	6.115	165.1	6.499
200	8.0	200.0	8.0	200.0	8.0
250	10.0	250.0	10.0	250.0	10.0
300	12.0	300.0	12.0	300.0	12.0

Notes

1. Internal and outside diameters are arithmetic means of values given in BS.1387 - (1967).
2. Nominal sizes have been used for internal and outside diameters for pipes in the range 200–300mm. (8–12 inch) for users' convenience.
3. This data is not part of BS.1387.

23.12 Schedule - ANSI B36.10 Schedule 40

Surface roughness 0.04572mm (0.0018 ins)

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
15	0.5	15.76	0.622	21.3	0.84
20	0.75	20.96	0.824	26.7	1.05
25	1.0	26.64	1.049	33.4	1.315
32	1.25	35.08	1.38	42.2	1.66
40	1.5	40.94	1.61	48.3	1.9
50	2.0	52.48	2.067	60.3	2.375
65	2.5	62.68	2.469	73.0	2.875
80	3.0	77.92	3.068	88.9	3.5
90	3.5	90.12	3.548	101.6	4
100	4.0	102.26	4.026	114.3	4.5
125	5.0	128.2	5.047	141.3	5.563
150	6.0	154.08	6.065	168.3	6.625
200	8.0	202.74	7.981	219.1	8.625
250	10.0	254.56	10.02	273.1	10.75
300	12.0	303.28	11.938	323.9	12.75
350	14.0	333.34	13.124	355.6	14.0
400	16.0	381.0	15	406.4	16.0
450	18.0	428.46	16.876	457.0	18.0
500	20.0	477.82	18.812	508.0	20.0
600	24.0	575.04	22.624	610.0	24.0

Notes

1. Internal and outside diameters are taken from ANSI B36.10 - (1979).
2. ANSI B36.10 does not include metric nominal sizes; however, these have been retained for users' convenience.

23.13 Schedule - ANSI B36.10 Schedule 80

Surface roughness 0.04572mm (0.0018 ins)

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
15	0.5	13.84	0.546	21.3	0.84
20	0.75	18.88	0.742	26.7	1.05
25	1.0	24.3	0.957	33.4	1.315
32	1.25	32.5	1.278	42.2	1.66
40	1.5	38.14	1.5	48.3	1.9
50	2.0	49.22	1.939	60.3	2.375
65	2.5	58.98	2.323	73.0	2.875
80	3.0	73.66	2.9	88.9	3.5
90	3.5	85.44	3.364	101.6	4.0
100	4.0	97.18	3.826	114.3	4.5
125	5.0	122.24	4.813	141.3	5.563
150	6.0	146.36	5.761	168.3	6.625
200	8.0	193.67	7.625	219.1	8.625
250	10.0	242.92	9.562	273.1	10.75
300	12.0	288.94	11.374	323.9	12.75
350	14.0	317.5	12.5	355.6	14.0
400	16.0	363.52	14.312	406.4	16.0
450	18.0	409.34	16.124	457.0	18.0
500	20.0	455.62	17.938	508.0	20.0
600	24.0	548.08	21.562	610.0	24.0

Notes

1. Internal and outside diameters are taken from ANSI B36.10 - (1979).
2. ANSI B36.10 does not include metric nominal sizes; however, these have been retained for users' convenience.

23.14 Schedule - BS 1211 Class C (cast Iron)

Surface roughness 0.254mm (0.01 ins)

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
15	0.5				
20	0.75				
25	1.0				
32	1.25				
40	1.5				
50	2.0				
65	2.5				
80	3.0	80.772	3.18	95.504	3.76
90	3.5				
100	4.0	106.172	4.18	121.92	4.8
125	5.0	132.588	5.22	149.86	5.9
150	6.0	158.496	6.24	177.292	6.98
200	8.0	210.312	8.28	232.156	9.14
250	10.0	262.128	10.32	286.004	11.26
300	12.0	319.024	12.56	345.44	13.6
350	14.0	370.84	14.6	399.288	15.72
400	16.0	422.656	16.64	453.136	17.84
450	18.0	474.472	18.68	506.984	19.96
500	20.0	526.288	20.72	560.3241	22.06
600	24.0	629.412	24.78	667.004	26.26

Notes

1. Imperial sizes taken from BS.1211 - (1958).
2. This standard does not include metric sizes; these have been obtained by direct conversion of the internal and outside diameters given in inches.

23.15 Schedule - Copper/Nickel (90/10) 14 bar g

Surface roughness $1.524 \times 10^{-3} \text{mm}$ ($6.0 \times 10^{-5} \text{in}$)

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
15	0.5	14.0	0.5512	16.0	0.6299
20	0.75	22.0	0.8661	25.0	0.9843
25	1.0	27.0	1.063	30.0	1.1811
32	1.25	35.0	1.378	38.0	1.4961
40	1.5	41.5	1.6339	44.5	1.752
50	2.0	54.0	2.126	57.0	2.2441
65	2.5				
80	3.0	72.1	2.8386	76.1	2.9961
90	3.5	83.9	3.3031	88.9	3.5
100	4.0	103.0	4.0551	108.0	4.252
125	5.0				
150	6.0	154.0	6.063	159.0	6.2598
200	8.0	212.1	8.3504	219.1	8.6260
250	10.0	259.0	10.1969	267.0	10.5118
300	12.0	313.9	12.3583	323.9	12.752
350	14.0	357.0	14.0551	368.0	14.4882
400	16.0	407.0	16.0236	419.0	16.4961
450	18.0	443.2	17.4488	457.2	18.0
500	20.0	493.0	19.4094	508.0	20.0
600	24.0	592.0	23.3071	610.0	24.0157

Notes

1. Outside diameters to BS.2871, Part 2, Table 3; BSMA 18, Table 2; BSMA 60. Wall thicknesses conform to Lloyds offshore requirements.
2. Imperial sizes are direct conversions of the metric data.
3. Nominal sizes are included for users' convenience.

23.16 Schedule - Copper/Nickel (90/10) 20 Bar g

Surface roughness $1.524 \times 10^{-3} \text{mm}$ ($6.0 \times 10^{-5} \text{in}$)

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
15	0.5	14.0	0.5512	16.0	0.6299
20	0.75	22.0	0.8661	25.0	0.9843
25	1.0	27.0	1.063	30.0	1.1811
32	1.25	35.0	1.378	38.0	1.4961
40	1.5	41.5	1.6339	44.5	1.752
50	2.0	54.0	2.126	57.0	2.2441
65	2.5				
80	3.0	72.1	2.8386	76.1	2.9961
90	3.5	83.9	3.3031	88.9	3.5
100	4.0	103.0	4.0551	108.0	4.252
125	5.0				
150	6.0	152.0	5.9843	159.0	6.2598
200	8.0	210.1	8.2717	219.1	8.6260
250	10.0	256.0	10.0787	267.0	10.5118
300	12.0	309.9	12.2008	323.9	12.752
350	14.0	352.0	13.8583	368.0	14.4882
400	16.0	401.0	15.7874	419.0	16.4961
450	18.0	438.2	17.252	457.2	18.0
500	20.0	486.0	19.1339	508.0	20.0
600	24.0	584.0	22.9921	610.0	24.0157

Notes

1. Outside diameters to BS.2871, Part 2, Table 3; BSMA 18, Table 2; BSMA 60. Wall thicknesses conform to Lloyds offshore requirements.
2. Imperial sizes are direct conversions of the metric data.
3. Nominal sizes are included for users' convenience.

Index

- . -

.DXF 96
.EMF 96
.WMF 96

- < -

<Any> 33

- A -

A maintenance area sprinkler system 88
A small sprinkler system 54
Add multiple nozzles 28
Add multiple pipes 28, 140
Add to report 110
Adding a pump 157
Adding a schedule 155
Adding components to pipes 103
Adding specification 118
Adding waypoints 97
Area tool 98
AutoCAD 206
Autolayout 144
Autosave 21

- B -

Background colour 172
Background colours 132
Basic Errors 210
Bernoulli 184
Bibliography 216
Blocks 118
Breaks 118
Browse 27
BS1042 191

- C -

Calculate 27
Calculation menu 27
Calculation options 26, 174
Calculation phase 39
Calculation toolbar 32
Cavitation 193
Cell shading 110
C-factor 185, 218
Changing units 167
Check 27
Checking specifications 118
Chinese Standard (GB) 164, 182, 191, 230
Closing a file 21
Colour schemes 128
Colouration menu 27
Complex colouring rules 128
Complex links 27
Complex nodes 27
Component properties 17
Component tools 102
Condensate tank deluge system 68
Constructing a path 134
Contacting Sunrise Systems 4
Control valves library 25
Convergence accuracy 174
Copy 23, 97
Copy/Paste 97
Copying cells 111
Copying selected items 98, 100
Coulson-Richardson 182, 184, 185
Crane 191
Creating a pipe type 171
Creating sub-networks 144
Crosses 199
Cubic pump curve 193
Cubic spline 193
Cursor keys 105
Cut 23

- D -

Defaults 177
Defaults options 26

Degeneration factor 193
 Deleteing a pump 157
 Deleteing a schedule 155
 Deleting a pipe type 171
 Deluge valves 188
 Deluge valves library 161
 Design facility 180
 Design phase 39
 Display options 26, 172
 Display precisions 167
 Drawing components 102
 DXF 206
 DXF file 105

- E -

Edit menu 23
 Editing a cell 110
 Editing a pipe type 171
 Editing a pump 157
 Editing a schedule 155
 Editing system libraries 25, 161
 Elastomeric valve 190
 Elevation profile 134
 Equipment items 187
 Errors 210
 Examples directory 3
 Exit the program 21
 Expired licence 8
 Expiry date checking 8
 Expiry date licencing 8
 Explode node 97
 Export 21
 Export clipboard 28
 Export DXF file 28
 Export HP-GL/2 28
 Exporting libraries 152
 Exporting the schematic 105, 206
 Exporting to an HPGL file 105

- F -

Fanning friction factor 184
 File menu 21
 Filters 187
 Find button 33

Finding components 33
 Fittings 199
 Fittings library 25
 Fittings window 19
 Fluids library 25
 FOC 164
 FOC rules 220
 Font sizes 172
 Frictional pressure loss 183, 184

- G -

Global errors 211
 Grid display options 172

- H -

Hazen Williams 183
 Hazen-Williams 182, 185
 Hazen-Williams coefficients 218
 Height check errors 134
 Height check tolerance 174
 Height checking 124, 125
 Help topics 29
 Henri Darcy 184
 Heriot-Watt 191
 HPGL 105
 HP-GL/2 206
 Hydraulic grade line 136

- I -

I/O nodes 37
 Import graphical underlay 24
 Import library 21
 Importing libraries 152
 Including validation in output file 176
 Inlets 37
 Input data 27
 Input errors 211
 Installation 8
 Invalid components 47, 102, 124
 Invert 23

- K -

Key status 29

- L -

Label display options 172

Labelling the diagram 36

Labels 36

Laminar flow 184

Libraries 152

Library editor 153

License 8

Line thickness 172

Lines per page 174

Linings 182

Linings library 160

Local user library 152

- M -

Make a path 28

Mass balance 180

Mass flow 167

Mirror 23

Module options 26, 164

Moody diagram 184

Most remote nozzle 117

Moving and editing text 101

Moving around the network 105

- N -

Network representation 36

Network topology 38

Next button 33

NFPA 164

NFPA 96 164

NFPA rules 224

NFPA Submittal Report 27, 202

No pipe type 171

Non-return valves 189

Nozzle generation 148

Nozzle library 156

Nozzle operating point 16

Nozzle specification 121

Nozzles 198

NPSH 193

NPSHA 193

NPSHR 193

Number of iterations 174

Numerical errors 214

- O -

Obtaining support 4

Open library 21

Opening a file 21

Opening libraries 152

Options menu 26

Options toolbar 31

Orifice plates 191

Outlets 37

Output tables options 26

Overall appearance 14

Overboard dump valves 190

Overboard dump valves and specifications 190

Overview window 20

- P -

Pan 98

Panning with the mouse 107

Paste 23, 97

Paste (Incl. layout) 97

Paste in column 23

Physical constants 220

Pipe component tools 103

Pipe grouping 39

Pipe hydrostatic pressure loss 185

Pipe modelling 182

Pipe schedules 155

Pipe sizing 27

Pipe type option 26

Pipe types 171

Polygon tool 100

Prerequisites 8

Pressures in a ring main 80

Print preview 21, 104

Print setup 104

Printing the grid 110

Printing the schematic 21
 Product modules 4
 Properties of water 216
 Properties window 16
 Property window scroll buttons 17
 Pump library - coefficients known 159
 Pump library - coefficients unknown 157
 Pump operating point 16
 Pumps 193
 Pumps - coefficients unknown 157
 Pumps library 25

- Q -

Quadratic pump curve 193

- R -

Redo 23
 Reference node 174
 Reference node height 174
 Remote nozzle 180
 Remote update 8
 Remove tags 28
 Removing specifications 118
 Results display options 172
 Roughness 217

- S -

Save as 21
 Saving a file 21
 Schedule - ANSI B36.10 Schedule 40 233
 Schedule - ANSI B36.10 Schedule 80 234
 Schedule - BS 1211 Class C (cast Iron) 235
 Schedule - BS 1387 Heavy 231
 Schedule - BS 1387 Medium 232
 Schedule - BS 3505 Class D (uPVC) 230
 Schedule - Copper/Nickel (90/10) 14 bar g 236
 Schedule - Copper/Nickel (90/10) 20 bar g 237
 Schedules library 25
 Schematic diagrams 36
 Schematic Overview window 20
 Schematic printing 104
 Schematic underlay 96
 Schematic window 15, 96

Scroll buttons 17
 Security key 8
 Security key problems 9
 Selected items 98, 100
 Selecting a row 110
 Selecting several components 97
 Selection tool 97
 Simple colouring rules 128
 Simple rules 27
 Simple three pipe system 38
 Smooth pump curve 193
 Sorting in the grid 110
 Spec. for calculation 27
 Specific component errors 213
 Specification checks 124
 Specification rules 117
 Specifications 116, 174
 Specifications - user interface 118
 Spray options 164
 Spray tables 176
 Spray toolbar 30
 Status checking 124
 Status dialog 124
 Support 4
 Surface roughness 217
 System library 152

- T -

Tables 176
 Tabular view 16, 110
 Tag selected items 28
 Tag toolbar 32, 75
 Tagged labels 75
 Tagging 132
 Tees 199
 Temporary path 174
 Text tool 101
 Title 164
 Title option 26
 Tool tips 172
 Tools menu 28
 Transitional flow 184
 Turbulent flow 184

- U -

Undefined components 47, 102
Undo 23
Undo/Redo buttons 23
Unit conversion tool 167
Units 167
Units option 26
Update key 29
Updating a license 8
Use PIPENET menu style 29
Use Windows menu style 29

- V -

Velocity pressure 164, 185
Version numbers 29
View graphical underlaid 24
View menu 24
View palette 24
View schematic overview 24
View schematic window 24
view status bar 24
View toolbar 24
Vista 3
Volumetric flow 167

- W -

Waypoints 102
Welcome 3
Window menu 29

- Z -

Zoom 98
Zooming with the mouse 107