

PIPENET VISION™

Transient Module
User and Reference Manual

Software Revision 1.5



© 2010 Sunrise Systems Limited

Foreword

Transient User and Reference Manual

Sunrise Systems Limited

This manual is designed to act as a user and reference guide for the Transient 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	16
6 Fittings window	19
7 The Schematic Overview Window	19
8 Menus	20
File Menu	20
Edit Menu	22
View Menu	23
Libraries Menu	24
Options Menu	25
Colouration Menu	26
Calculation Menu	26
Output Menu	26
Tools Menu	27
Window Menu	27
Help Menu	28
9 Toolbars	29
Transient Toolbar	29
Options Toolbar	30
Calculation Toolbar	31
Tag Toolbar	31
Find toolbar	32

Part 4 Tutorial Example 1 - One Pipe System	34
1 Labelling the diagram	34
2 Labels	34
3 Problem description	34
4 Stage 1 - Problem initialisation	35
5 Stage 2 - Entering The Network	38
6 Stage 3 - Checking the network	47
7 Stage 4 - Saving the network	47
8 Stage 5 - Performing a calculation	47
9 Stage 6 - Viewing the results	50
Part 5 Tutorial Example 2 - Small Sprinkler System	54
1 Problem description	54
2 Stage 1 - Adding a pump to the library	55
3 Stage 2 - Problem initialisation	56
4 Stage 3 - Entering the network details	58
5 Stage 4 - Defining output tables and graphs	61
6 Stage 5 - Performing a calculation	66
Part 6 Tutorial Example 3 - Force on a 90 degree Bend	70
1 Problem description	70
2 Stage 1 - Creating the library items	71
3 Stage 2 - Problem Initialisation	74
4 Stage 3 - Entering the network details	75
5 Stage 4 - Specifications	76
6 Stage 5 - Defining forces	76
7 Stage 6 - Specifying tables	77
8 Stage 7 - Specifying graphs	79
9 Stage 8 - Performing the calculation	80
Part 7 Other Topics	86
1 Forces	86
2 Time control and time steps	89
3 Variable time step	92
4 Vapour cavitation	95
5 Channel cavitation	96
6 Long sections	100

7 Pressure envelopes	102
Part 8 The Schematic	106
1 Schematic Window	106
2 Schematic Underlay	106
3 Selection Tool	107
4 Pan and Zoom Tool	108
5 Area Tool	108
6 Polygon Tool	110
7 Text Tool	111
8 Link Component Tools	112
9 Pipe Component Tools	113
10 Schematic Printing	113
11 Exporting the Schematic	115
12 Moving around the network	115
13 Use of the mouse	116
Part 9 The Tabular View	120
1 Tabular Grid	120
2 Validation	122
3 Copying Cells	122
Part 10 Status Checking	126
1 Status checking	126
2 Height Checking	126
Part 11 Colour Schemes	130
1 Colour Schemes	130
2 Tagging	133
3 Background Colours	134
Part 12 Elevation Profile and Pressure Envelope	136
1 Elevation Profile Window	136
Part 13 Add multiple pipes	140
1 Add Multiple Pipes	140
Part 14 Libraries	144
1 Libraries	144
2 Schedules	146
3 Fittings	147

4	Control Valves	149
5	Pumps - Coefficients Unknown	150
6	Pumps - Coefficients Known	152
7	Pumps Turbo	153
8	Linings	154
9	General Pressure Losses	155
10	Editing System Libraries	156
Part 15	Specifying Options	158
1	Title	158
2	Transient options	158
3	Units	159
4	Fluid	163
5	API Fluid	165
6	Pipe types	167
7	Schematic display options	168
Part 16	Performing a Calculation	172
1	Transient calculations	172
2	Selecting graphs for components	172
3	Defining Tables	173
4	Defining Forces	175
5	Displaying Results	176
6	Controls	177
7	Absolute tolerances	177
8	Relative tolerances	178
9	Initial state	179
10	Transient Output	179
11	Maximum and minimum pressure	181
Part 17	Graph Viewer	184
1	Graph Viewer introduction	184
2	Different graph types	187
3	Using the property tabs	189
4	Curve property tab	190
5	Y Axis property tab	192
6	X Axis property tab	193
7	Limit Lines properties tab	194
8	Labels and Arrows	196
9	Hot spot	197

10 Zooming	198
Part 18 Modelling	202
1 Standard and Spray options	202
2 HazenWilliams	202
3 Coulson and Richardson	203
4 Pipe C-factor	204
5 Linings	204
6 Fluids	204
7 Fluid Equations	206
8 Wave Speed	207
9 Specifications	208
10 Initial Guess File and the initial state	213
11 Elastic Pipe	214
12 Short Pipe	216
13 Pipe Bundle	217
14 Compressible Pipe	222
15 Fittings and equipment items	223
16 Simple Pump	225
17 Turbo Pump	229
18 Inertial Pump	233
19 Valves	241
20 Operating Valve	244
21 Non-return valve	245
22 Check Valve	246
23 Fluid Damped Check Valve	248
24 Liquid Surge Relief Valve	250
25 Regulator Valve	252
26 Inertial Check Valve	254
27 Bursting disc	255
28 Pressure surge relief valve	257
29 Accumulator	261
30 SurgeTank	264
31 Receiving Vessel	266
32 Simple Tank	270
33 Vacuum Breaker	271
34 One-node caisson	275
35 Two-node caisson - type 1	276
36 Two-node caisson - type 2	277
37 General Pressure Loss	280

38	Control Systems	283
39	Sensors	284
40	Transfer Function	287
41	PID Controller	289
42	Cascade PID Controller	292
43	Switch	295
44	Signal Selector	299
Part 19	Exporting the Schematic	302
1	HP-GL/2 Output	302
2	DXF Output	302
Part 20	Errors	306
1	Errors	306
2	Basic errors	306
3	Input errors	307
4	Global errors	307
5	Specific component errors	309
6	Numerical errors	310
Part 21	Reference Data	312
1	Bibliography	312
2	Surface Roughness	312
3	Hazen-Williams Coefficients	314
4	Physical Constants	315
5	Built-in schedules and fittings	316
6	ANSI B36.10 Schedule 10	318
7	ANSI B36.10 Schedule 20	319
8	ANSI B36.10 Schedule 30	320
9	ANSI B36.10 Schedule S	321
10	ANSI B36.10 Schedule 40	322
11	ANSI B36.10 Schedule 60	323
12	ANSI B36.10 Schedule XS	324
13	ANSI B36.10 Schedule 80	325
14	ANSI B36.10 Schedule 100	326
15	ANSI B36.10 Schedule 120	327
16	ANSI B36.10 Schedule 140	328
17	ANSI B36.10 Schedule 160	329
18	ANSI B36.19 Schedule 5S	330
19	ANSI B36.10 Schedule XXS	331

20	ANSI B36.19 Schedule 10S	332
21	ANSI B36.19 Schedule S	333
22	ANSI B36.19 Schedule XS	334
23	ANSI B36.19 Schedule XXS	335
24	JIS 1990 Steel Schedule 10	336
25	JIS 1990 Steel Schedule 20	337
26	JIS 1990 Steel Schedule 30	338
27	JIS 1990 Steel Schedule 40	339
28	JIS 1990 Steel Schedule 60	340
29	JIS 1990 Steel Schedule 80	341
30	JIS 1990 Steel Schedule 100	342
31	JIS 1990 Steel Schedule 120	343
32	JIS 1990 Steel Schedule 140	344
33	JIS 1990 Steel Schedule 160	345
34	JIS 1990 Steel Schedule 5S	346
35	JIS 1990 Steel Schedule 10S	347
36	JIS 1990 Steel Schedule 40S	348
37	DIN Standard	349
38	Pipe Schedule 1 (SPRAY) BS 1387 Heavy	350
39	Pipe Schedule 2 (SPRAY) BS 1387 Medium	351
40	Pipe Schedule 3 (SPRAY) ANSI B36.10 Schedule 40	351
41	Pipe Schedule 4 (SPRAY) ANSI B36.10 Schedule 80	353
42	Pipe Schedule 5 (SPRAY) BS3505 Class D	354
43	Pipe Schedule 6 (SPRAY) BS.1221 Class C	355
44	Pipe Schedule 7 (SPRAY) BS.2871	356
45	Pipe Schedule 8 (SPRAY) BS.2871	357
46	Lining Type	358
47	K-factor - Type of Fitting or Valve	358
48	Entry and Exit K-factors	358
49	Equivalent Lengths (FOC)	360
50	Equivalent Lengths (NFPA)	362
	Index	365

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 pipenet@sunrise-sys.com (general enquiries)
email support@sunrise-sys.com (customer support queries)
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 250M 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.

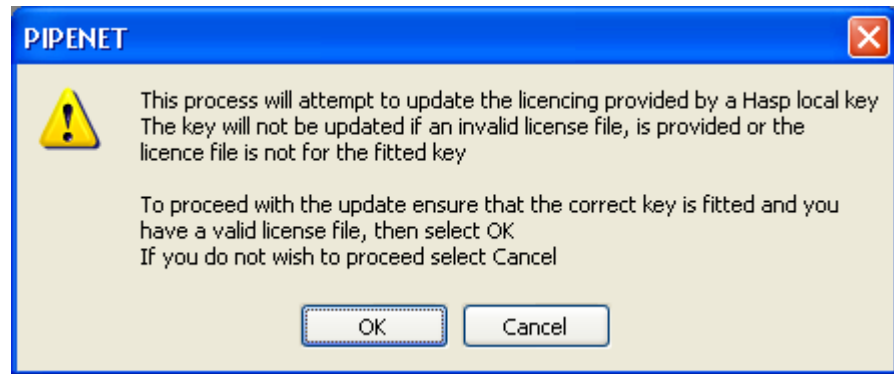
Expiry dates

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.

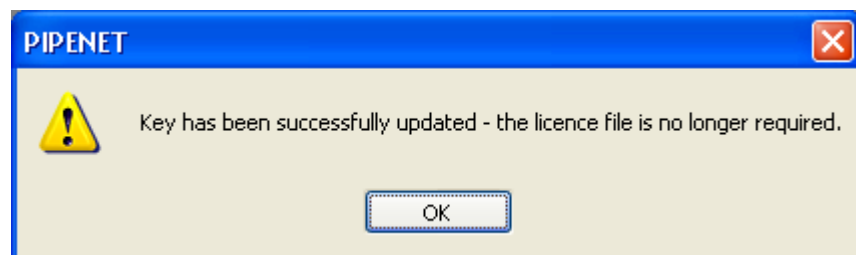
Updating a licence file

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 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

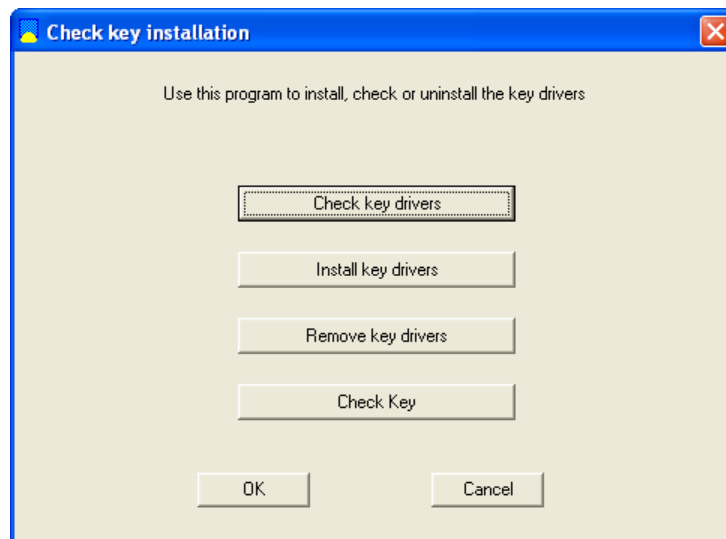
If you have followed all of the instructions above and you have a key licence for the module you are trying to run then you should have no further problems. However, if you are still having problems then please perform the following checks and email the results Sunrise Systems. Be sure to include the 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

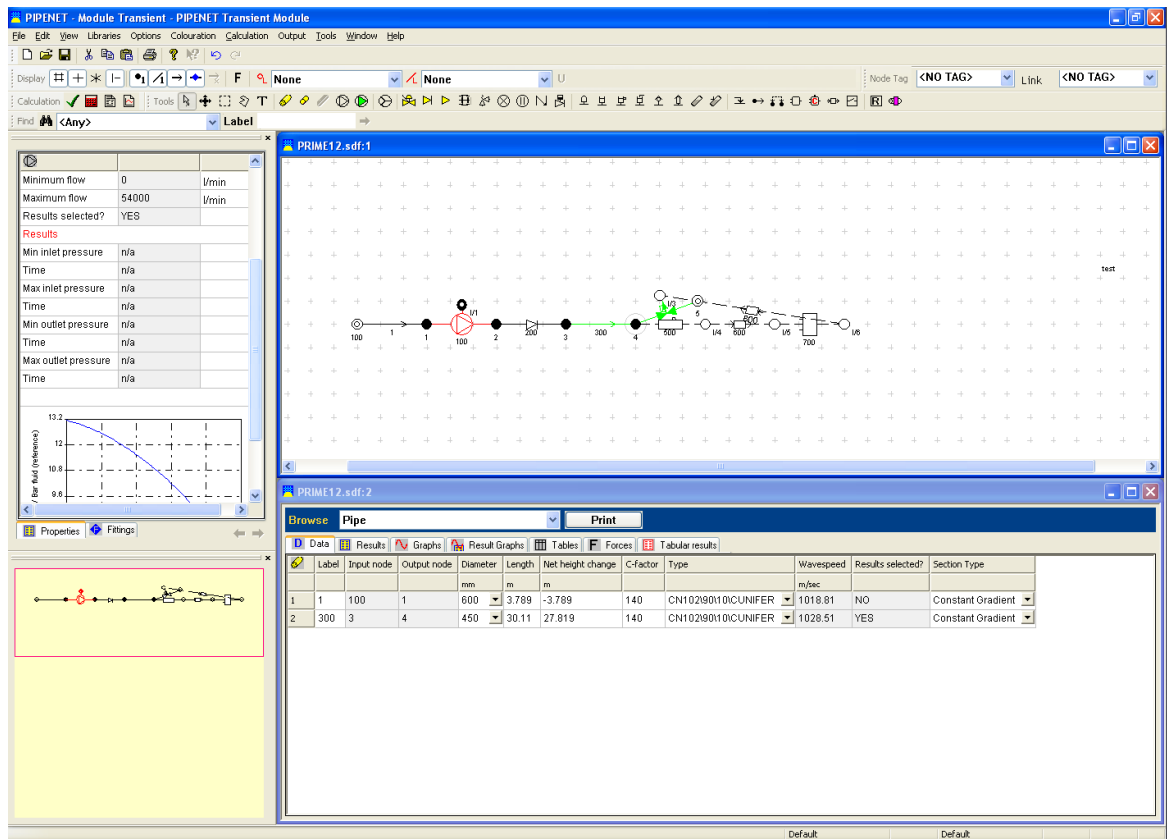
Part

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: 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.
- Lower-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.

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.

One other window is available the [Elevation Profile Window](#) which can be used to plot the elevation along a selected flow path defined by a sequence of nodes.

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.

3.5 Component Properties

This window displays the properties associated with the currently selected component. If no component is selected the window is blank.

Label	3	
Input node	6	
Output node	5	
Diameter	100	mm
Length	50	m
Net height change	0	m
Roughness	0.0457	mm
Additional k-factor	0	
Type	AnsiB3610...	
Wavespeed	1429.03	m/sec
Results selected?	NO	
Section Type	Constant Gr...	
Results		
Min inlet pressure	-0.6397986	Bar G
Time	6.2	sec
Max inlet pressure	14.5540229	Bar G
Time	6.06	sec
Min internal press...	-0.8897368	Bar G
Time	6.19	sec
Max internal press...	19.6303789	Bar G
Time	6.03	sec
Min outlet pressure	-0.8897368	Bar G
Time	6.19	sec
Max outlet pressure	19.6303789	Bar G
Time	6.03	sec

If a calculation has been performed the Properties window will also display a summary of minimum and maximum pressures at the inlet and outlets For pipes, the maximum and minimum internal pressures are also displayed.

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.

The above example shows the attributes for a pipe. All attributes can be edited. The length and elevation of the pipe are shown in metres and the diameter in millimetres.

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

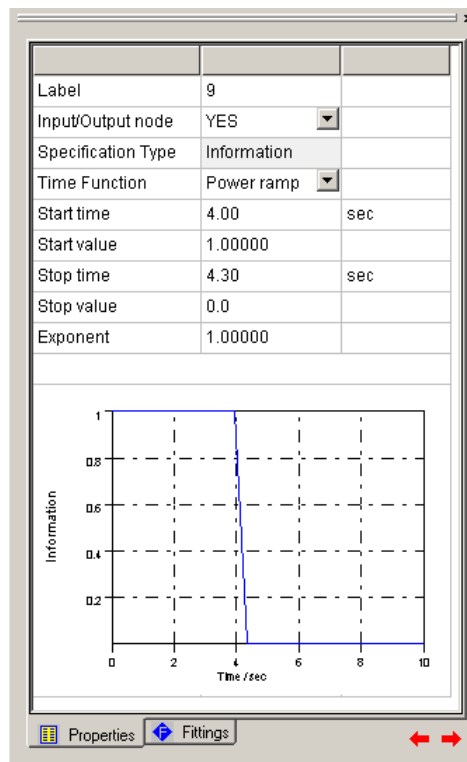
In the bottom right-hand corner of the window are two scroll buttons, these are used to move from the currently displayed component to the next or previous component of the same type.



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.

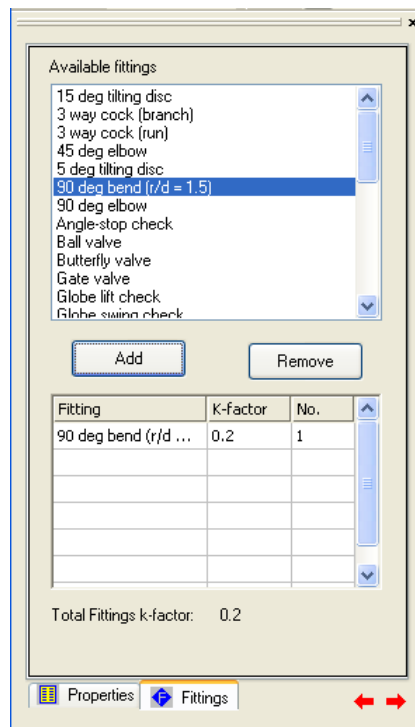
Display of accompanying graph

Some components may have an accompanying graph, if so this is also displayed in the Properties window. For example, selecting a node with a specification will display the specification curve.



3.6 Fittings window

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; below the window is shown the sum of the K-factors for all fittings.

Adding a fitting

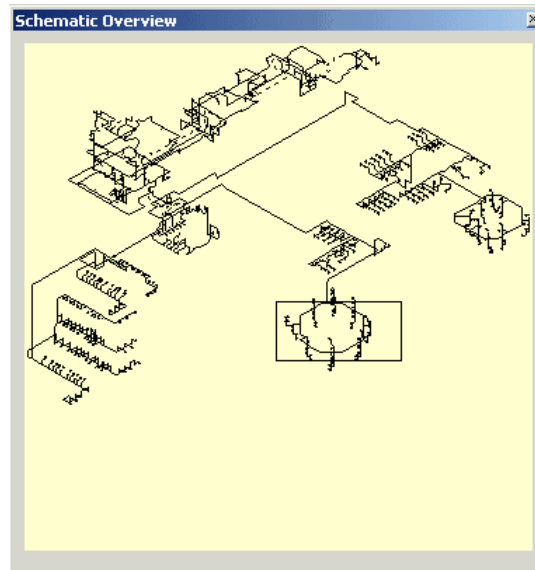
To add a fitting to a pipe select the desired fitting from the top window, and then the **Add** button. Each selection of the Add button adds one fitting of the selected type to the pipe.

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 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.

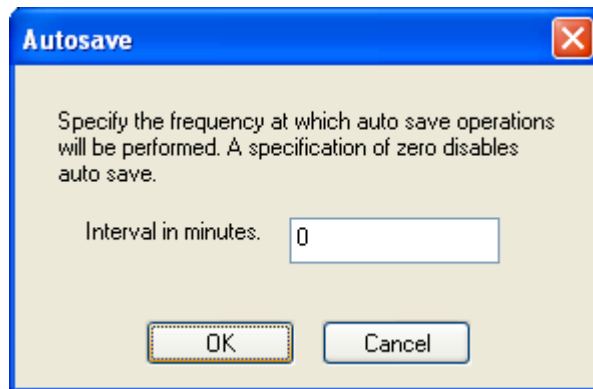
Export...

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

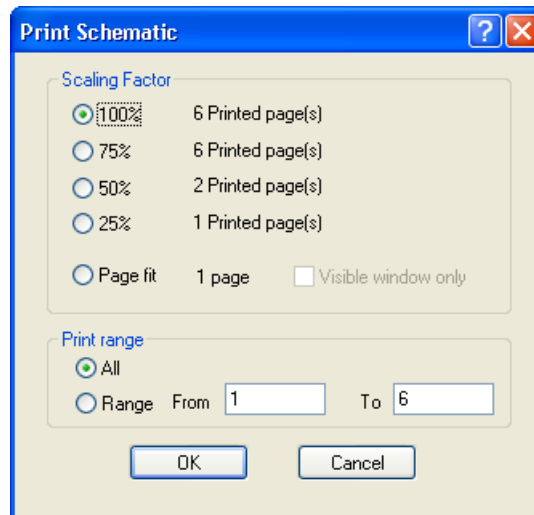
Opens an external [system library](#).

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

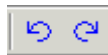
Used with the [Area Tool](#) to mirror the contents of the selected area left-right.

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, 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 .EMF file.

Import graphical underlay

Import a graphical underlay from a Windows extended 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 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.

Fittings

Define or edit pipe fittings in the local user library.

Valves

Define or edit valves in the local user library.

Pumps - coefficients unknown

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

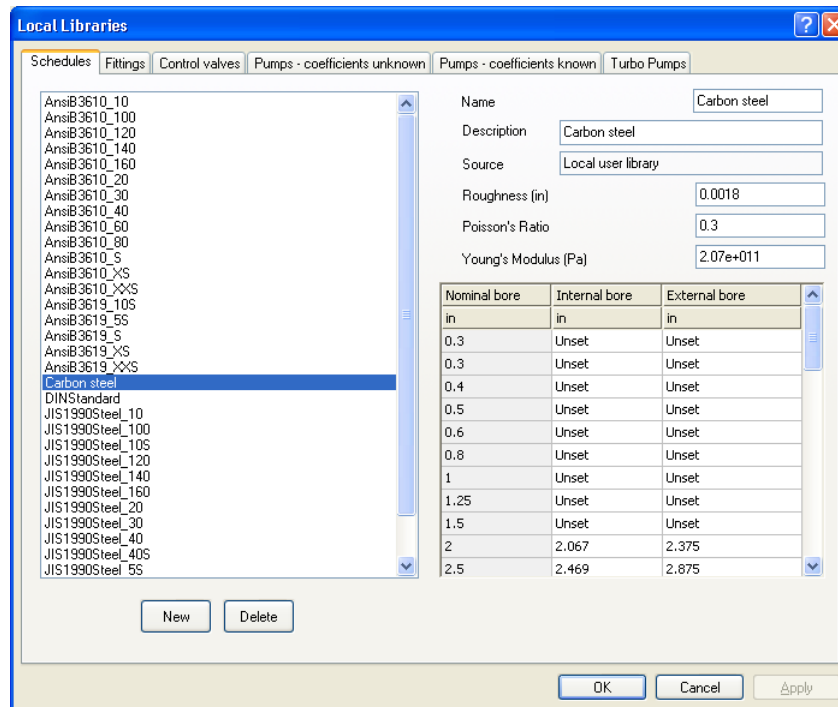
Pumps - coefficients known

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

Turbo pumps

Define or edit turbo pumps in the local user library..

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.
- [Transient Options](#) - Various modelling options.
- [Units](#) - Units to be used.
- [Fluid](#) - Fluid properties.
- [Pipe Types](#) - Define or edit pipe types.
- [Display options](#) - Schematic display options.

3.8.6 Colouration Menu

This menu contains three items which can be used to setup the colouring of nodes and/or pipes and ducts 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 and ducts based on the values of more than one attribute or result.

3.8.7 Calculation Menu

Options...

Displays the set of tabbed dialogs for setting the calculation options.

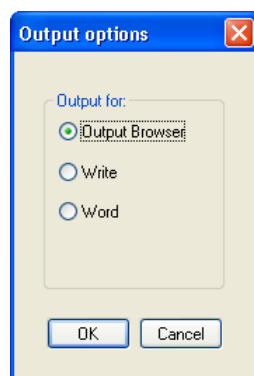
Go...

Displays the dialog for setting the calculation output options prior to performing a calculation.

3.8.8 Output Menu

Report...

This will prompt for an output viewer, by default the PIPENET supplied viewer:



Select the desired viewer from the options provided.

Graphs...

This will start the graph view for displaying the graphical output produced by the calculator.

3.8.9 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](#).

3.8.10 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 of 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**.

3.8.11 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

Transient calculator - version 6.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:

The screenshot shows a 'Security Key Status' dialog box with the following details:

- Number: 10025
- Customer: Sunrise - Robert
- Key type: Old style regular key
- Issued: 2005-8-3
- Standard License: Licenced , Expires: 2006-8-3, Runs: Unlimited
- Spray/Sprinkler License: Licenced , Expires: 2006-8-3, Runs: Unlimited
- Transient License: Licenced , Expires: 2006-8-3, Runs: Unlimited
- Note: Note all dates are in ISO format YYYY-MM-DD.
- Buttons: 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.

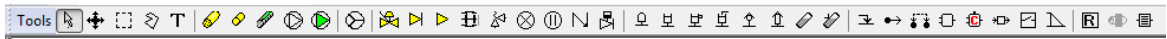
Sunrise on the Internet

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 Transient 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 a number of tools for creating the different types of link components (i.e. components with an input and output node) available with the Transient module:

- Pipe
- Short pipe
- Compressible pipe - only one allowed in network
- Pump
- Turbo pump
- Inertial pump
- Operating valve
- Non-return valve
- Check valve
- Fluid-damped check valve
- Liquid surge relief valve
- Regulator valve
- Bursting valve
- Inertial check valve
- Pressure surge relief valve
- Accumulator
- Surge tank
- Receiving vessel
- Simple tank
- Vacuum breaker
- Caisson
- Two-node caisson
- Two-node caisson - type 2

Then there is a group of seven control item:

- Flow sensor
- Pressure sensor
- Pressure Difference sensor
- PID controller
- Cascade PID controller
- Transfer function
- Switch

Finally, Three buttons:

- General Pressure Loss
- Equipment Item.
- Pipe bundle

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.
- Calculated flow direction - unused in Transient

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

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](#).

Calculate

Runs a calculation.

Browse

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

Graphs

Opens the graph viewer for viewing results.

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.

3.9.5 Find toolbar

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



The toolbar consists of:

- Find button
- 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
- edit box for entering the label of the component to find
- 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.

Tutorial Example 1 - One Pipe System

Part



4

4 Tutorial Example 1 - One Pipe System

4.1 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.

4.2 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 0-9999. Tagged labels consist of a 'tag' (a string of up to 8 characters) followed by a slash (/) and a number in the range 0-9999. 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. 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.
2. Tags are case insensitive, that is, RING is taken to be the same as Ring and ring.

4.3 Problem description

In the first example, we consider a pipe of diameter 25 cm and length 1000 m leading from the bottom of a reservoir of average depth 500 m.

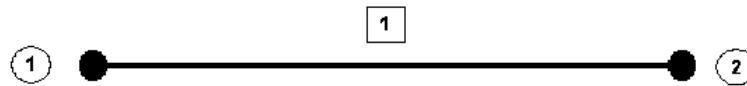


The waves on the surface of the reservoir cause a fluctuation in pressure at the entrance to the pipe of the form:

$$P = 50 + 0.01 \sin (2t/10)$$

where P is measured in bar G, and t in seconds. The aim of the calculation is to observe the resulting pressure fluctuations inside the pipe.

The first stage of any simulation is to represent the network in schematic form for input to PIPENET. The sketch below shows such a schematic representation of the problem:



In this case, the problem is represented by a single pipe, labeled 1, having an input node 1 and an output node 2. This example is supplied in the EXAMPLES directory as ex1.dat.

Setting up the problem

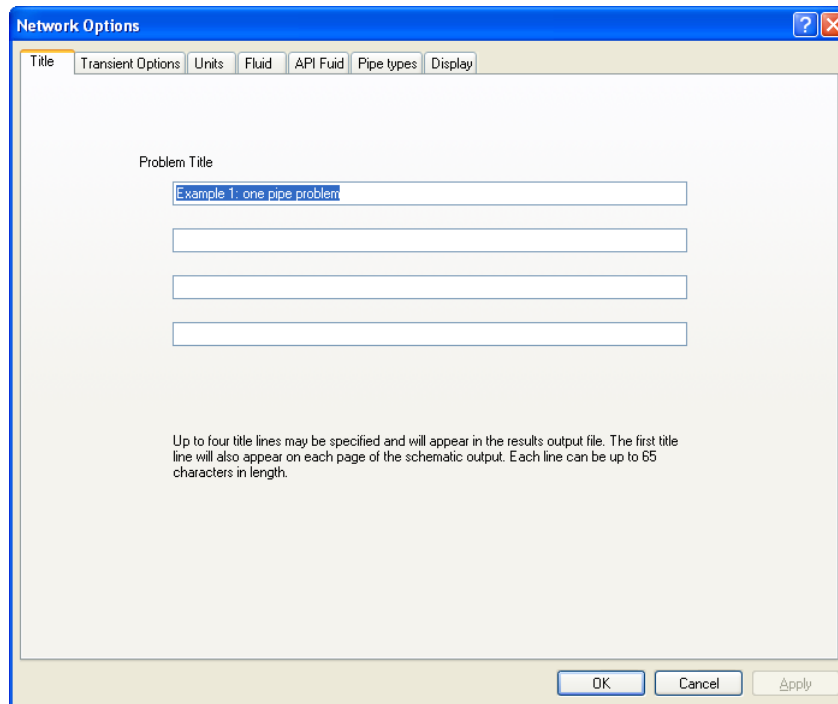
On running the program, select the [Standard option](#). The Transient Module User Interface is now available for data entry.

In this section, we take the User through the setup stage by stage.

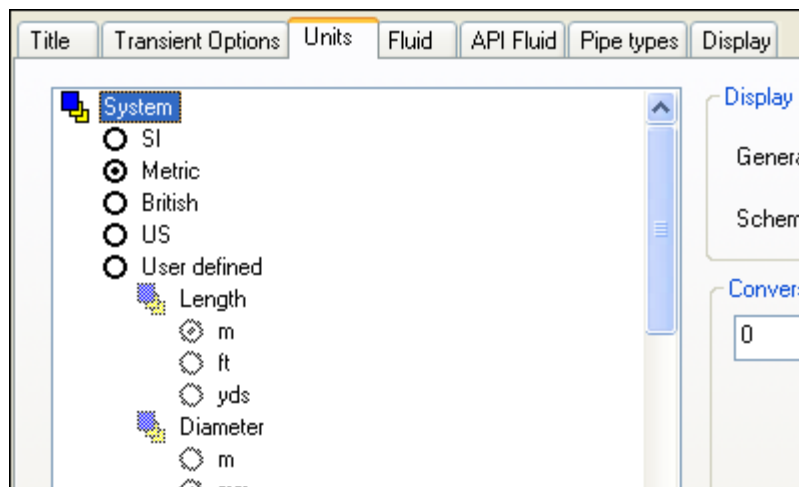
4.4 Stage 1 - Problem initialisation

No library files are required for this problem. Because no pipe material details are available we will have to set a default wavespeed.

1. Select **Options | Title** to enter the title:

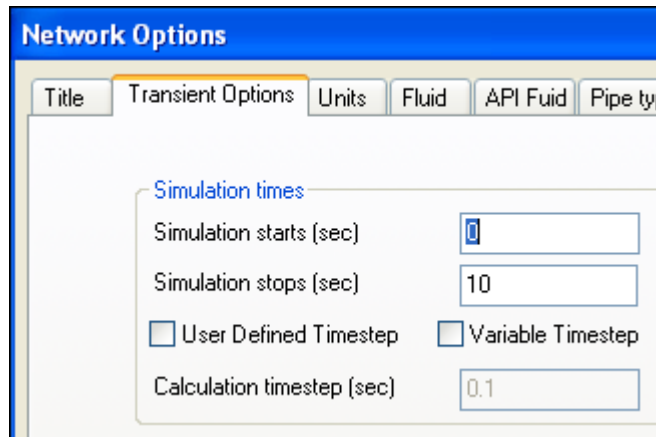


2. Select the **Units** tab to define the unit system. Units can be user-defined or set to SI, British or Metric or user-defined. In this example, **Metric** units will be used:

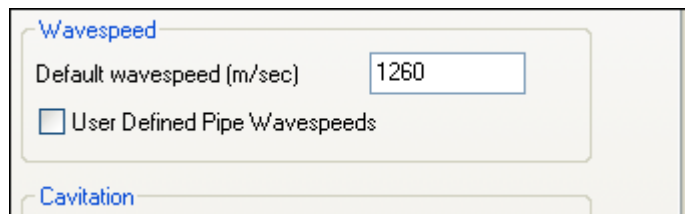


Information on the other features available in this dialog are discussed in [Specifying options - Units](#).

3. Select the **Transient Options** tab to define the simulation start and stop times. In this example we select start and stop times of 0 and 20 seconds, respectively:

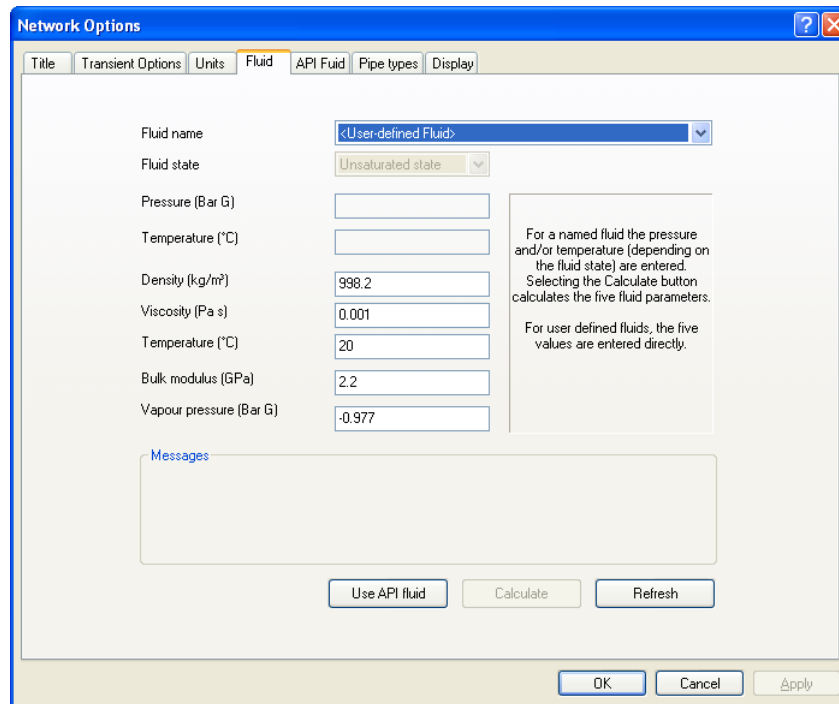


4. On the same dialog define a default wavespeed for the pipe in the system. A default wavespeed is required in this example, because pipe and fluid details that allow Transient to automatically calculate a wavespeed are not available:



If the User-defined Wavespeeds box is checked each pipe in the system may be given a different wavespeed..

5. Select the **Fluid** tab to define the fluid properties. In this example the fluid will be water at 20°C. Fluid temperature, bulk modulus and vapour pressure are not used in this example.





4.5 Stage 2 - Entering The Network

Creating the single pipe

With the schematic and the editable [Properties Window](#) this is a simple task. You simply draw the pipe in the schematic window and then define the properties of the pipe.

To draw the single pipe 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. A line representing the link element is drawn and tracks mouse movements.
4. Place the cursor at the point where you want the output node to appear and left click.
5. If, between defining the input node and the output node, you want to abort creation of the link, select the Escape key.
6. 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 show in the schematic representation above.

Now we can add the properties to the pipe, for example the bore and length of each pipe. Click on the pipe and its current properties appear in the [Properties Window](#).

Label	1	
Input node	1	
Output node	2	
Diameter	Unset	m
Length	Unset	m
Net height change	0	m
Roughness	Unset	m
Additional k-factor	0	
Type	No pipe type	
Wavespeed	1260	m/sec
Results selected?	NO	
Section Type	Constant Gr...	
Results		
Min inlet pressure	n/a	
Time	n/a	
Max inlet pressure	n/a	
Time	n/a	

In this example the Young's modulus and Poisson's ratio for the pipe material are unavailable and the default wavespeed is displayed. If some pipe schedules had been defined they would appear in, and could be selected from, the Pipe Type list. Multiple fittings can be inserted on the pipe if required. An additional k-factor for the pipe can also be entered in Additional k-factor.

Click in the value field for the pipe diameter and enter the value 250 followed by Tab to move to the next field; the length. Enter a value of 1000.0 for the length of the pipe followed by either Tab or Return to accept the value.

Label	1	
Input node	1	
Output node	2	
Diameter	250	mm
Length	1000	m
Net height change	0	m
Roughness	10	mm
Additional k-factor	0	
Type	No pipe type	
Wavespeed	1260	m/sec
Results selected?	YES	
Section Type	Constant Gr...	
Results		
Min inlet pressure	n/a	
Time	n/a	
Max inlet pressure	n/a	
Time	n/a	

Undefined and 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.

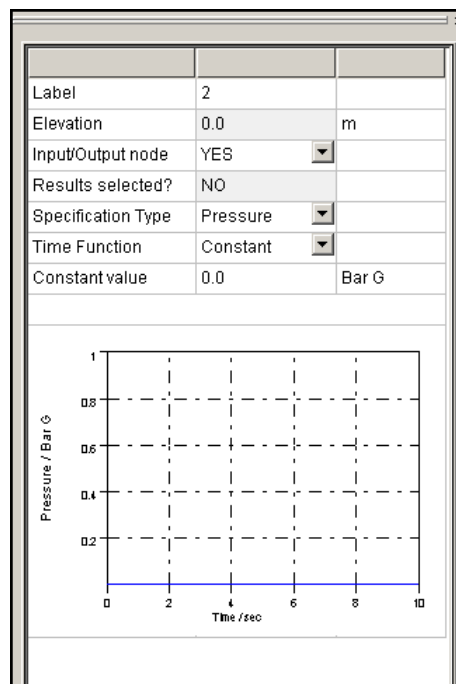
Entering the Specifications

Two specifications are required in this example. Pressure at the downstream end of the pipe is constant, while the pressure at the upstream end varies sinusoidally.

Select the node which is to have the constant specification, and in the Properties window:

1. Select the Specification type as **Pressure**.
2. Tab to the next field and select the time function as **Constant**.
3. Tab to the next field and enter the constant value as 0.0.
4. Enter tab or the Return key to complete editing.

The Properties window should appear as:

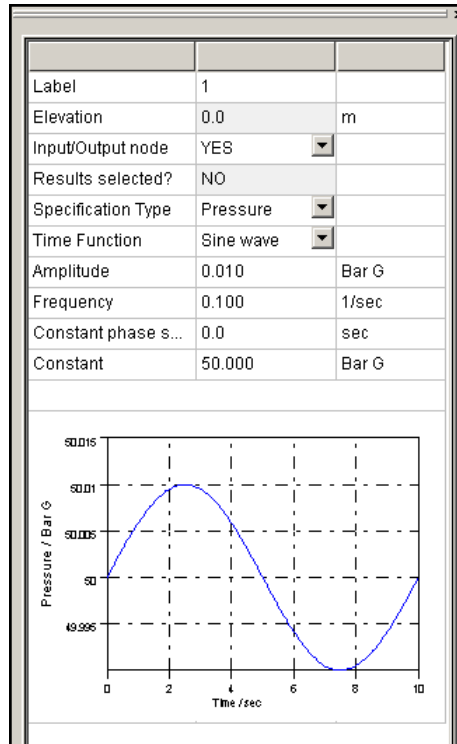


Select the other node and in a similar manner in the Properties window:

1. Select the specification type as Pressure.
2. Tab to the next field and select the time function as Sine wave.
3. Tab to the next field and enter an amplitude of 0.010.

4. Tab and enter the frequency as 0.1.
5. Tab and enter the constant phase as 0.0.
6. Tab and enter the constant value as 50.
7. Tab or select Enter to complete data entry.

The Properties window should appear as:



Another way to look at pipes is via the [Tabular View](#), although it is not a significant advantage with only one pipe, but makes a lot of sense when there are many pipes whose properties you wish to view. Select the menu option **View | Data** window to display the window:

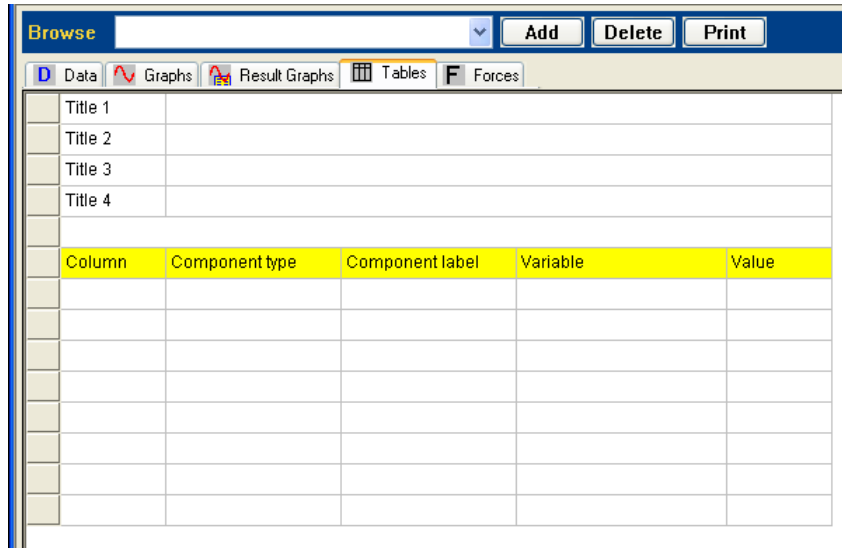
Label	Input node	Output node	Diameter	Length	Net height change	Roughness	Additional k-factor	Type	Wavespeed	Results selected?	Section Type
			mm	m	m	mm			m/sec		
1	1	2	250	1000	0	10	0	No pipe type	1260	YES	Constant Gradient

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. Moving to the next field is achieved by either entering a tab or by pressing the Return key.

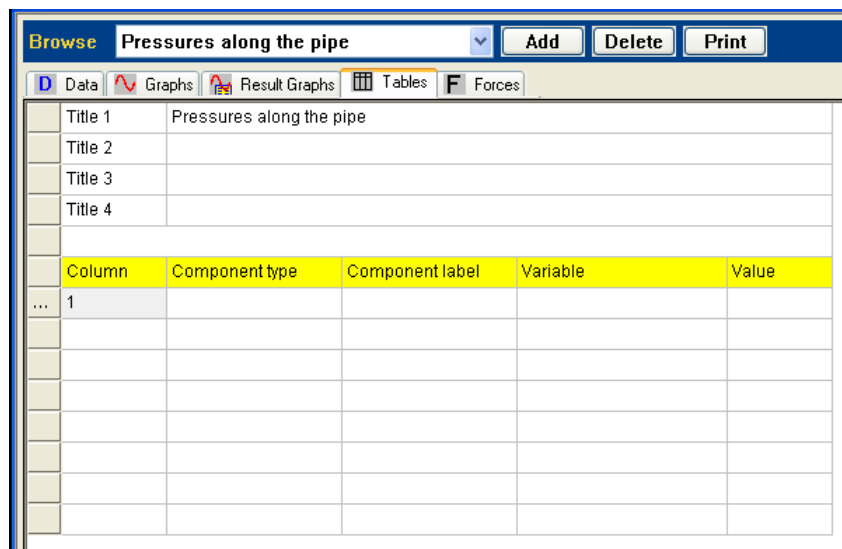
Both the Schematic Window and the Tabular View can be displayed simultaneously by selecting the menu option **Windows | Tile horizontally** or **Windows | Tile vertically**.

Defining the output tables

Simple or Complex tables can be defined. Simple tables contain output results for a single component only. Complex tables can include output results for more than one component. In this example we will request output results for pressures and flow rates along the pipe by using two simple tables. Tables are edited via the [Tables tab](#) in the [Tabular view](#); selecting the tab displays:



Select the Add button in the heading of the window and enter the title as "Pressures along the pipe" in the Title 1 field (the name in the title 1 field appears in the combo box in the window heading) and the window should appear as:



Now select the symbol ... in the first column, adjacent to the column number 1; this expands the information stored starting in column 1, select the component type as Pipe and the window changes to:

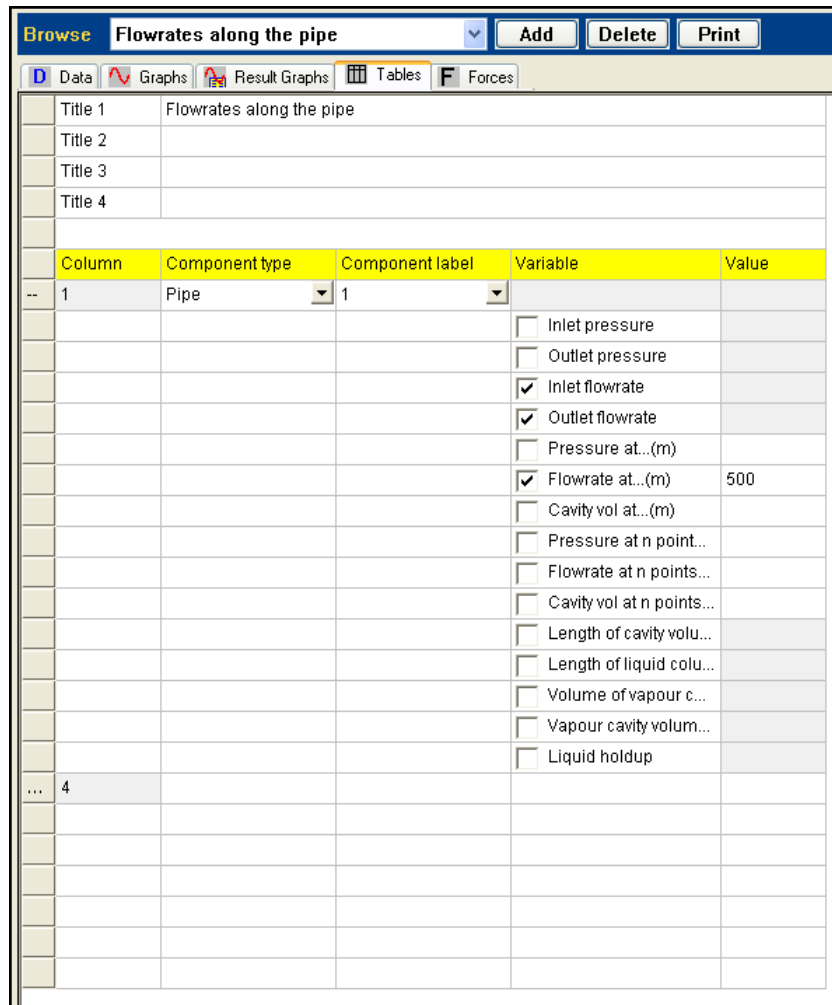
Browse Pressures along the pipe				
Column	Component type	Component label	Variable	Value
1	Pipe	1	<input checked="" type="checkbox"/> Inlet pressure	
			<input checked="" type="checkbox"/> Outlet pressure	
			<input type="checkbox"/> Inlet flowrate	
			<input type="checkbox"/> Outlet flowrate	
			<input checked="" type="checkbox"/> Pressure at...(m)	500
			<input type="checkbox"/> Flowrate at...(m)	
			<input type="checkbox"/> Cavity vol at...(m)	
			<input type="checkbox"/> Pressure at n point...	
			<input type="checkbox"/> Flowrate at n points...	
			<input type="checkbox"/> Cavity vol at n points...	
			<input type="checkbox"/> Length of cavity volu...	
			<input type="checkbox"/> Length of liquid colu...	
			<input type="checkbox"/> Volume of vapour c...	
			<input type="checkbox"/> Vapour cavity volum...	
			<input type="checkbox"/> Liquid holdup	
...	4			

Note that the next available column is now shown as 4.

Creating another table for the flow rate is archived in a similar manner:

1. Select the Add button to create a new table.
2. Enter the title for the table in the Title 1 field.
3. Select the component type as Pipe again.
4. Select the options inlet flow rate, outlet flow rate and Flow rate at ...(m).
5. Provide the value for the flow rate at 500 m.

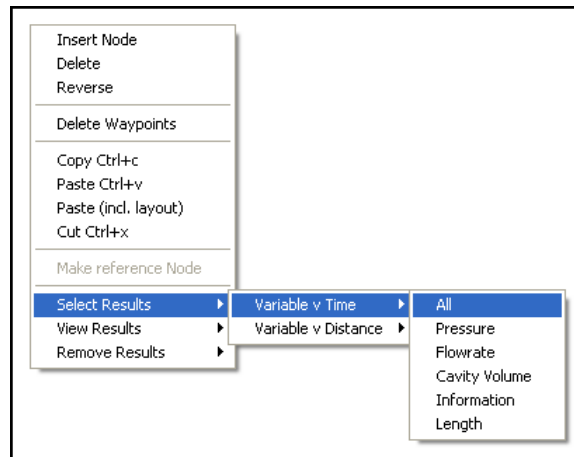
The definition should appear thus:



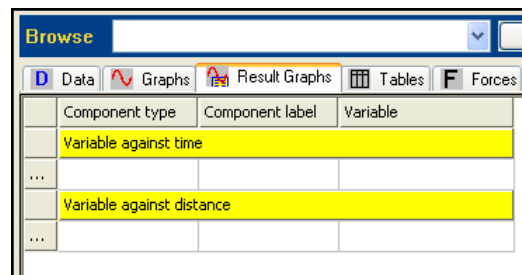
Defining the graphs

Two types of Output Graph are available: variable against time, or pipe variable against distance. In this example we will request graphical output of pipe pressures and flow rates against time:

Graphs can be associated with a component in two ways. In the first we right-click on the desired component. Select the option **Select Results** with sub option **Variable v Time** and then **All** variables:

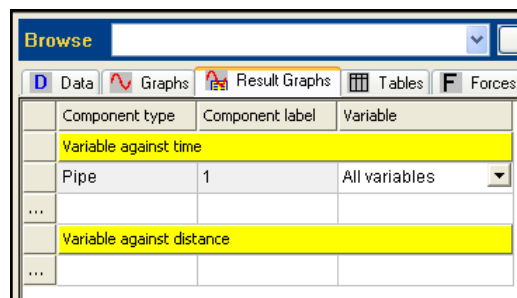


This is probably the simplest way since we only have one pipe. The alternative is via the [Graphs tab](#) in the [Tabular View](#):




which can also be used for displaying all defined graphs. Select the ... in the first column of the first available line under the Variable against time section, followed by selection of the component type as Pipe, the component label as 1 and the variable All, all being selectable from drop-down lists of available items.

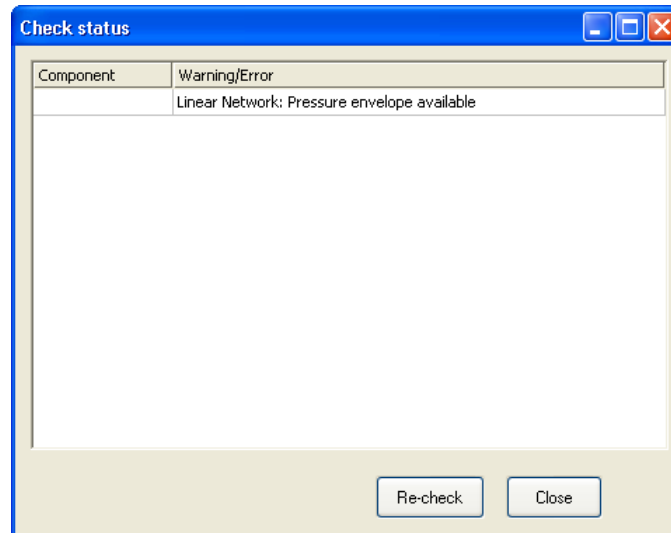
Regardless of the method used the Graphs tabbed window should appear as the following:



The start and stop times and output time step for graphical output are defined in the Calculate..Output dialog (see Stage 4) where a filename for graphical output can also be defined.

4.6 Stage 3 - Checking the network

Check various aspects of the entered network (such as connectivity and size) by selecting the **check** button on the **calculation** toolbar . If the network is complete and well connected a dialog similar to the following will be shown:



If there are any errors or warnings they will be displayed in the dialog. The User should always perform a check on the network before proceeding with a calculation.

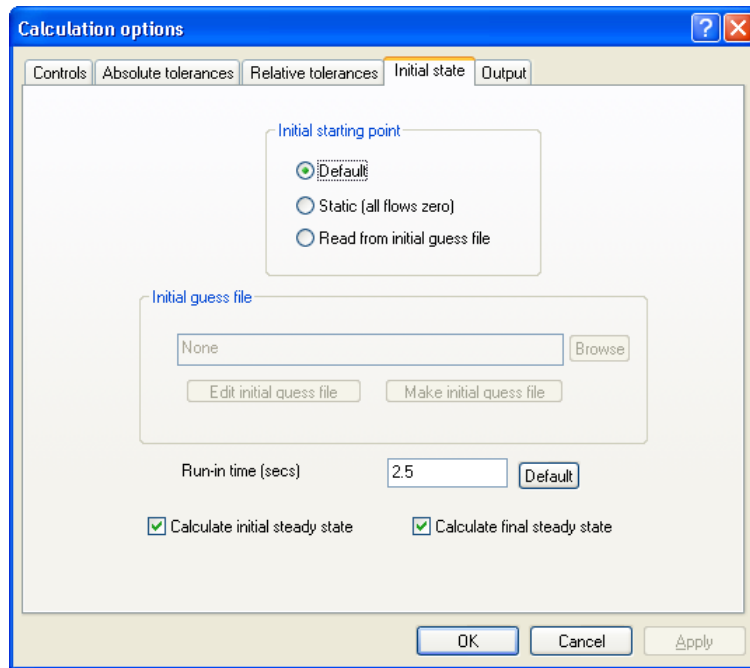
4.7 Stage 4 - Saving the network

Once the problem data has been entered, it should be saved to a PIPENET data file with a .SDF extension. Select the **File | Save As** option and enter a filename. The directory to which the file is saved can also be specified (by default the EXAMPLES subdirectory of the directory to which PIPENET was installed). A different filename should be used for each different scenario that is run.

4.8 Stage 5 - Performing a calculation

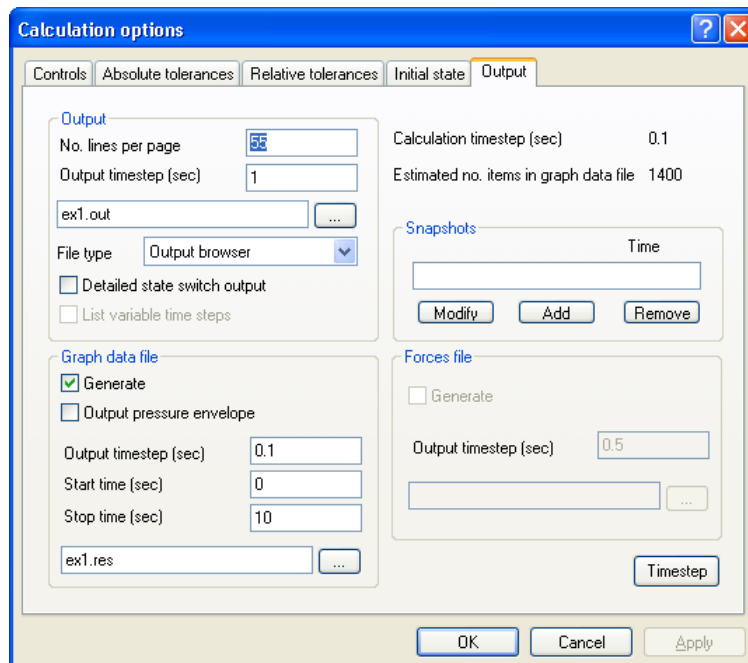
1. Specify the initial state for the system by selecting the Calculation Options menu and then selecting the [Initial State tab](#).

In this example we will use a short transient run-in (to stabilize the solution), before calculating the initial steady state for the calculation:



Pressing the default button sets the run-in time to a quarter of the total simulation time. Other options in the dialog allow the User to specify a [Static initial state](#) (with all flows zero) or to use an initial guess file as the starting point for the simulation.

- Specify the output filenames and time steps by selecting the Output tab:

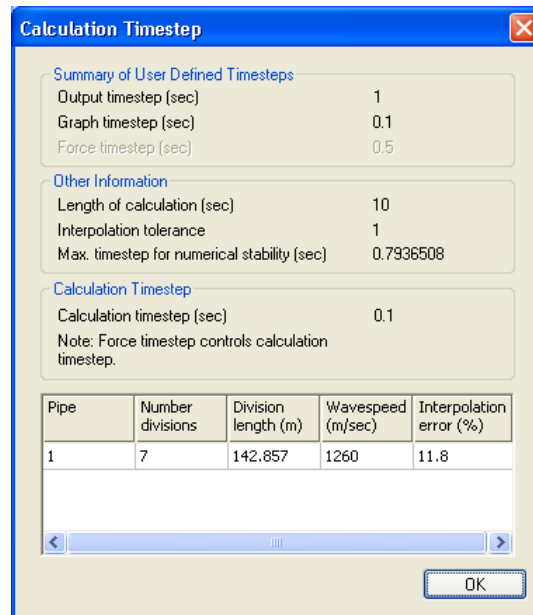


No forces are calculated in this example so the generate forces file option is unchecked. Example 3 demonstrates the calculation of forces in Transient

Module.

A restricted time window for graphical output can be specified by changing the graph start and stop times (by default the simulation start and stop times are 0.0 and 10.0 seconds respectively). This can be useful in reducing the size of the graph file by concentrating on an interesting part of the simulation.

Pressing the **Timestep** button gives information about the simulation time steps and the division of pipes:

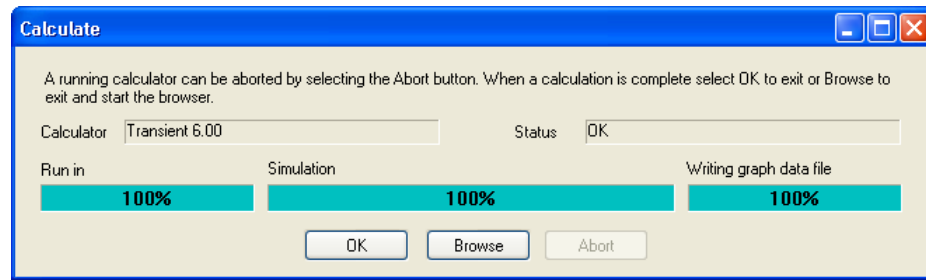



The maximum time step for numerical stability is dependent on the pipe lengths and wave speeds. All pipes in the network are listed. The number of divisions in each pipe (based on the calculated time step) is shown along with the interpolation error based on these divisions.

The interpolation error tolerance is changed in the **Controls** tab. In this case the default tolerance of 1 is used which means PIPENET chooses the simplest way of dividing the network.

3. Run the simulation by selecting **Calculation | Go**.

The **Calculation...Output** dialog appears and any final changes to the output details can be made. Selecting **OK** runs the calculation. The Transient Module progress bar illustrates the progress of the simulation through each of its three stages - run-in, simulation and writing of the graph data file:



The calculation uses a temporary copy of the network information so it is possible to perform other tasks while waiting for the calculation to be completed. As an alternative the calculation can be started using the calculate button on the toolbar .

4.9 Stage 6 - Viewing the results

Results may be viewed in a number of different ways:-

- Summary results can be viewed in the Data window
- The report viewer, or browser, can be used to display the calculator output
- Output graphs can be viewed using the Graph Viewer

Viewing summary results in the data window

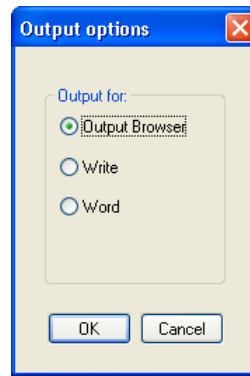
When a calculation has been performed summary pressure extrema for each component are displayed in the Properties windows and in the Results page in the [Tabular View](#). To display the tabular view, if it is not already displayed, select the menu option **View | Data window** and then the **Results** tab:-

Label	Input node	Output node	Min inlet pressure	Time	Max inlet pressure	Time	Min internal pressure	Time	Max internal pressure	Time	Min outlet pressure	Time	Max outlet pressure	Time
			Bar G	sec	Bar G	sec	Bar G	sec	Bar G	sec	Bar G	sec	Bar G	sec
1	1	1	2	49.9899899	7.5	50.0099899	2.5	0	0	50.0099899	2.5	0	0	0

Output report

Each Transient Module calculation produces an **output report**. This contains details of the calculation including a summary of the network and specifications, any output tables that have been requested and the minimum and maximum pressures observed during the simulation. The report file is viewed by selecting **Output | Report**.

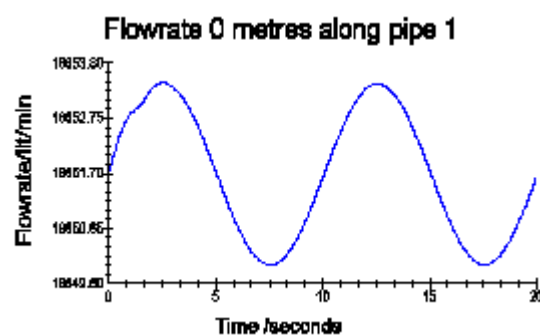
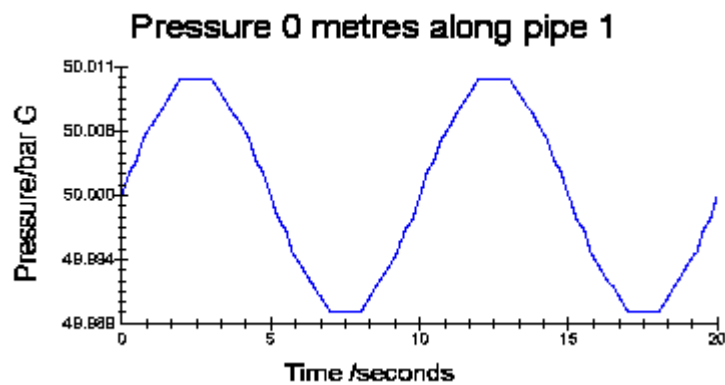
The report viewer will be the PIPENET Output Browser, Write or Word depending on the option specified in the calculation output dialog:

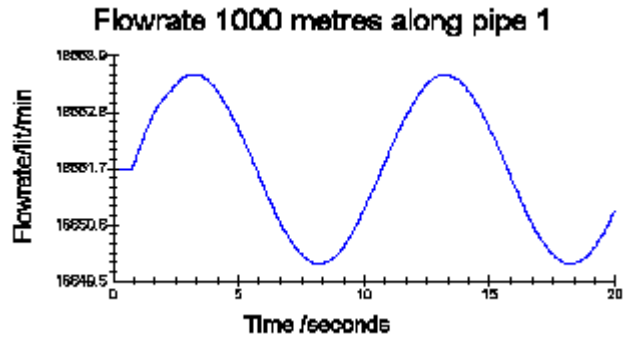


If there are errors at any stage in the calculation they will be listed in the output report.

Output graphs

The graphical results can be drawn using the [Graph Viewer](#) which is accessed by selecting **Output | Graphs...** For more details on using the Graph Viewer to display and edit graphs see the [Graph Viewer chapter](#). In our example the results show that a sinusoidally varying pressure and flow rate at various points along the pipe can be observed.





Note that the flow rate graphs clearly show the time delay for the pressure variation at the upstream end of the pipe to affect flow results 1000 metres away at the downstream end of the pipe.

Tutorial Example 2 - Small Sprinkler System

Part

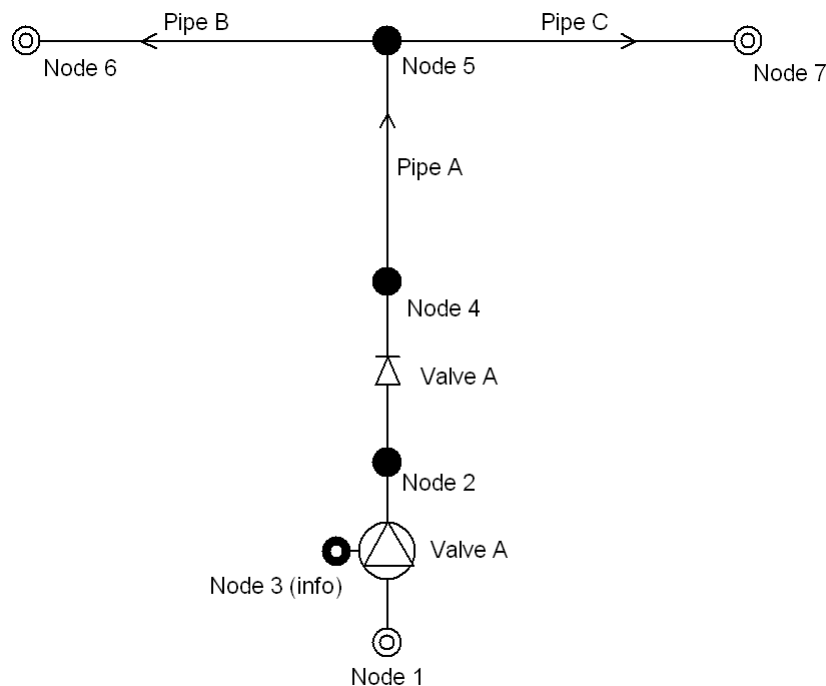
5

5 Tutorial Example 2 - Small Sprinkler System

5.1 Problem description

This second example illustrates a use of the [Spray option](#). It also describes how to create and use library files to hold project-specific information relating to pipes, valves or pumps.

The system consists of a pump which supplies two sprinkler outlets. The pump is connected to the outlets by one vertical length of pipe and two horizontal sections which branch off it. A diagram of the system, including node and component labels, is shown below:



Under normal conditions, each outlet (node 6 and 7) makes a steady demand of 25 litres/min. The aim of the simulation is to observe the effect of sudden closure of one of the outlets, in particular the size of the pressure surge on all intermediate pipework.

This example is supplied in the EXAMPLES directory as ex2.dat.

Setting up the problem

On running the program select the Spray option. In this example, we will show how to use library files to hold project-specific information. Libraries of pumps, valves, pipe schedules or user-defined fittings can be defined and saved independently of the network data. The library files can then be loaded in Transient Module prior to defining the network details.

In setting up the problem, we will first show how to define, save and retrieve a pump library. Then, we shall use the pre-defined pump characteristic curve for the pump in the

sprinkler system.

5.2 Stage 1 - Adding a pump to the library

Specifying units

Specify the units in which the pump library is to be defined by selecting **Options | Units**. In this example the pump performance curve data is in Metric units.

Specifying the pump

Select the tab **Libraries | Pump Coefficients Known**, select **New** and enter the following data:

1. The name of the pump as Test 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).
2. A minimum flow of 20 litres/minute.
3. A maximum flow of 90 litres/minute.
4. Constant coefficient of 16.51.
5. Linear coefficient of 0.009285.
6. Quadratic coefficient of -0.001 (note the negative sign).

Finally select **Apply** to have the pump curve calculated and displayed:

The screenshot shows the 'Local Libraries' dialog box with the 'Pumps - coefficients known' tab selected. The 'Name' field is 'Test pump'. The 'Curve type' is 'Quadratic'. The 'Ref. Fluid Density (kg/m³)' is 998.2343. The 'Flowrate' is set to 'lit/min' and 'Head' is 'bar'. The 'Working range' shows a minimum flow of 19.998 and a maximum of 90. The 'Curve coefficients (reference fluid)' are: Constant 16.51, Linear 0.009285, Quadratic -0.001, and Cubic 0. A table on the right shows the calculated head values for various flow rates, and a graph below it plots Pressure (Bar fluid (reference)) against Flow Rate (l/min).

Flow rate (l/min)	Head (input) Bar fluid (reference)	Head (curve) Bar fluid (reference)	Head (calculation) Bar fluid (current)
20		16.296	16.295
23.5		16.176	16.175
27		16.032	16.031
30.5		15.863	15.862
34		15.67	15.669
37.5		15.452	15.452
41		15.21	15.209
44.5		14.943	14.943
48		14.652	14.651
51.5		14.337	14.336
55		14.0	14.0
58.5		13.641	13.64
62		13.258	13.257
65.5		12.851	12.85
69		12.42	12.419
72.5		11.965	11.964
76		11.486	11.485
79.5		10.983	10.982
83		10.456	10.455
86.5		9.905	9.904
90		9.33	9.329

Specifying the pump - alternative method with coefficients unknown

If the pump coefficients are unknown a similar pump could be created using the **Libraries | Pump Coefficients Unknown**, with the pump curve data is as follows:

Flow rate (litres/minute)	Pressure (bar G)
20	15.4
30	14.7
40	14.2
50	13.5
60	12.6
70	11.2
80	9.8

To specify the pump details in this way proceed as follows:

1. Select the **New** button.
2. Provide the name for the pump as Test 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 20 litres/minute and a maximum flow rate of 90 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 20.0.
6. tab to the next field and enter the corresponding pressure of 15.4.
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.

5.3 Stage 2 - Problem initialisation

Title

Select **Options | Title** to define the title as 'Example 2 : Small Sprinkler System'.

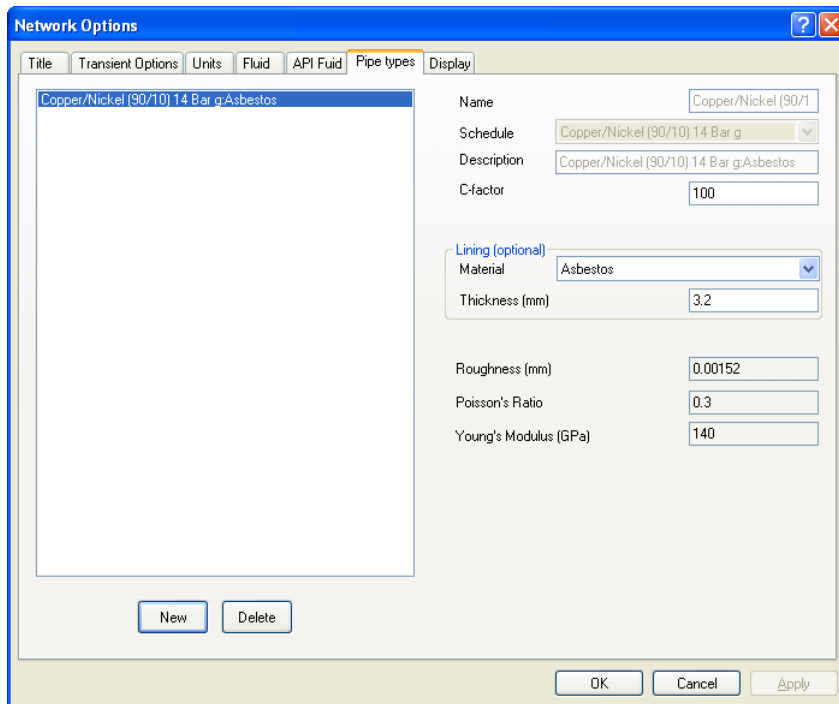
Units

The Metric Units system has already been selected before defining the pump library.

Pipe type

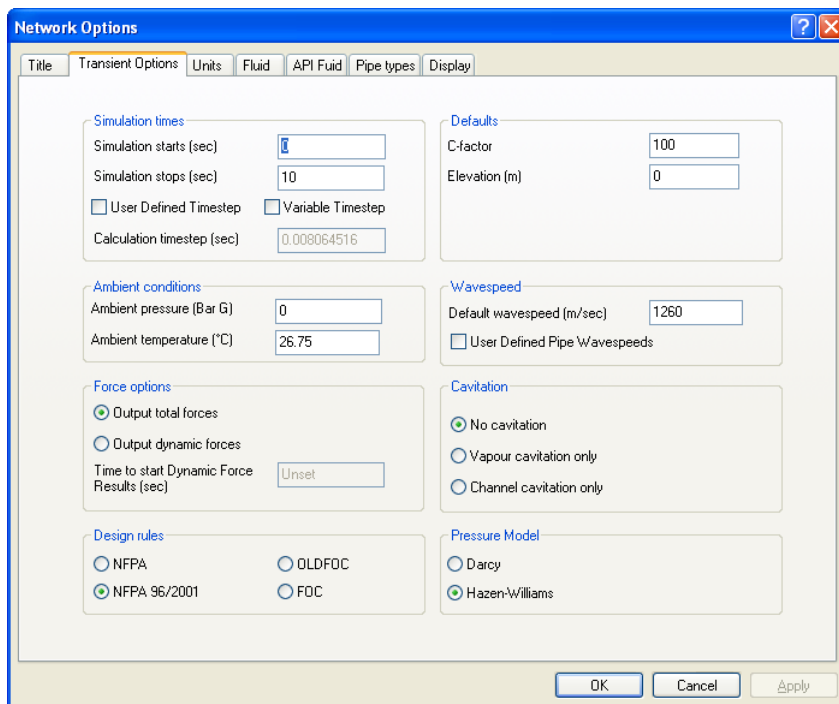
We are going to use a built-in pipe schedule so all we need to do is to define the pipe type.

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. Select an Asbestos lining with a thickness of 3.2 mm.
5. The dialog should appear as follows:



Transient module options

Select **Options | Transient options** and enter the simulation start and stop times as 0 seconds and 10 seconds respectively. Also in this dialog select the use of NFPA design rules and the Hazen-Williams equation with no vapour cavitation. Note that if the pressure drop method is changed to Coulson and Richardson at a later stage then all pipe types should be redefined, replacing C-factors with values for pipe roughness.



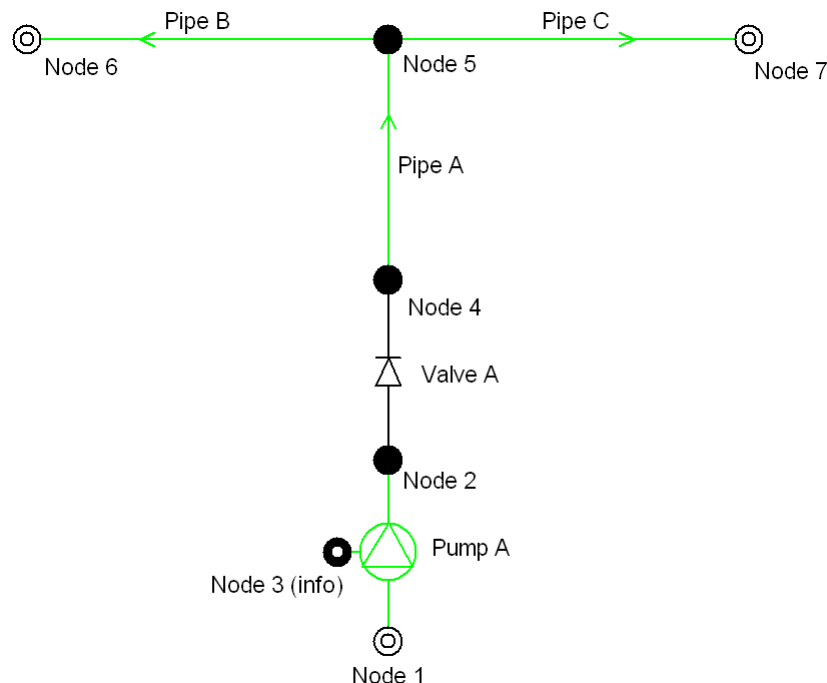
Defaults

Select **Options | defaults** to set the default values for pipes. It is useful to set default pipe values before entering large networks to reduce the amount of data that has to be entered. In the **Spray** option default values for C-factor and pipe elevation can be set. In the **Standard** option C-factor is replaced by roughness. In addition default values for k-factor and pipe diameter can be specified.

5.4 Stage 3 - Entering the network details

Components

Draw the network, such that the layout appearance is similar to the following:



Thus there is one pump, a non-return valve and three pipes. Ignore the green colour of some components and the appearance of nodes 1, 6 and 7. These are to do with graphical results and specifications respectively. For a small network like this it is probably best to layout all of the components and then enter the attributes for each component in turn using the [Properties window](#) or the [Tabular grid](#).

For the non-return valve no attributes are required, for the pump, the one and only pump from the library will be selected, and for pipes you need to enter the following data:

Pipe label	Input node	Output node	Diameter	Length	Elevation
A	4	5	250	100	100
B	5	6	100	10	0
C	5	7	100	10	0

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.

If the data was entered via the Properties window then the vales for all pipes can be viewed via the tabular grid - select **View | Data window** and you should see something similar to the following:

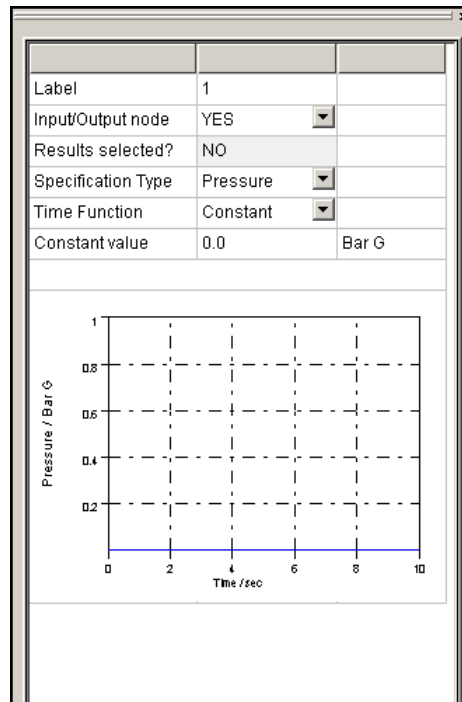
Label	Input node	Output node	Diameter mm	Length m	Net height change m	C-factor	Type	Wavespeed m/sec	Results selected?	Section Type
1	A	4	250	100	100	100	Copper/Nickel (90/10) 14 Bar g:Asbestos	1122.39	YES	Constant Gradient
2	B	5	100	10	0	100	Copper/Nickel (90/10) 14 Bar g:Asbestos	1239.38	YES	Constant Gradient
3	C	5	100	10	0	100	Copper/Nickel (90/10) 14 Bar g:Asbestos	1239.38	YES	Constant Gradient

Specifications

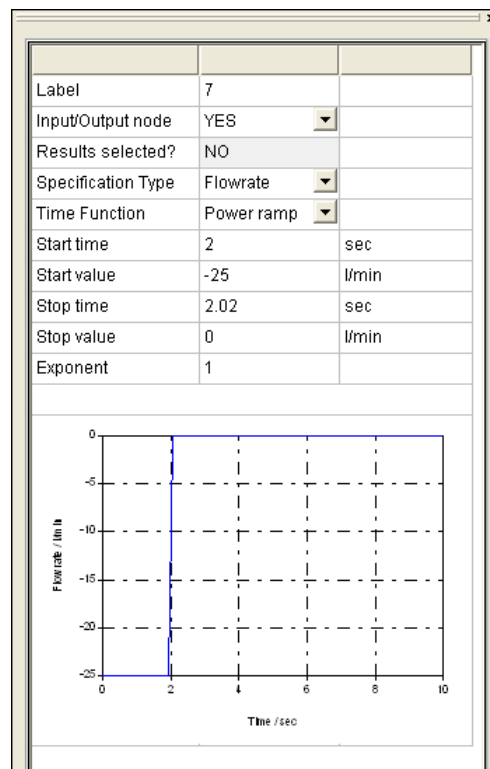
There are four specifications in the system - three I/O specifications at nodes 1, 6 and 7 and one information specification (for the pump) at node 3. The details of the three constant specifications are given in the table below, note that a negative flow rate indicates flow out of the system.

Spec. label	Spec. type	Node	Time function	Constant value
1	Pressure	1	Constant	0
2	Flow rate	6	Constant	-25
3	Information	3	Constant	1

For each of the nodes 1,3 and 6 the information can be entered in the Properties window, for node 1 the Properties window appears thus:



The constant value of one for information specification 3 indicates that the pump operates at full speed throughout. We wish to simulate sudden closure of the outlet at node 7. We do this by defining a Power Ramp time function, with the flow rate value at the node decreasing from -25 to 0 lit/min between 2 and 2.02 seconds. Again this can all be done in the Properties window (the exponent of 1 indicates a linear closure):



5.5 Stage 4 - Defining output tables and graphs

Output tables

Tables are defined using the **Tables** tab in the Tabular grid. When first selected the display might typically be as follows:

Column	Component type	Component label	Variable	Value
+	1	Pipe	1	
...	5			

The table is divided into two parts; at the top are four title lines and below this the definition of the table. The drop-down box at the top indicates which table is currently displayed, in this case "Pump operation".

The table definition as initially displayed displays a line for each component, identified by its type and label together with a start column. the last row (with the symbol ... in the first column) indicates the next free column available (there are a maximum of eight columns allowed in a table). In the above example the pump with label uses the first five columns of the table. To view which attributes have been selected for the table select the + symbol in column one and the display expands to include a list of the variables:

Browse Pipe 1 - Reservoir to Bend					Add	Delete	Print	
D Data					Graphs	Result Graphs	Tables	F Forces
Title 1	Pipe 1 - Reservoir to Bend							
Title 2								
Title 3								
Title 4								
Column	Component type	Component label	Variable	Value				
-- 1	Pipe	1	<input checked="" type="checkbox"/> Inlet pressure <input checked="" type="checkbox"/> Outlet pressure <input checked="" type="checkbox"/> Inlet flowrate <input checked="" type="checkbox"/> Outlet flowrate <input type="checkbox"/> Pressure at...(m) <input type="checkbox"/> Flowrate at...(m) <input type="checkbox"/> Cavity vol at...(m) <input type="checkbox"/> Pressure at n point... <input type="checkbox"/> Flowrate at n points... <input type="checkbox"/> Cavity vol at n points... <input type="checkbox"/> Length of cavity volu... <input type="checkbox"/> Length of liquid colu... <input type="checkbox"/> Volume of vapour c... <input type="checkbox"/> Vapour cavity volum... <input type="checkbox"/> Liquid holdup					
... 5								

The + symbol now changes to a - symbol which if selected will once again hide the details of the attributes selected. De-selecting or selecting one or more of the available attributes will immediately update the number of columns used.

To add extra columns to the grid click on the last row (the one containing the ... symbol in column 1) and select from the list of available component types and component labels displayed. Finally select the attributes to be displayed:

Column	Component type	Component label	Variable	Value
-- 1	Pump	A	<input checked="" type="checkbox"/> Inlet pressure	
			<input checked="" type="checkbox"/> Outlet pressure	
			<input checked="" type="checkbox"/> Inlet flowrate	
			<input checked="" type="checkbox"/> Outlet flowrate	
			<input checked="" type="checkbox"/> Setting	
-- 6	Node	4	<input checked="" type="checkbox"/> Cavity volume	
... 7				

We need to define five tables; one for the pump, two tables for pipe A and one table for each of the pipes B and C. For the pump we select all variables as in the second image above. For pipe A, we define one table which contains the flow rates at 6 points along the pipe and one for the pressures at 6 points along the pipe. The need for two tables is because a table can have a maximum of 8 columns. The display for the pressure table appears as:

Browse Pressures and flowrates along pipe B				
Column	Component type	Component label	Variable	Value
--	1	Pipe	B	
			<input type="checkbox"/> Inlet pressure	
			<input type="checkbox"/> Outlet pressure	
			<input type="checkbox"/> Inlet flowrate	
			<input type="checkbox"/> Outlet flowrate	
			<input type="checkbox"/> Pressure at...(m)	
			<input type="checkbox"/> Flowrate at...(m)	
			<input type="checkbox"/> Cavity vol at...(m)	
			<input checked="" type="checkbox"/> Pressure at n point...	3
			<input checked="" type="checkbox"/> Flowrate at n points...	3
			<input type="checkbox"/> Cavity vol at n points...	
			<input type="checkbox"/> Length of cavity volu...	
			<input type="checkbox"/> Length of liquid colu...	
			<input type="checkbox"/> Volume of vapour c...	
			<input type="checkbox"/> Vapour cavity volum...	
			<input type="checkbox"/> Liquid holdup	
...	7			

Output Graphs

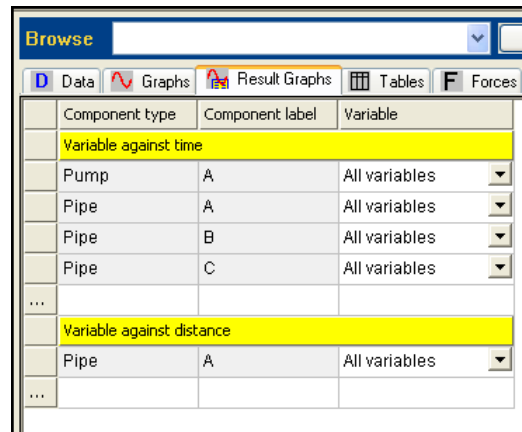
Components for which results (graphs) have been selected are displayed in green using the default colouring scheme. To request results for a particular component, right-click on the component and select the **Select Results** option. This is the same method that was used in previous versions of PIPENET Transient. For non-pipe components this will display a list of available variables - select the single result required, for example **Pressure**, or **All** to accumulate results for all attributes.

Removing results

To remove results for a component right-click on the component and select the **Remove Results** option.

Selecting and removing results in the Tabular Grid

A summary of which results have been selected can be viewed on the [tabular grid](#) - select the **Graphs** tab when all selected results are displayed, each result being in one of two categories: variable against time (all components) or variable against distance (pipes only):



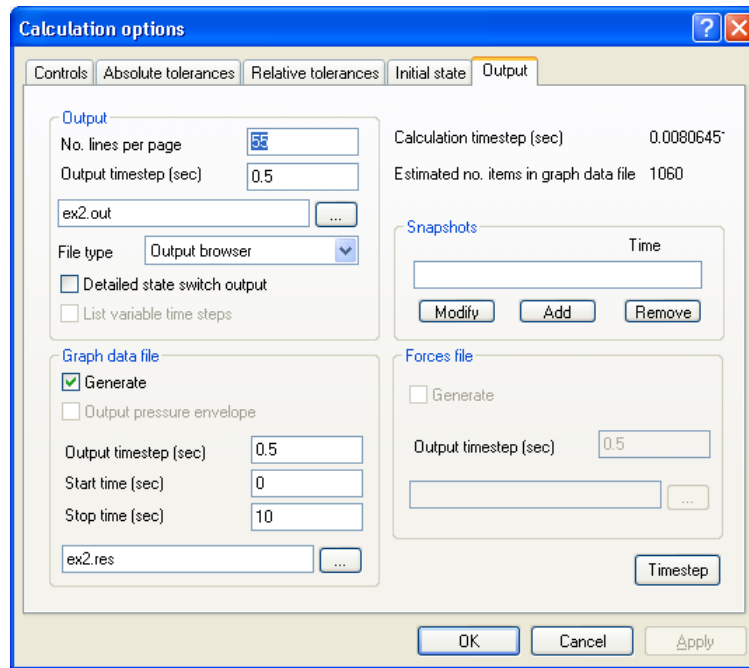
The first column provides the component type, the second the component label and the third the selected variable, only the last of these is directly editable. To add a new result in one of the two categories:

1. Select line containing ... in the first column the first blank line.
2. Select the component type from the drop-down list in the second column of the newly created row.
3. Select the component label in the third column.
4. Select the variable(s) required.
5. Finally, select the symbol in the first column to commit the change.

A result may be deleted by right-clicking in the first column of a row and selecting the **Delete graph** option.

5.6 Stage 5 - Performing a calculation

The default initial state is used here (a 2.5 second run-in followed by calculation of the initial steady state). Select **Calculation | Options** and the Output tab:



To select output filenames and time steps for the calculation. We generate a graph output file having time step 0.5 s. Note that the graph data file has a relatively small size of 1060 items.

Press **Timestep** to view pipe and time step details.

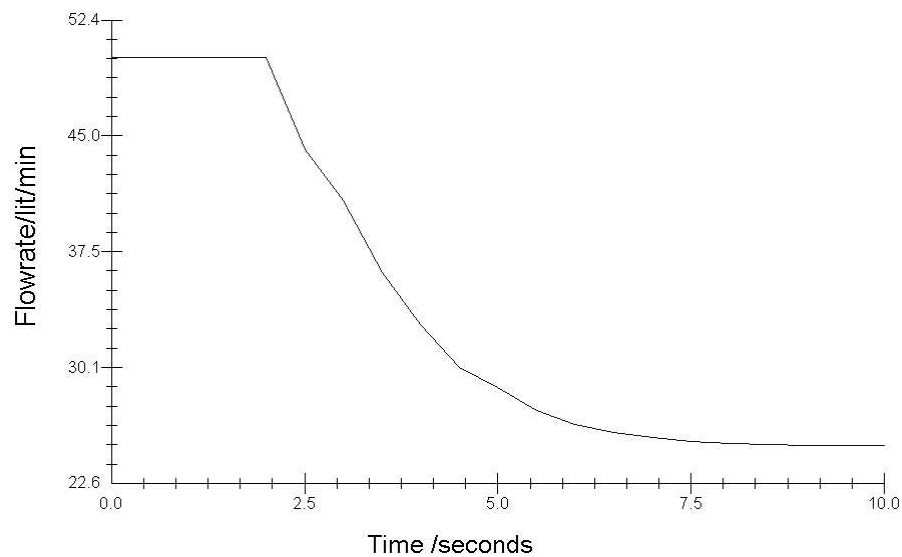
In this example the longest allowed calculation time step is short relative to the total simulation time. This is because the sprinkler pipes B and C (10 m. in length) are short relative to the longer pipe A (100 m. in length). The pipe lengths are crucial in determining a time step for the simulation. For a quick qualitative analysis of the problem we could replace pipes B and C with the Short Pipe model and this would increase the calculation time step to 0.072 s

Select **Calculation | Go...** and press **OK** to perform the simulation. The simulation is slower than the first example because of the shorter calculation time step.

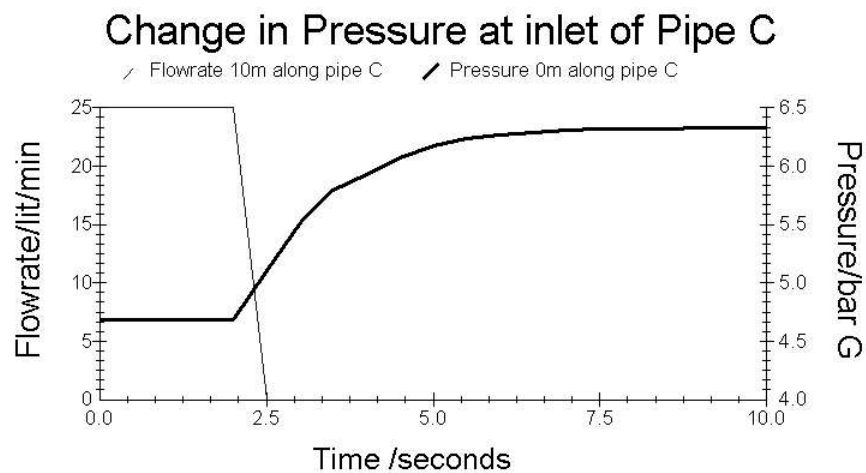
Viewing and Interpreting the Results

As in example 1 results can be viewed in tabular form using the Output Browser, or in graphical form using the Graph Viewer. The graphical results show a decrease in the pump outlet flow rate due to the decreased demand in the sprinkler system.

Outlet flowrate of pump A



The effect of closing the sprinkler at the outlet of pipe C on the system pressure is illustrated by overlaying a graph of the pressure at the inlet of the sprinkler arms with one of the flow rate out of pipe C:



The system pressure rises to a safe value of about 6.4 bar G.

See the [Graph Viewer](#) chapter for more details on using the viewer.

Tutorial Example 3 - Force on a 90 degree Bend

Part

6

6.2 Stage 1 - Creating the library items

Select **Options | Units** to define the unit system for the problem. The pipe schedule and valve characteristic data are in Metric units so we select that unit system for entering library data.

The pipe schedule

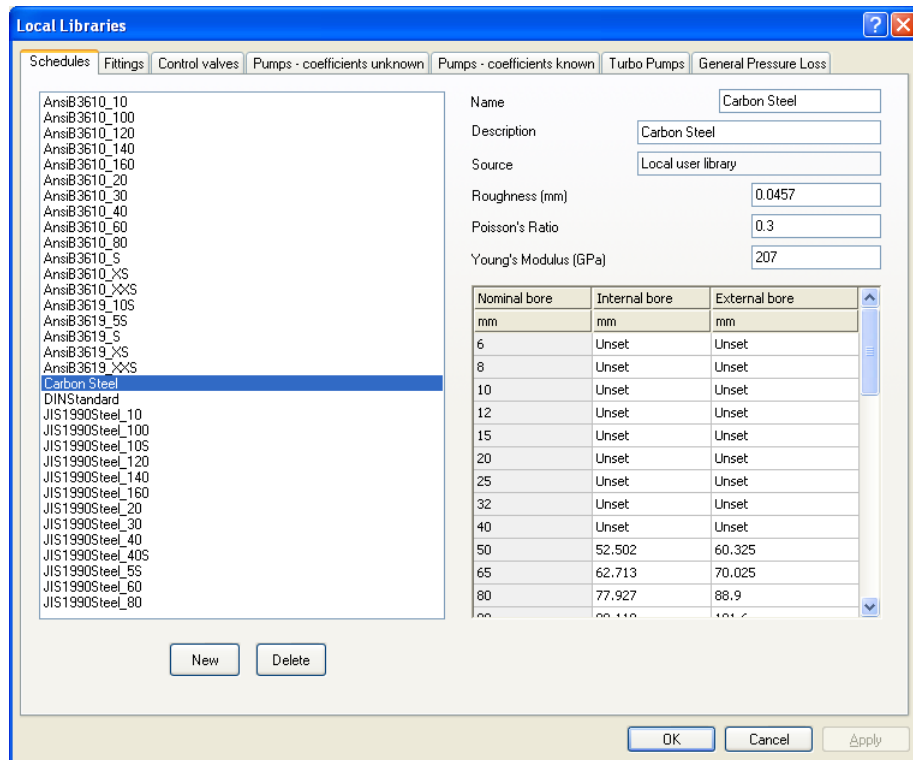
Select **Libraries | Pipe Schedule** to define a pipe schedule for the example. The pipe schedule dialog appears in which details of the pipe material and bore sizes can be specified. We will use Steel piping having the following properties:

- Schedule Name : Carbon Steel
- Roughness : 0.0457 mm
- Young's Modulus : 207 GPa
- Poisson Ratio : 0.3

The schedule diameters (in mm) are:

Nominal	Internal	External
50	52.502	60.325
65	62.713	70.025
80	77.927	88.900
90	90.119	101.600
100	102.260	114.300
125	128.190	141.300
150	154.050	168.280

Click **Apply** to accept the data. The schedule will be added to the list of pipe schedules in the left-hand window.



The valve library item

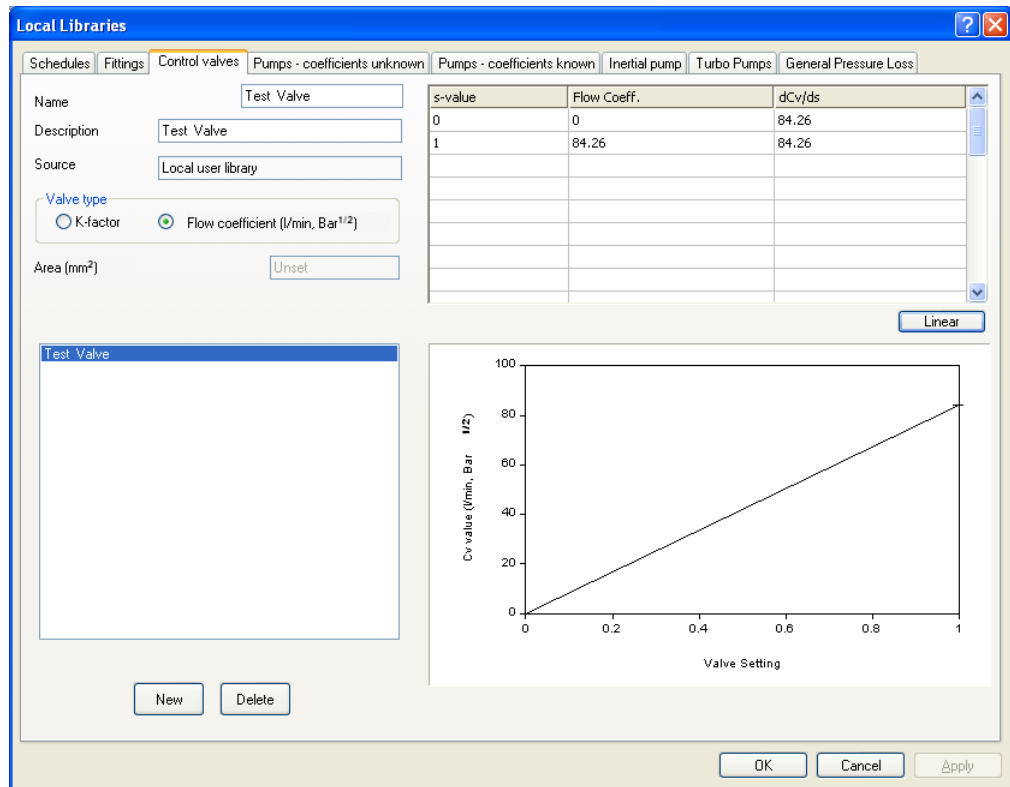
A valve characteristic can be defined by providing a series of data points, in this case describing the Cv factor versus setting (s) curve. In our example we will use a simple valve where Cv is linear with respect to the setting s.

Select the **Control Valve** tab and enter the following data in the dialog:

- Description : Test Valve
- Details of the valve characteristic are given in the table:

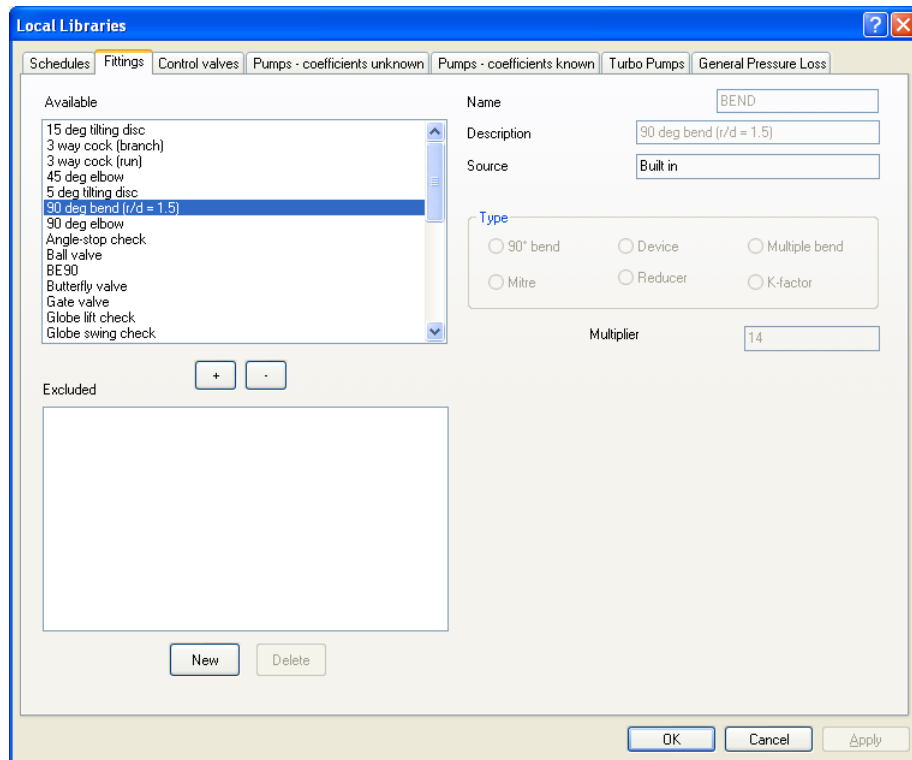
s	Cv - l/min, Bar
0	0
1	84.26

PIPENET can calculate the curve gradients based on the provided data points by pressing the **Linear** button. Alternatively the user can supply their own gradient values. If you do this you should see the valve definition as:-



Fittings

We will use a user-defined fitting, placed on the pipe labeled 'Bend', to represent the frictional effect of the 90 degree bend. Select the **Fittings** tab. Select a 90° bend and enter its details as below:



On pressing **Apply** the fitting will be added to the list of user-defined fittings in the top left-hand window and is available for use. If you want to retain a fitting but make it unavailable for use, select the fitting in the top left-hand window followed by the - button - the fitting now appears in the bottom left-hand window. To make it available again, select the fitting in the bottom right-hand window, followed by the + button.

Saving the library

Once all the items have been defined simply select the **File | Save** menu option to save the data file along with the local user library.

6.3 Stage 2 - Problem Initialisation

Having defined and saved the library files we can initialise the problem from the options menu.

Title

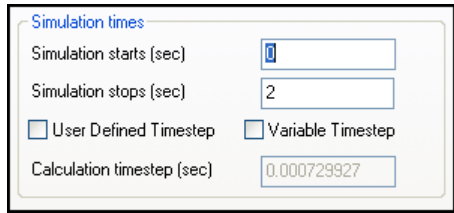
Select **Options | Title** and define the problem title as 'Forces on a 90 degree bend'.

Units

Select the **Units** tab and select the **Metric** option as the Units system.

Time step

Select the **Transient options** tab to define the start and stop times for the simulation. In this example we will consider the time interval 0 to 2 seconds.



6.4 Stage 3 - Entering the network details

Pipes

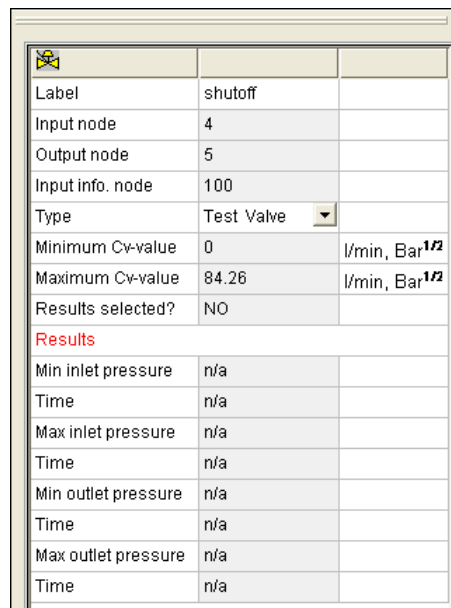
We are now ready to enter the physical components of the network by drawing the three pipes on the schematic and setting the properties as:

Label	Input node	Output node	Diameter (mm)	Length (metres)	Elevation (metres)	Fittings
1	1	2	100	10	0	none
2	2	4	100	10	-10	none

For the third pipe we include the user-defined fitting by selecting it from the Available list and Inserting one fitting:

Valves

To enter the valves we use the Operating valve tool and when drawn, select the type as 'Test valve'. The Properties window should then appear as:



Here we show the valve with the valve label changed to shutoff.

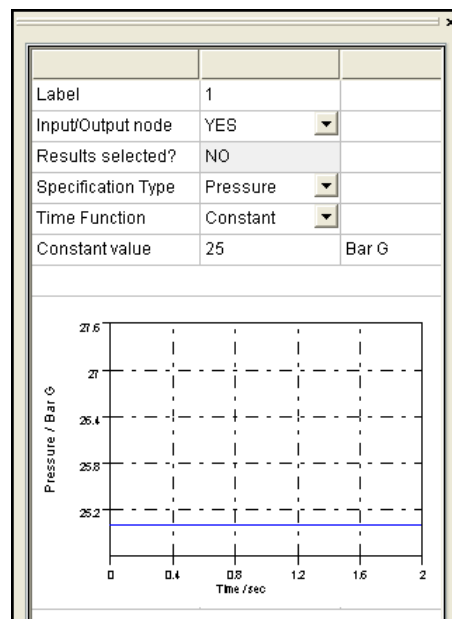
6.5 Stage 4 - Specifications

There are three specifications in our problem: pressure specifications at the two boundaries and the information setting of the valve. Specify constant pressures at the boundary nodes 1 and 5 of 25 bar G and 10 bar G respectively.

We wish to simulate the sudden closure of the valve. To do this we define a *Power Ramp* as its information setting. The valve setting will change linearly from fully open ($s = 1$) to closed ($s = 0$) between 0.1 and 0.3 seconds.

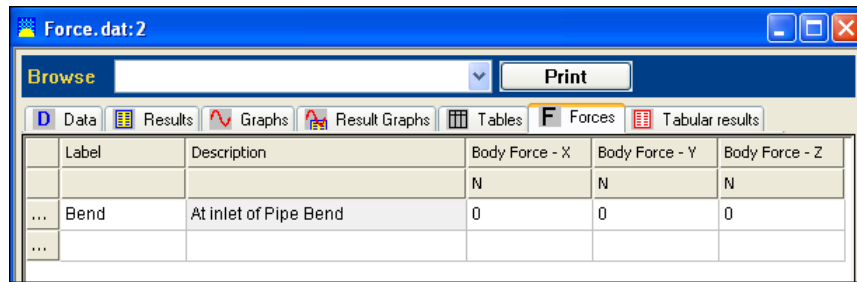
Label	Reserv	Outlet
Type	Pressure	Pressure
Node	1	5
Time function	Constant	Constant
Value	25 bar Gauge	10 bar Gauge

For example, the Properties window with node 1 selected appears as:



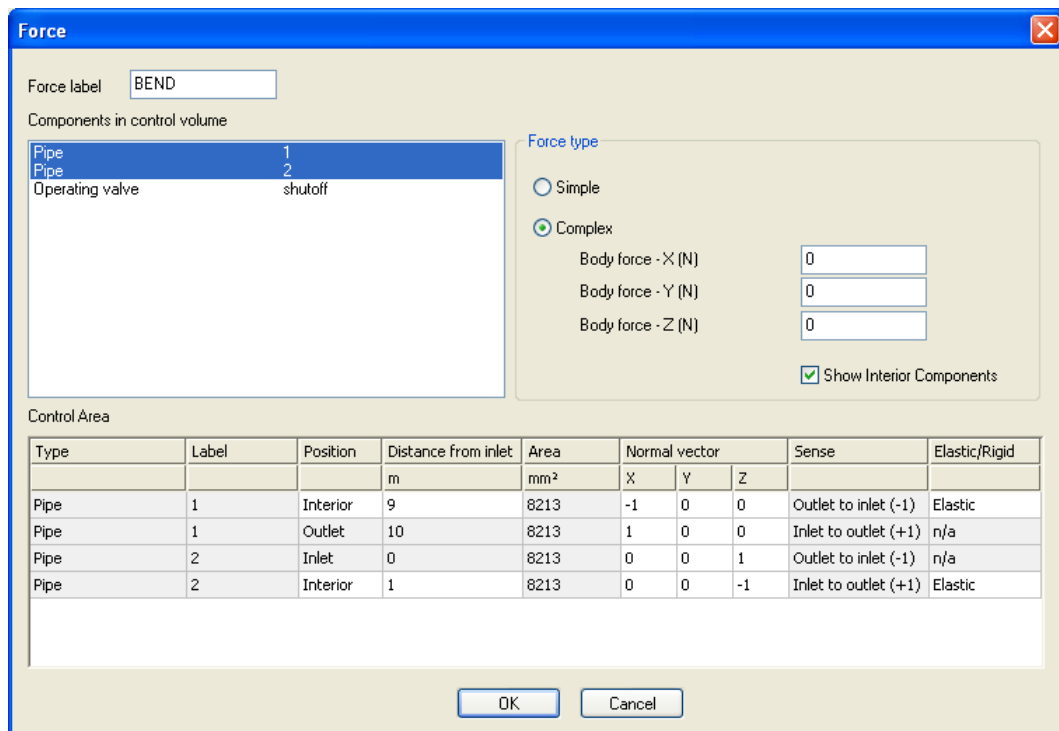
6.6 Stage 5 - Defining forces

Forces are defined using the **Forces** tab in the Tabular grid. When first selected the display might typically be as follows:



Label	Description	Body Force - X	Body Force - Y	Body Force - Z	
...	Bend	At inlet of Pipe Bend	0	0	0
...					

This view shows the list of defined forces and a description. Clicking on the ... symbol in the first column of one of the rows displays a dialog where full details of the force can be viewed and edited:



Force label: BEND

Components in control volume:

Pipe	1
Pipe	2
Operating valve	shutoff

Force type:

Simple

Complex

Body force - X (N): 0

Body force - Y (N): 0

Body force - Z (N): 0

Show Interior Components

Control Area:

Type	Label	Position	Distance from inlet m	Area mm ²	Normal vector			Sense	Elastic/Rigid
					X	Y	Z		
Pipe	1	Interior	9	8213	-1	0	0	Outlet to inlet (-1)	Elastic
Pipe	1	Outlet	10	8213	1	0	0	Inlet to outlet (+1)	n/a
Pipe	2	Inlet	0	8213	0	0	1	Outlet to inlet (-1)	n/a
Pipe	2	Interior	1	8213	0	0	-1	Inlet to outlet (+1)	Elastic

OK Cancel

The force dialog has two options - **Simple** or **Complex**. The Simple option is used to calculate forces along a straight line. In our example we wish to calculate the total force on the bend so we use the **Complex** option. This allows more complicated force definitions, for example involving multiple components. Click the **Complex** option at the upper right of the dialog. The external body force is zero here as there are no restraints. To define the force we simply have to define the control surface for the control volume containing the bend. Select the component as the pipe labeled 'Bend' and define the position, normal vector and flow direction of the two control surfaces.

6.7 Stage 6 - Specifying tables

Tables are defined using the **Tables** tab in the Tabular grid. When first selected the display might typically be as follows:

Browse Pipe 1 - Reservoir to Bend				
Add Delete Print				
D Data G Graphs R Result Graphs T Tables F Forces				
Title 1	Pipe 1 - Reservoir to Bend			
Title 2				
Title 3				
Title 4				
Column	Component type	Component label	Variable	Value
+ 1	Pipe	1		
... 5				

The table is divided into two parts; at the top are four title lines and below this the definition of the table. The drop-down box at the top indicates which table is currently displayed, in this case **Pipe 1 - Reservoir to Bend**.

The table definition as initially displayed displays a line for each component, identified by its type and label together with a start column. The last row (with the symbol ... in the first column) indicates the next free column available (there are a maximum of eight columns allowed in a table). In the above example the pipe with label **1** uses the first four columns of the table. To view which attributes have been selected for the table select the + symbol in column one and the display expands to include a list of the variables:

Browse Pipe 1 - Reservoir to Bend				
Column	Component type	Component label	Variable	Value
--	1	Pipe	1	
			<input checked="" type="checkbox"/> Inlet pressure	
			<input checked="" type="checkbox"/> Outlet pressure	
			<input checked="" type="checkbox"/> Inlet flowrate	
			<input checked="" type="checkbox"/> Outlet flowrate	
			<input type="checkbox"/> Pressure at...(m)	
			<input type="checkbox"/> Flowrate at...(m)	
			<input type="checkbox"/> Cavity vol at...(m)	
			<input type="checkbox"/> Pressure at n point...	
			<input type="checkbox"/> Flowrate at n points...	
			<input type="checkbox"/> Cavity vol at n points...	
			<input type="checkbox"/> Length of cavity volu...	
			<input type="checkbox"/> Length of liquid colu...	
			<input type="checkbox"/> Volume of vapour c...	
			<input type="checkbox"/> Vapour cavity volum...	
			<input type="checkbox"/> Liquid holdup	
...	5			

The + symbol now changes to a - symbol which if selected will once again hide the details of the attributes selected. De-selecting or selecting one or more of the available attributes will immediately update the number of columns used.

To add extra columns to the grid click on the last row (the one containing the ... symbol in column 1) and select from the list of available component types and component labels displayed. Finally select the attributes to be displayed.

6.8 Stage 7 - Specifying graphs

Selecting results

Components for which results (graphs) have been selected are displayed in green using the default colouring scheme. To request results for a particular component, right-click on the component and select the **Select Results** option. This is the same method that was used in previous versions of PIPENET Transient. For non-pipe components this will display a list of available variables - select the single result required, for example **Pressure**, or **All** to accumulate results for all attributes.

Removing results

To remove results for a component right-click on the component and select the **Remove Results** option.

Selecting and removing results in the Tabular Grid

A summary of which results have been selected can be viewed on the [Tabular grid](#) - select the **Graphs** tab when all selected results are displayed, each result being in one of two categories: variable against time (all components) or variable against distance (pipes only).

Component type	Component label	Variable
Variable against time		
Pipe	1	All variables
Pipe	2	All variables
Transient force	BEND	All variables
...		
Variable against distance		
Pipe	1	All variables
Pipe	2	All variables
...		

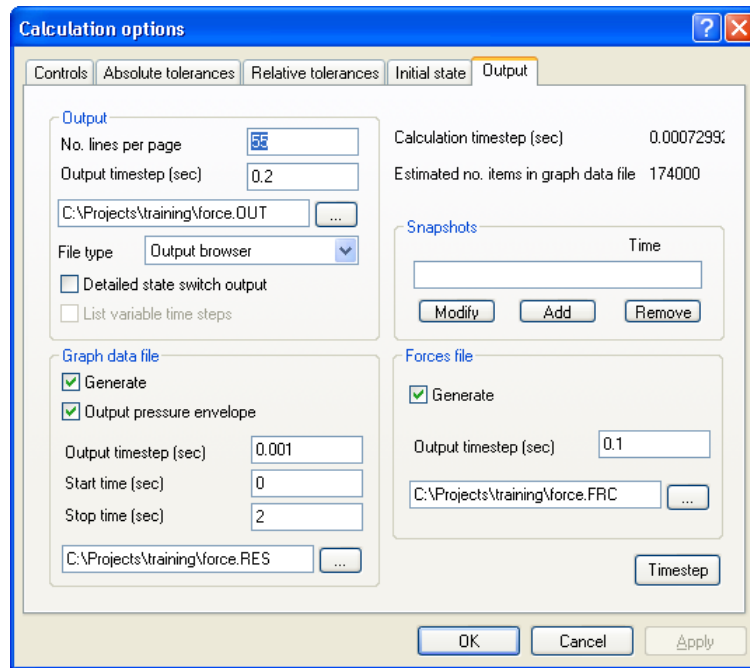
The first column provides the component type, the second the component label and the third the selected variable), only the last of these is directly editable. To add a new result in one of the two categories:

1. Select line containing ... in the first column the first blank line.
2. Select the component type from the drop-down list in the second column of the newly created row.
3. Select the component label in the third column.
4. Select the variable(s) required.
5. Finally, select the symbol in the first column to commit the change.

A result may be deleted by right-clicking in the first column of a row and selecting the **Delete graph** option.

6.9 Stage 8 - Performing the calculation

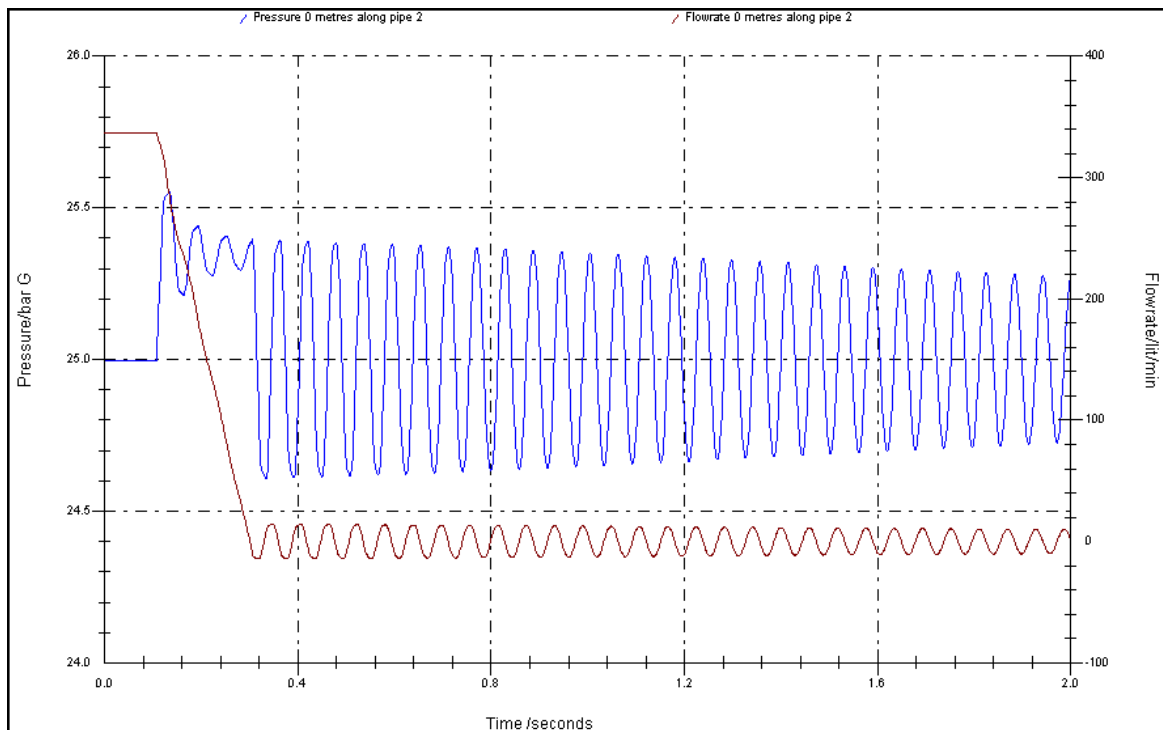
We will use default settings for the initial state and control parameters. Select **Calculation | Options** to define output filenames for the report, graph and force files. The force output file is an ASCII file containing results for the defined forces. It is useful for interfacing with pipe stress analysis programs. We select a small output timestep for the graph file so the Pipe Variable Movie runs smoothly. The completed output dialog is shown below:



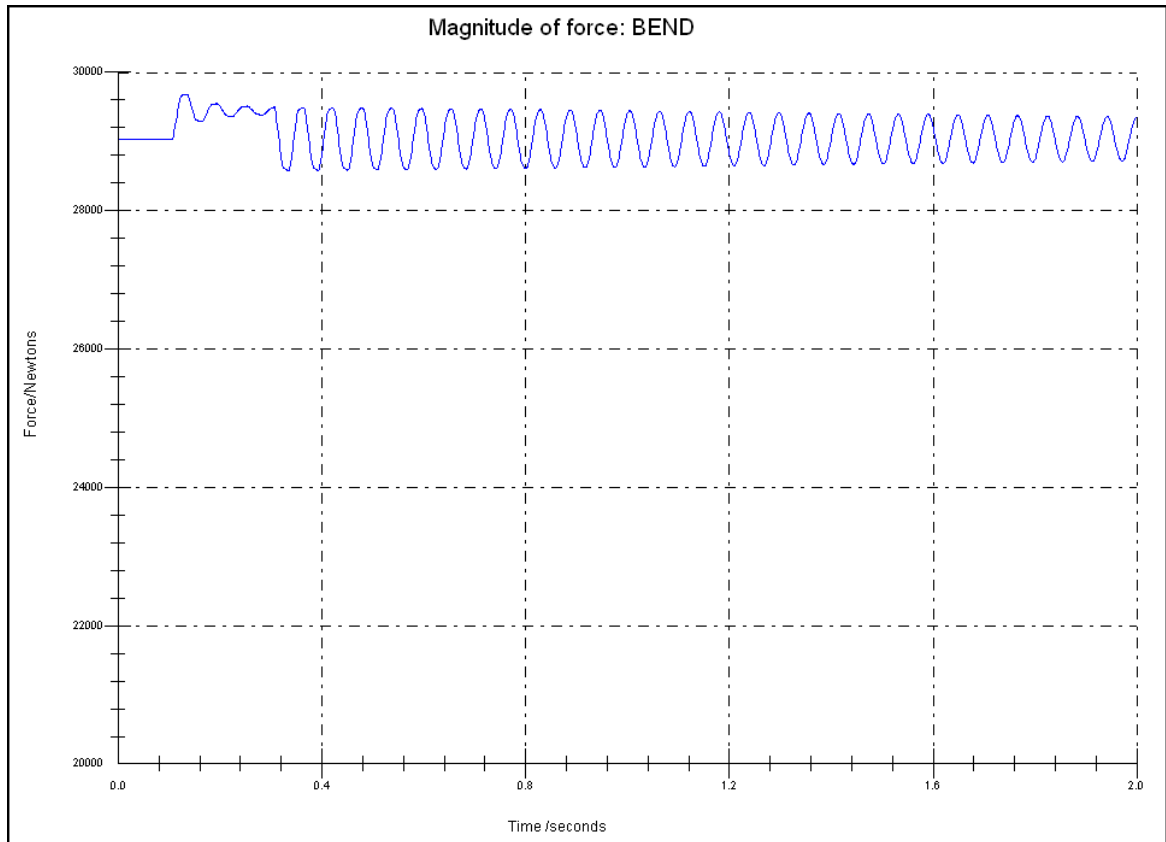
Select **OK** and run the calculation.

Viewing and Interpreting the Results

The results are easier to interpret in graphical form. The variable against time graphs show the solution is constant up to 0.11 seconds. When the valve closes a pressure wave propagates through the system. The resulting oscillations are lightly damped.



When the valve closes, the force on the bend oscillates between about 28500 and 29500 Newtons:



While the actual dynamic force (the difference from the steady state value) is not excessive here, the lightly damped oscillations could cause fatigue damage in the valve or pipe.

It is also interesting to observe the movie of pipe pressure for either of the pipes. This gives a clear demonstration of the pressure wave moving along the pipe after the valve has closed.

Viewing Forces Extrema

On completion of a calculation, the forces extrema can be viewed in the Tabular grid by selecting the **Tabular Results** tab and then the option **Forces Extrema** from the combo box.

The screenshot shows a software window titled "Force.dat:2" with a menu bar containing "Browse", "Forces Extrema", and "Print". Below the menu bar is a toolbar with icons for "Data", "Results", "Graphs", "Result Graphs", "Tables", "Forces", and "Tabular results". The main content area displays a table titled "Complex forces" with the following data:

Complex force for BEND	Maximum	Time	Minimum	Time
	N	sec	N	sec
FX	-20217.98	0.34	-20991.63	0.14
FY	0	0	0	0
FZ	-20191.82	0.34	-21036.63	0.14
Magnitude	29717.9	0.14	28575.75	0.34

Other Topics

Part

7

7 Other Topics

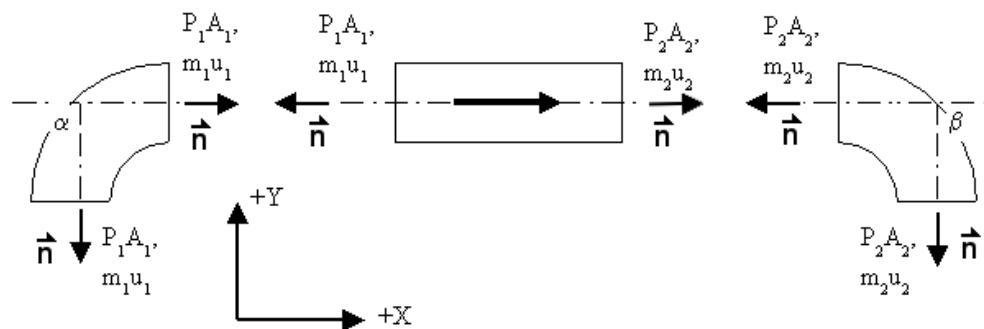
7.1 Forces

A powerful feature of Transient Module is its ability to calculate the hydraulic forces acting on specific parts of the network. It is also possible to compute dynamic forces.

Control volumes and surfaces

When performing a force calculation, information about the local geometry of the network is required. This is given by defining a Control Volume that encloses the part of the network where the force is to be calculated. A control volume can enclose any number of network elements (except elastic connections) and can have an external force F_e associated with it. The control surfaces are the surfaces through which fluid flows and are defined by their normal vector \mathbf{n} . The direction of vector \mathbf{n} is shown in the figure below.

An example of a control volume about a straight pipe with an elbow at each end is shown below:



The total force, ΣF , acting on the defined control volume can be calculated by:

$$\sum F = \sum_{i=1}^n (F_e + F_p + F_m + F_f)$$

where:

- F_e is the external force
- F_p is the force due to fluid pressure acting on the elbow
- F_m is the force due to fluid momentum changing at the elbow
- F_f is the friction between the fluid and the pipe or other devices.
- \vec{n} is the normal vector of the control surface

In the above scenario, we assume

1. The physical volume in the elbows is ignored, or included in the straight pipe

2. Friction in the elbows is ignored, or included in the straight pipe
3. The inlet pressure of the elbow equals to the outlet pressure.

Two types of force are defined in PIPENET, a **simple force** and a **complex force**.

Simple Force

A simple force can contain any number of components, or any type, as long as they are on the same axis. It is one dimensional force and the direction is always along this axis. Therefore, the external force cannot be included because the direction may be different. In addition, all components must have the same direction, that is outlet to inlet.

In the results, a positive force means the force is directed with the group direction and a negative force means the force is directed against the group direction.

The definition of simple force is very simple. It just need to know the position of the control surfaces and if fluid changes direction there. See the dialogue below:

Force label: F345v1

Components in control volume

Pipe	1
Pipe	3
Pipe	5
Pipe	2
Pipe	4
Pipe	6
Pipe	7
Operating valve	1
Operating valve	2

Force type

Simple

Complex

Body force -X (N): 0

Body force -Y (N): 0

Body force -Z (N): 0

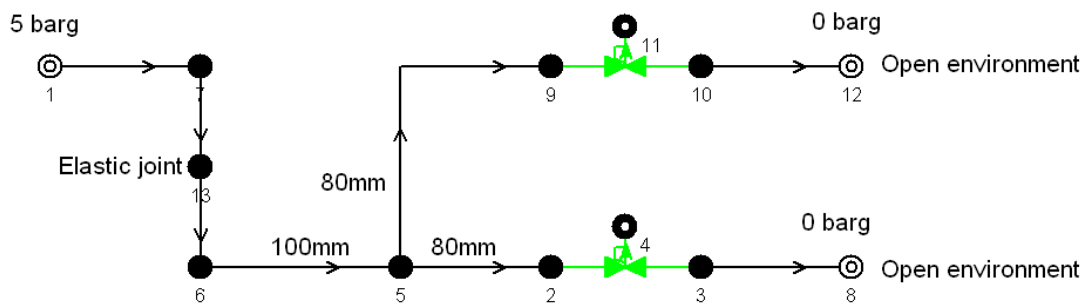
Show Interior Components

Control Area

Type	Label	Position	Distance from inlet	Area	Elastic/Rigid
			m	mm ²	
Pipe	3	Inlet	0	8229	Elastic
Pipe	5	Outlet	25	4776	Elastic

OK Cancel

The above simple force includes two different size pipes, i.e. there is a reducer between them. In fact, a simple force can also contain valves, tees etc.



In the above network connections in the force definition window must be set to **Elastic** at elastic joints and at the boundaries to the open environment (e.g. nozzle) as in nodes 8, 12 and 13. At other internal points and at boundaries to closed environments (e.g. closed tanks).

Complex force

Complex force: this 3D force can include external force F_b . The defined control volume can contain any number of components and flow direction can change any times. Besides the position of the control surfaces and fluid direction, the normal vector (pointing outwards from control volume) of the control surfaces is also required in a complex force, see the dialogue below:

Force

Force label: F345v1

Components in control volume

Pipe	1
Pipe	3
Pipe	5
Pipe	2
Pipe	4
Pipe	6
Pipe	7
Operating valve	1
Operating valve	2

Force type

Simple

Complex

Body force - X (N): 0

Body force - Y (N): 0

Body force - Z (N): 0

Show Interior Components

Control Area

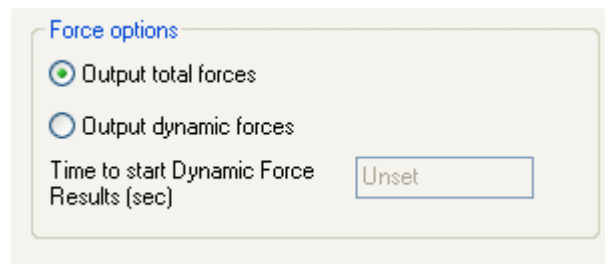
Type	Label	Position	Distance from inlet m	Area mm ²	Normal vector			Sense	Elastic/...
					X	Y	Z		
Pipe	3	Inlet	0	8229	-1	0	0	Outlet to inlet (-1)	Elastic
Pipe	4								
Operating valve	1								
Pipe	5	Outlet	25	4776	1	0	0	Inlet to outlet (+1)	Elastic

OK Cancel

Dynamic forces

It is often useful to see how the force varies from a particular static value. This is of particular use if results from the force calculation are to be used by a pipe stress analysis

program. Transient Module can produce results of Dynamic Force rather than Total Force. This can be defined in the **Options | Transient Options** menu item.



The user specifies a time at which forces are considered static. From this time onwards the force results show the deviation from this static value.

Maximum and minimum force values

The calculator determines the maximum and minimum values for every force during the simulation. For each simple force the following values are recorded:-

- the maximum force
- the minimum force
- the time at which the minimum value occurs
- the time at which the maximum value occurs

For each complex force the following values are recorded:-

- the maximum F_X , F_Y , F_Z
- the minimum F_X , F_Y , F_Z
- the maximum and minimum magnitudes
- the times at which the maximum and minimum values are recorded.

7.2 Time control and time steps

Of crucial importance to the accuracy of results produced by Transient Module is the calculation time step. This can be generated automatically by PIPENET based on the pipes in the system, or specified directly by the User. The automatic generation of timestep is influenced by two factors:

1. The relative length of pipes in the network.
2. The user-defined time steps for tabular, graphical and force output.

Pipe length and calculation timestep

The maximum allowed time step for a calculation is the time taken for a pressure wave to travel the length of the shortest pipe in the network. In calculating a time step PIPENET divides all pipes in the network into sections. This division depends on the shortest pipe length and something called the interpolation error.

Consider a system with two pipes having lengths 100 m and 150 m and a wavespeed of 1000 m/s in each pipe



The simplest way to divide the network is to use 100 m long sections. The error in using a 100 m section on the longer pipe is:

$$e = \frac{150 - 100}{150} = 0.333\dots$$

This error e is the interpolation error. The default tolerance for e is one (the network is divided in the simplest possible way). This tolerance can be changed by selecting the **Calculation | Controls...** dialog.

The interpolation error can be eliminated on both pipes by using 50 m sections. We can achieve this by reducing the interpolation error tolerance to 0.28, as can be seen in the table below.

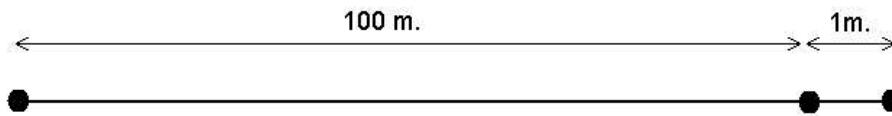
Interpolation tolerance	1 (default)	0.3	0.28
Number of sections in Pipe A	1	1	2
Interpolation error in Pipe A	0	0.29	0
Number of sections in Pipe B	1	2	3
Interpolation error in Pipe B	0.33	0.05	0
Automatically calculated time step/s	0.1	0.071	0.05

Output time steps and calculation time steps

The output time steps (for tabular, graphical and force results) also have an influence on the calculation time step. The calculation time step cannot be larger than any of the output time steps. Thus the automatically calculated time step can be reduced by reducing one of the output time steps.

User-defined time steps and short pipes

If the network contains pipes having short length the calculation time step can be very small. Sometimes these short lengths of pipe have little influence on the transient results even though they have significant effect on the time step chosen. For example, consider two pipes in series having length 100 m and 1 m. The wavespeed in each pipe is 1260 m/s.

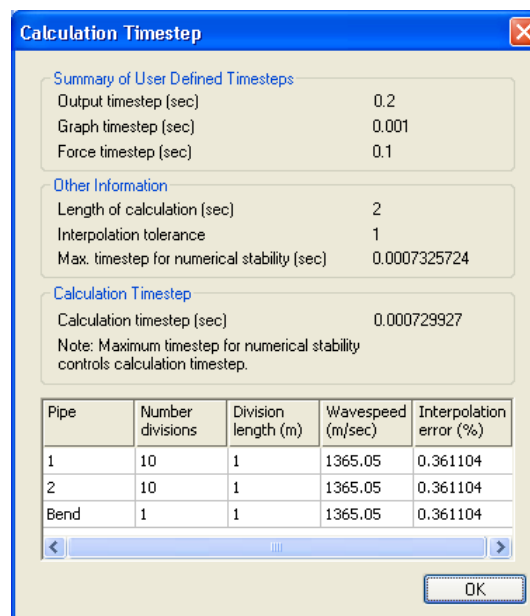


The calculation time step chosen by PIPENET is 0.000794 seconds because this is the time taken for a pressure wave to travel across the shorter pipe. But this time is insignificant compared to the time the pressure wave takes to cross the longer pipe (0.0794 seconds). The transient results are not affected significantly by the shorter pipe. In fact if the shorter pipe had no influence on the calculation time step the simulation would run 100 times faster.

There are three ways for a User to remove the influence of these shorter lengths of pipe from the calculation of a time step:

1. Merge or extend some of the shorter pipes in the network. It is sometimes useful to do this initially when obtaining preliminary results. The final runs can then be made with the correct pipe lengths for maximum accuracy.
2. Replace the short lengths of pipe with a short pipe component (from the View..Short Pipes menu). This uses a different modelling equation for flow in the pipe, assuming pressure waves travel instantaneously across the pipe.
3. Define a User-defined time step from the **Options | Transient options** menu. PIPENET then automatically calculates which pipes are to be treated as 'short pipes' during the simulation.

The User can see the effect of the chosen timestep on the pipes by selecting **Calculation | Output** and then selecting the Timestep button:



The pipes which are to be treated as short pipes are labeled as such in the list. The 'short pipes' are also indicated in the output report. Note that user-defined time steps and the

short pipe model should be used with care. A pipe should only be modelled as a 'short pipe' if the User wishes to ignore transient behaviour in the pipe. Sometimes the User will want to define a user-defined timestep which is smaller than the automatically calculated timestep. This may occur in a problem having complex boundary specifications. As the specifications are not taken into account when calculating the timestep some of the detailed behaviour may not be represented properly in the calculations. For example, if a pressure specification is varying sinusoidally, the calculation timestep should be smaller than the period of oscillation.

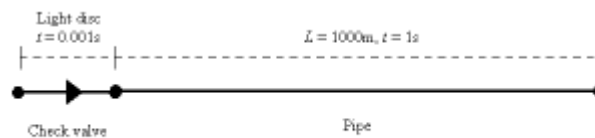
7.3 Variable time step

The need for a VTS algorithm

The need for a variable time step (VTS) algorithm arises because of the unsatisfactory performance of the existing fixed time step (FTS) algorithm which is based on the pipe only, and because of the interest in analyzing the dynamics of some components in a network in a smaller time scale. This occurs when fast components such as check valves, inertial check valves, liquid surge relief valves are parts of a network, which control the dynamics of the network.

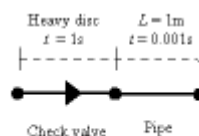
The fast component – the troublemaker

A component is said to be fast if it is the controlling factor in governing the dynamics of the network at any given time. Suppose a network consists of a check valve and a pipe of 1000 meters in length as shown in figure (a) below. If the pressure wave travels at a speed, say, 1000 m/s, upstream towards the check valve, it will take 1 s to reach the check valve and then light disc of the check valve will only take a further 0.001 s to slam shut. In this scenario, the factors governing the dynamics of the system include the pipe for $0 < t < 1\text{s}$; the check valve for $1.0\text{s} < t < 1.001\text{s}$; and, the pipe for $1.001\text{s} < t < 3.001\text{s}$ etc., but the controlling factor is the check valve. It acts as a fast component.



(a) The check valve is the fast component

The second network (b) below is identical to the first one (a) except for the physical dimensions. Here, the pipe is only 1 meter in length and the check valve has a heavy disc in contrast with the first one in figure (a) which has a light disc. The pressure wave, traveling at the same speed as above, only needs 0.001 s to reach the check valve and then the valve needs to take a further 1 s to close because of the inertia of the disc. In this case, the fast component is the pipe for $0 < t < 0.001\text{s}$.



(b) The pipe is the fast component

Understand the problem in depth – the limitation of the FTS algorithm

If for all time the only fast components in the system are pipes as in the second network (b), then the FTS algorithm can be used to solve the flow equations. This is because the time step for the integration is calculated so as not to violate the numerical stability criteria at that time: the solver determines the automatic time step at a certain point in time, by firstly, searching through the networked pipes, finds the one in which the ratio of the pipe length to wavespeed is smallest, and then, chooses the time step to be less than this ratio. (Numerically this is the Courant-Friedrichs-Levy (CFL) condition and physically, it simply ensures that causality is not violated: information can travel only as fast as the medium's sound speed.). However, such a method fails to account for other components in the network. If the user specifies the time step, s/he must ensure that the time step does not violate any stability criteria including those of fast components, to avoid program failures.

If at a particular point in time, the fast component in the network is a component other than a pipe as in the first network (a), the integration needs to be slowed down so as to fully simulate the dynamics of the component and its subsequent influence on the fluid flow through the network. If the same time step, that is used during the integration of the flow equations through the pipes, is employed then integrating past such a component may be numerically unstable leading to incorrect and more fatally, unbounded solutions being computed that force the program to crash.

Within FTS framework, the only way to achieve correct simulation (i.e. numerically stable integration) of a network consisting of the fast components other than pipes is by choosing the fixed time step small enough to cover the fastest such component. As a result, calculations are rather computationally expensive, as smaller pipe sections are taken, resulting in larger matrix problems that need to be solved at shorter time intervals.

The benefits – to the user

Using a VTS allow users to investigate the interesting dynamics of networks of a much broader application base than with a FTS. Use of the VTS will enable the user to compute flows through networks without experiencing any instabilities in the computation, due to the improved robustness and reliability. The VTS facility also provides the opportunity for a more in depth analysis of the fast dynamics in the networks that may arise from surges propagating within control loops, for example.

Because a variable time step is always chosen to be smaller than the fixed time step, it's expected that more accurate results should be achieved. This is visualised in the following figure (A) which does show the pressure curve for the VTS is smoother than the one for the FTS does when some sharp changes in the curve are involved.

A side effect of using VTS is that when a variable undergoes rapid oscillation, the extremes of this oscillation are reduced as a result of the VTS containing the effect of any "overshoot" [figure (B)] and sometimes, eliminating the effect completely [figure (C)], even after the VTS algorithm is stopped. This can have the effect of making output graphs much clearer or tidier than their FTS counterparts, allowing other interesting behavior to be more easily noticed.

Please note that in the current release, all dynamic output data files are still in fixed regular time steps, not in variable time step yet. The latter will be set on in future release.

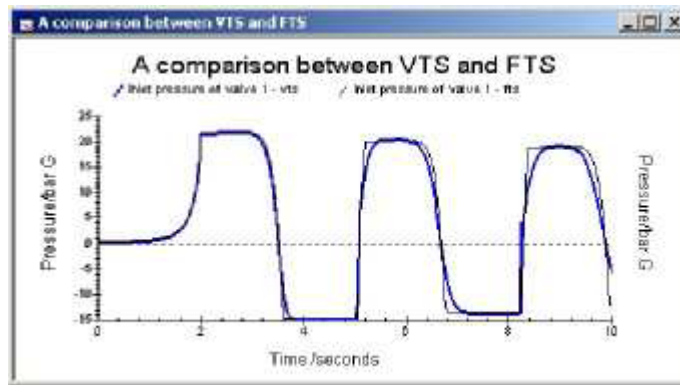


Figure A

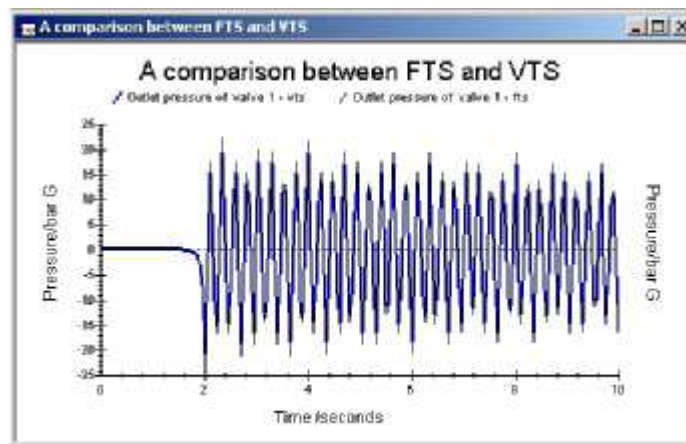


Figure B

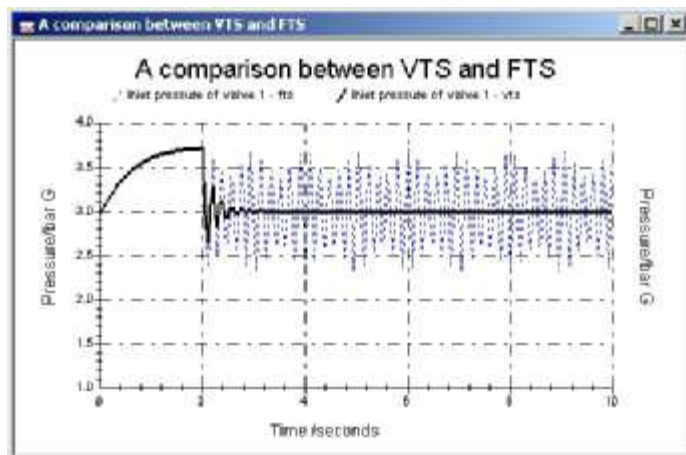
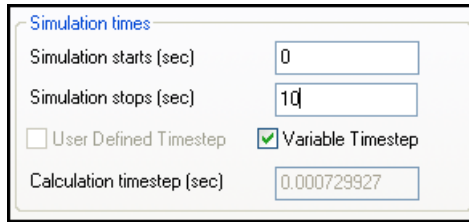


Figure C

To choose VTS, select the menu option **Options | Transient Options** and select the VTS option:



Simulation times

Simulation starts (sec) 0

Simulation stops (sec) 10

User Defined Timestep Variable Timestep

Calculation timestep (sec) 0.000729927

Please note that the User-defined time step cannot be used simultaneously with the VTS. Enabling one will disable the other.

Choosing the Graphical Timestep

While choosing the calculation is important in determining the accuracy of the results contained within the calculation itself, it is also important to consider the accuracy of the results which are returned to the user. For example, if a network experiences transient behaviour of a high frequency but the graphical timestep is chosen to be of the order of seconds, the majority of the interesting behaviour of the system will pass unnoticed, including potentially significant pressure surges.

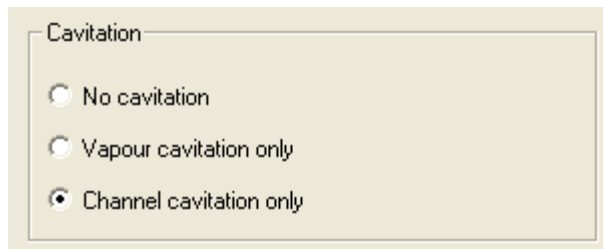
There are a number of different factors that affect the choice of graphical timestep. Of these, the calculation timestep is the most important.

There is little point in choosing a graphical timestep which is smaller than the calculation timestep, since more detail will not be discernible. In fact, if you are using the automatic timestep and choose a graphical timestep which is smaller than the automatic timestep, the automatic timestep is altered to use the smaller graphical timestep.

7.4 Vapour cavitation

Vapour cavitation occurs in a system when the pressure at a point falls below the fluid vapour pressure. The fluid vapourises, forming a cavity in the system. Cavitation generally occurs in the higher parts of the network, where the pressure due to static head is less. When the pressure rises back above the fluid vapour pressure the vapour condenses and the cavity collapses. The collapse of a cavity can cause a very large pressure surge to occur and must be avoided by careful design of the system. Transient Module is an invaluable aid in both predicting and eliminating problems due to cavitation.

The vapour cavitation option is chosen by selecting **Options | Transient Options:**



Cavitation

No cavitation

Vapour cavitation only

Channel cavitation only

In both cases the cavitation option can be switched 'on' or 'off'. Selecting the vapour cavitation option greatly increases the complexity of the calculation. For this reason it is

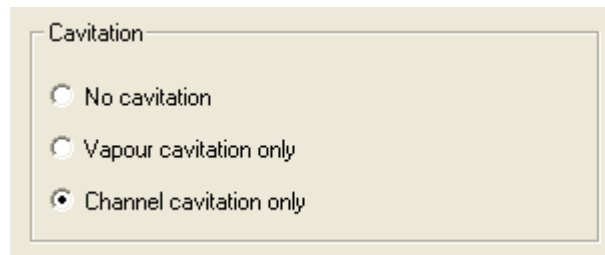
recommended that a simulation is always initially run with the vapour cavitation option switched 'off'. If a minimum pressure below the vapour pressure is listed in the output report, the simulation should be run again with vapour cavitation switched 'on'. If cavitation occurs the output results for 'pipes' will include results for the 'cavity volume' at points along the pipe. This result is the size of the cavity about the point, and increases from zero as a cavity forms in the neighbourhood of a point.

PIPENET Transient Module includes a [Vacuum Breaker](#) model which can be used to eliminate vapour cavitation from the system.

7.5 Channel cavitation

While vapour Cavitation is intended to model the growth and collapse of cavities and the consequent pressure surge in the system, it cannot predict vapour bubble formation, so it does not take into account the effect this has on fluid flowing through components. In particular, pipes, where the majority of cavities form continue to behave as though they are completely full. As a result of this, the phenomenon known as "slack line flow" or "open channel flow" is not experienced if the presence of cavities is modelled using the vapour cavity model. The open channel cavitation model is intended to remedy this. Specifically designed to be applicable to pipes, it realistically models the flow through pipes when a section of the fluid in the pipe is actually vapour.

The vapour cavitation option is chosen by selecting **Options | Transient Options:**



There is a choice between no cavitation modelling, modelling using the vapour cavitation model or modelling using the channel cavitation model.

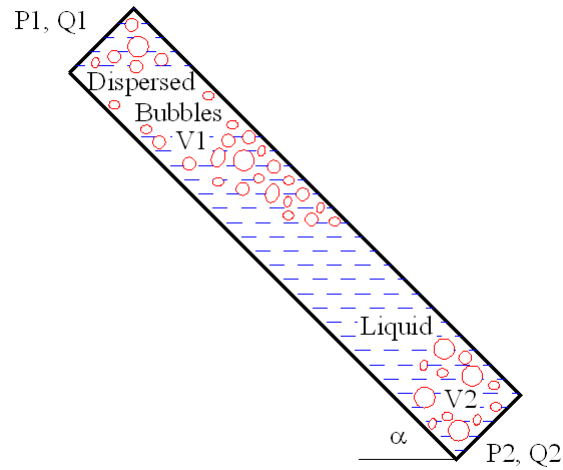
Selecting the cavitation option greatly increases the complexity of the calculation. For this reason it is recommended that a simulation always initially runs with the channel cavitation switched off. If a region exists with minimum pressure below the vapour pressure is listed in the output report, the simulation should be run again with channel cavitation switched 'on'.

If cavitation occurs in a pipe, the output results for pipes will include results for the:

- Volume of vapour cavity
- Volume fraction of vapour cavity.
- Position of vapour cavity.
- Length of vapour cavity
- Liquid holdup of vapour cavity.

These results define the size and shape of the vapour cavity formed within the pipe and are best described with the aid of diagrams:-

Bubbly flow



$$V_c = 0$$

$$K_c = 0$$

$$X_c = 0 \quad (\alpha < 90^\circ) \quad X_c = L \quad (\alpha > 90^\circ)$$

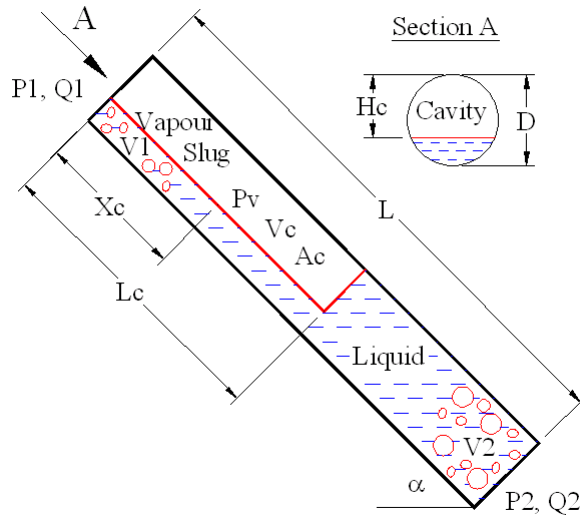
$$L_c = 0$$

$$K_f = 1$$

$$Vn_1 = V1$$

$$Vn_2 = V2$$

Slug flow



$$dV_c/dt = Q_2 - Q_1$$

$$K_c = V_c/V$$

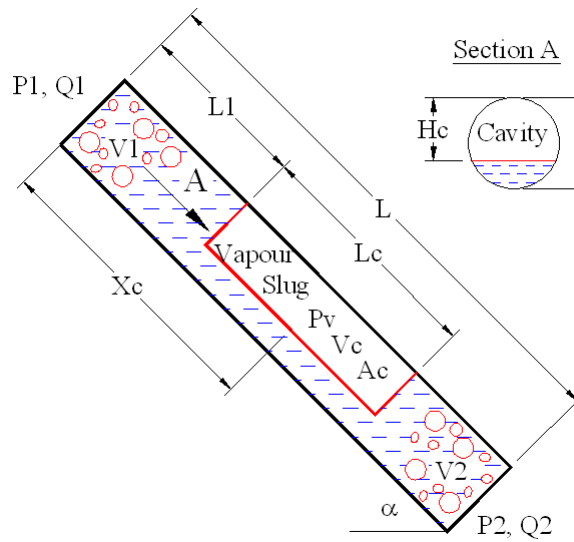
$$X_c = L_c/2$$

$$L_c = V_c/A_c$$

$$K_f = 1 - H_c/D$$

$$V_{n_1} = V_1 + V_c$$

$$V_{n_2} = V_2$$



$$dV_c/dt = Q_2 - Q_1$$

$$K_c = V_c/V$$

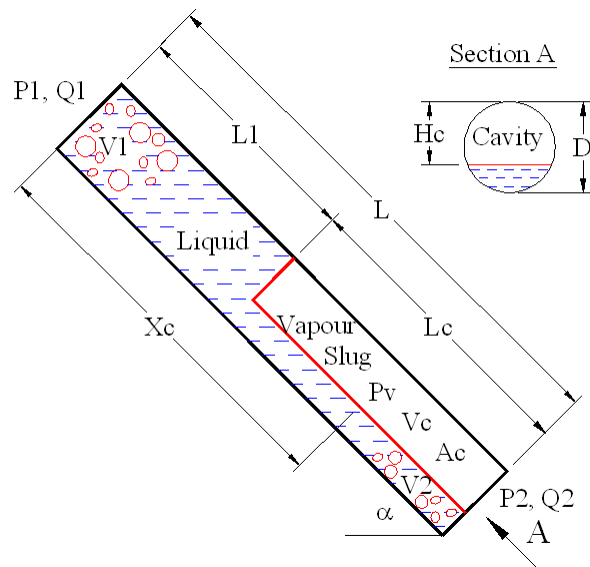
$$X_c = L_1 + L_c/2$$

$$L_c = V_c/A_c$$

$$K_f = 1 - H_c/D$$

$$V_{n_1} = V_1$$

$$V_{n_2} = V_2$$



$$dV_c/dt = Q_2 - Q_1$$

$$K_c = V_c/V$$

$$X_c = L - L_c/2$$

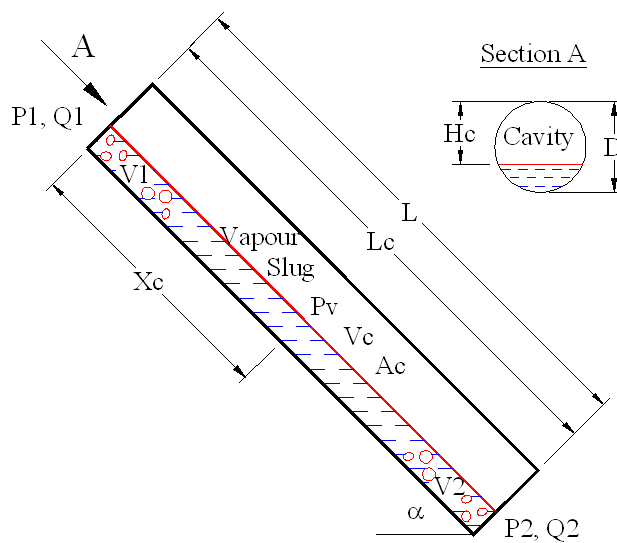
$$L_c = V_c/A_c$$

$$K_f = 1 - H_c/D$$

$$Vn_1 = V1$$

$$Vn_2 = V2 + V_c$$

Stratified flow



$$dV_c/dt = Q_2 - Q_1$$

$$K_c = V_c/V$$

$$X_c = L/2$$

$$L_c = L$$

$$K_f = 1 - H_c/D$$

$$Vn_1 = V1 + V_c$$

$$Vn_2 = V2 + V_c$$

Where:

Ac cross-section area of vapour slug.

D pipe diameter.

Hc cross-section height of vapour slug.

Kc volume fraction of vapour slug.

Kf liquid holdup of vapour slug.

L pipe length.

Lc length of vapour slug.

Pv vapour pressure.

P1 inlet pressure.

P2 outlet pressure.

Q1 inlet flow rate.

Q2 outlet flow rate.

V pipe volume.

Vc volume of vapour slug.

Vn1 volume of vapour cavity (dispersed bubbles and vapour slug) at the inlet node.

Vn2 volume of vapour cavity (dispersed bubbles and vapour slug) at the outlet node.

VI volume of dispersed bubbles formed at the pipe inlet.

V2 volume of dispersed bubbles formed at the pipe outlet.

Xc position of vapour slug (distance from the pipe inlet).

α pipe inclined angle.

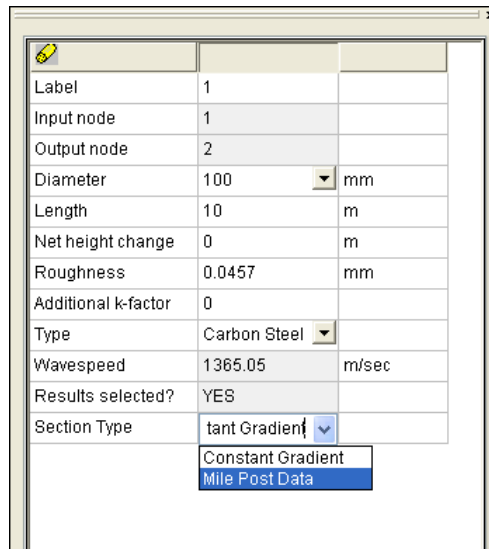
There are a number of restrictions on the use of channel cavitation model that users should be aware of and these are listed below, including a workaround wherever appropriate.

- Channel cavitation can model bubbly flow, slug flow and stratified flow but cannot model churn, annular and mist flows.
- Channel cavitation model is not applicable for long section pipes.
- Slip velocity between liquid and vapour cavity is ignored.
- Each pipe can and only includes a vapour slug. Vapour slugs in a pipe will merge automatically.
- Vapour slug cannot merge with the dispersed bubbles formed at the flow nodes.

7.6 Long sections

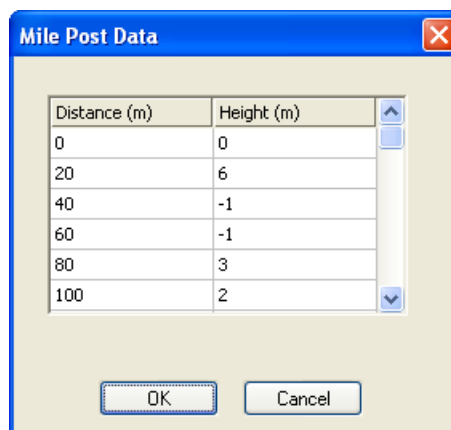
Pipes may be modelled using either a **constant gradient** or **mile post data**. Constant gradient pipes act as a single pipe with inlet at an initial height and outlet at a final height and the pipe running straight between them. A pipe with the mile post data attribute acts as a number of pipes, all sharing the same pipe type, roughness and diameter, where the inlet is at an initial height and the outlet at a final height but where the pipe itself may have bends which result in the gradient of the pipe being piecewise linear rather than constant.

A pipe is modelled using mile post data, by selecting the appropriate pipe and then selecting the **Mile Post Data** option in the **Section Type** field:



Label	1	
Input node	1	
Output node	2	
Diameter	100	mm
Length	10	m
Net height change	0	m
Roughness	0.0457	mm
Additional k-factor	0	
Type	Carbon Steel	
Wavespeed	1365.05	m/sec
Results selected?	YES	
Section Type	Constant Gradient Mile Post Data	

Selecting the **Mile Post Data** option and entering return or the **Tab** key will display a dialog via which can be entered a set of mile-post data, where the distance along the network is paired with the height at various points in order to define the height profile of the section.



Distance (m)	Height (m)
0	0
20	6
40	-1
60	-1
80	3
100	2

OK Cancel

When the **OK** button is pressed, the profile is checked for consistency and any errors are displayed. If the profile is correct then on selecting the **OK** button the profile is displayed in the Property Window for the pipe. Also, upon returning to the Pipe Dialog, the values for net height change and length will no longer be editable and will instead reflect the properties of the entered mile post data.

Label	LONG/1	
Input node	1	
Output node	2	
Diameter	100	mm
Length	200	m
Net height change	-4	m
Roughness	0.0457	mm
Additional k-factor	0	
Type	Carbon Steel	
Wavespeed	1365.05	m/sec
Results selected?	YES	
Section Type	Mile Post D...	

(Double click graph to edit mile post data)

The model will not give correct results if there is vapour cavitation within a pipe defined using mile post data. In such a case, the section(s) of the pipe which experience vapour cavitation should be separated and modelled as individual pipes. Note that selection of the **Mile post data** option will not permit any channel cavitation within any of the sections.

7.7 Pressure envelopes

For networks in which the actual piping system is linear, it is possible to obtain information on the pressure extrema experienced by the network. Select the **Calculation options** dialog and then the **Output** tab, and then select the Output pressure envelope option - the pressure envelope information will be output to the results file and may then be viewed in the Graph Viewer.

Graph data file

Generate

Output pressure envelope

Output timestep (sec)

Start time (sec)

Stop time (sec)

...

A number of graphs are available including the maximum and minimum pressures experienced along the piping section of the network. In addition, the numerical values are output to the output file for reference.

In order to present the pressure envelope results in a meaningful way, PIPENET will choose a path through the network from one specification to another that passes through all "pipe-like" components. In addition, the direction of the network will be taken to be that which has the fewest reversed components. If it is found that a network appears to be backwards (as a result of viewing the Network Height Profile for example), this can be remedied by reversing the direction of components which are not oriented in the correct direction.

The Schematic

Part

8

8 The Schematic

8.1 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.

8.2 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](#).

8.3 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.

8.4 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.

8.5 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.

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 or copied. 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.

8.6 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.

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 or copied. 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.

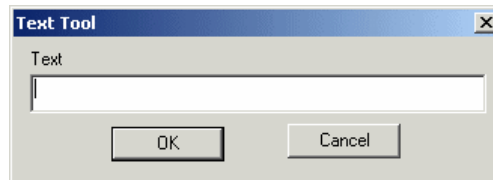
Copying selected items

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

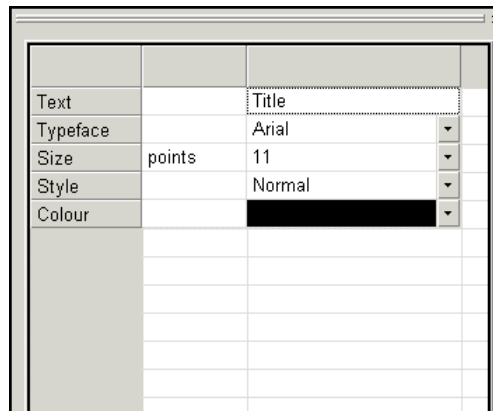
module. Copy/paste are activated via the **Edit** menu options, the **Copy** and **Paste** buttons on the toolbar, or by right clicking within the defined rectangle to display a pop-up menu of options.. or the Copy and Paste buttons on the toolbar..

8.7 Text Tool

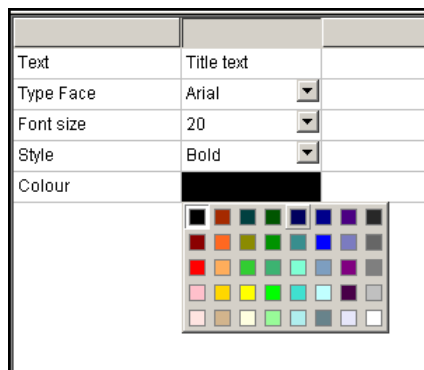
The Text Tool is used to place text on the schematic, for example, as titles and additional labeling information. To place a text item on the schematic click the left mouse at the approximate point at which the first character is to appear; the following dialog appears:



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:



Selecting one of the small coloured cells will use that colour for the text.

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 edit the properties 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.

8.8 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 pipe with negative length. 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.

8.9 Pipe Component Tools

These tools are used for placing components on a pipe, the only example in Transient with the Spray option, being 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.

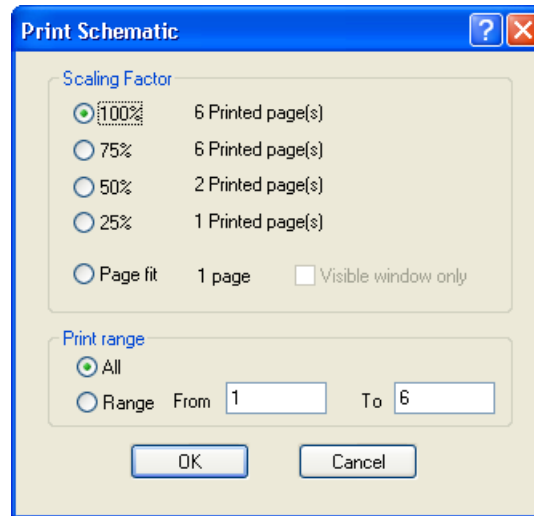
With any of the attribute element tools selected it is possible to move any attribute element along the length of a pipe using click and drag; see also the section on the [Selection Tool](#).

8.10 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.

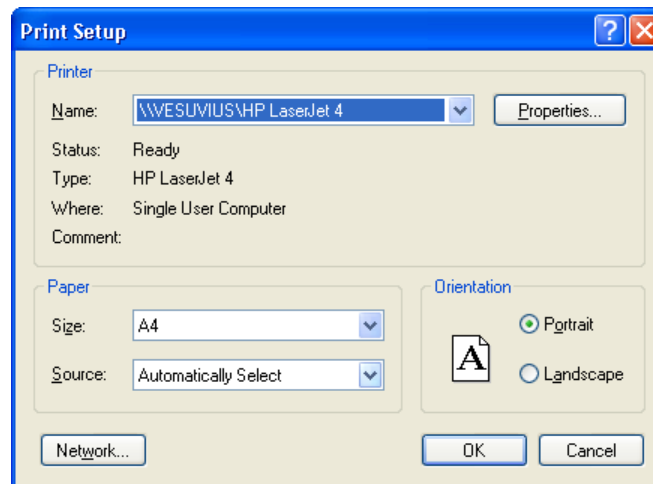
To print a schematic select the **File| Print** option, this will display a dialog box showing a

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.

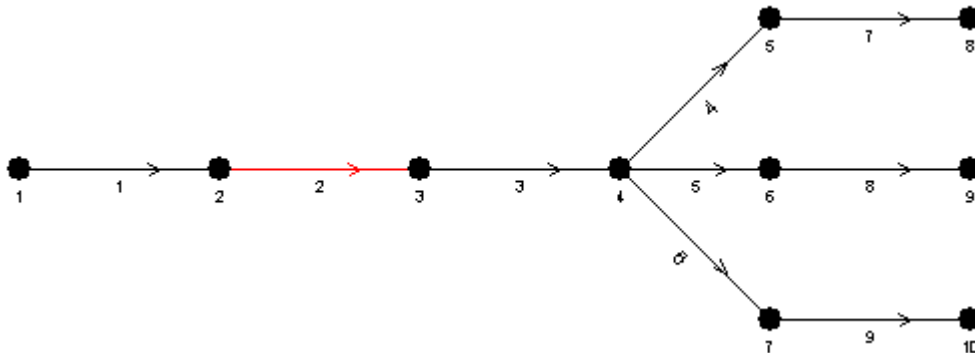
8.11 Exporting the Schematic

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](#).

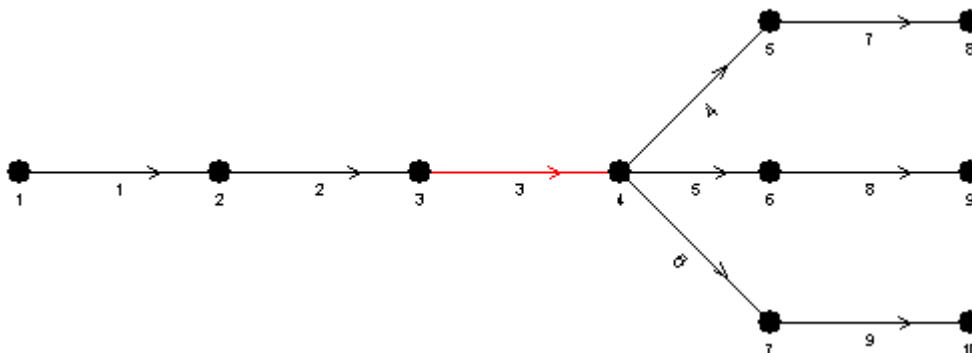
8.12 Moving around the network

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).

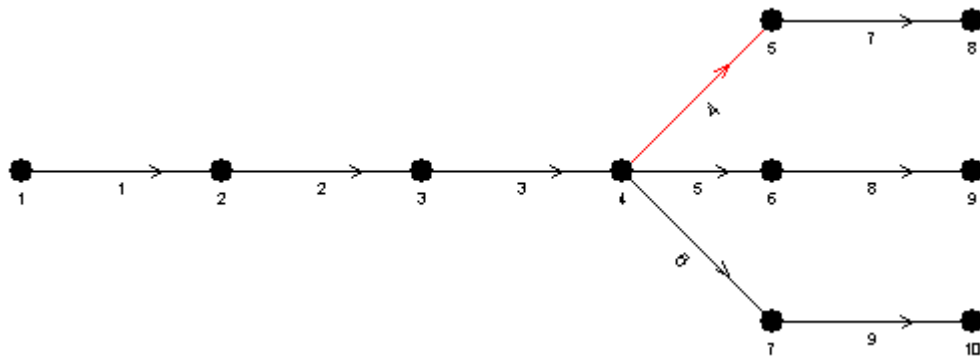
There is another way to move around the network, highlighting components in turn, using the four cursor keys on the keyboard. This is best illustrated by the following example. With pipe 2 the currently selected component and all component directions being from left to right, selecting the right cursor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



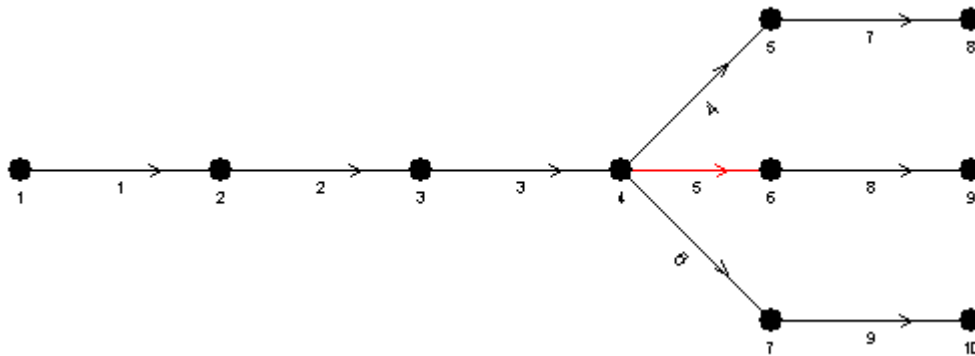
Now consider what happens when we are on pipe 3:



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.

8.13 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

9

9 The Tabular View

9.1 Tabular Grid

The Tabular View window and the Properties Window are used to enter and edit the attributes of components displayed on the schematic, and to display some summary results following a calculation. The Tabular View window cannot be used for adding new components or deleting existing components. These operations must be performed via the schematic.

Label	Input node	Output node	Diameter	Length	Net height change	C-factor	Type	Wavespeed	Results selected?	Section Type
			mm	m	m			m/sec		
1	A	4	250	100	100	100	Copper/Nickel (90/10) 14 Bar g:Asbestos	1122.39	YES	Constant Gradient
2	B	5	100	10	0	100	Copper/Nickel (90/10) 14 Bar g:Asbestos	1239.38	YES	Constant Gradient
3	C	5	100	10	0	100	Copper/Nickel (90/10) 14 Bar g:Asbestos	1239.38	YES	Constant Gradient

Pages available in the Data window

A number of pages are provided, each page consisting of one or more grids:-

Data

The data page displays editable component data for a specific component type, selectable from the drop-down list at the top of the window:

height change	Roughness	Additional k-factor
	mm	
0.0457	0	
0.0457	0	
0.0457	0	

Results

The results page displays maximum and minimum pressure results for components, the component type to be displayed is selected as in the Data page.

Graphs

Displays ad-hoc graphs produced in the user interface, mainly the [elevation profile and pressure envelope](#).

Result graphs

This page is used for defining the [output graphs](#) to be produced by the calculation.

Tables

This page is used for defining the [output tables](#) to be produced by the calculation.

Forces

This page is used in [defining the forces](#) to be calculated

Tabular results

On completion of a calculation some of the results appearing in the printed browser report may be viewed here. Currently [Forces Extrema](#) and Pipe type extrema are available, selectable via the combo at the head of the page..

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.

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

Printing

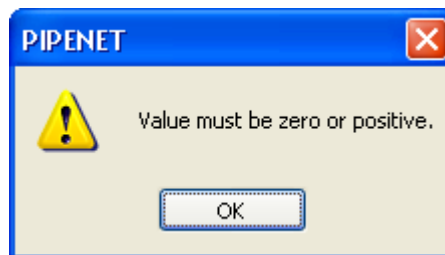
The current grid can be printed by selecting the Print button.

9.2 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.

9.3 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.

Status Checking

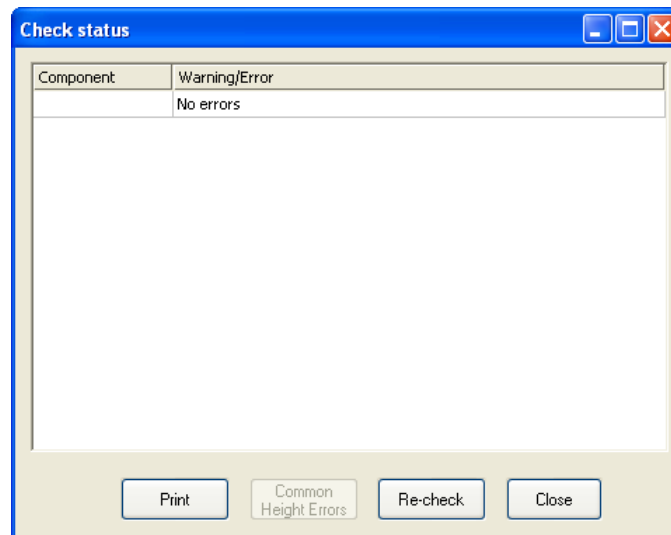
Part

10

10 Status Checking

10.1 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.

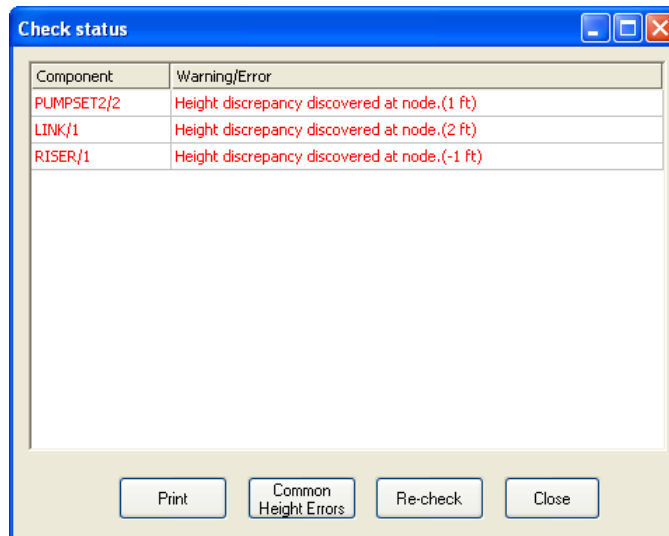
Warnings are shown in back text, and indicate possible problems with the network. Errors are shown in red text and indicate problems that may prevent a successful calculation.

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.

10.2 Height Checking

Height checking will be performed, 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

11

11 Colour Schemes

11.1 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.
- GREEN - item has graphical results

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.

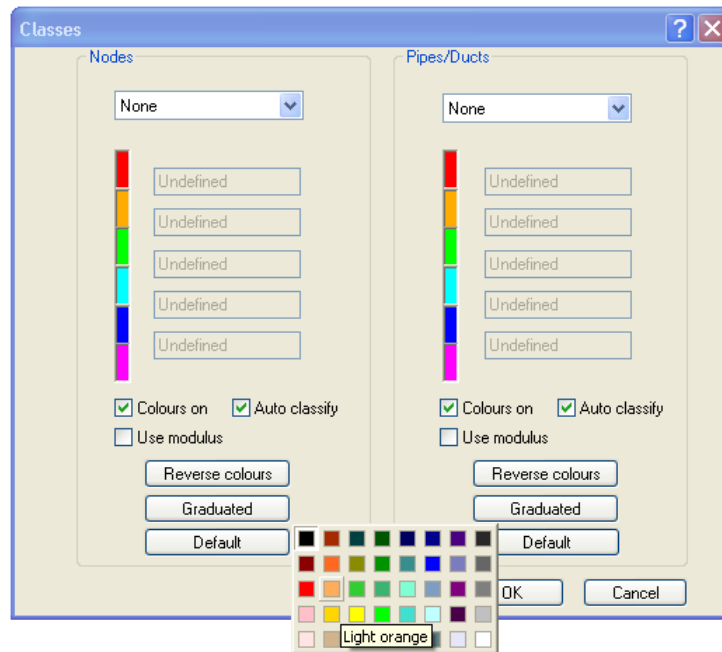
Simple Colour schemes

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.

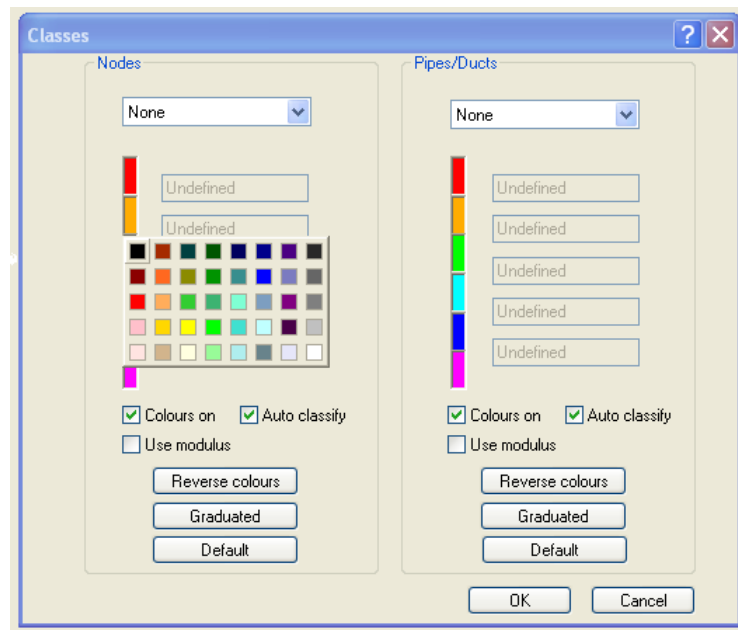
Immediately below each column of five edit boxes are two check boxes:

- **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.

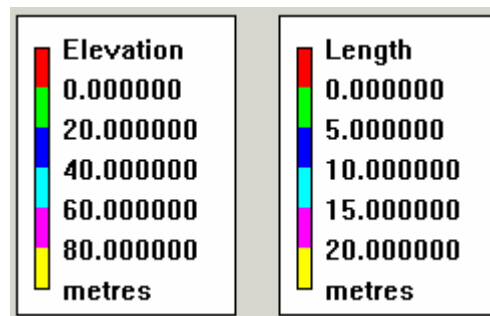
Finally, it is possible to change the default colours by left-clicking on one of the coloured boxes, when a colour selection pop-up appears. For example, in the following image the

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

This feature is provided to retain compatibility with the Standard and Spray modules, however, since the Transient calculator does not return simple single value results this feature is of limited use.

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:

The screenshot shows a dialog box titled "New Colour Schemes" with a close button (X) in the top right corner. The dialog is divided into several sections:

- Pipes/Ducts**: A section header.
- Name**: A dropdown menu containing "High velocity in big pipe".
- Condition 1**: A dropdown menu containing "Pipe bore", a relational operator dropdown containing ">", a text input field containing "100", a unit dropdown containing "mm", and a checkbox labeled "Use modulus".
- Condition 2**: A dropdown menu containing "Pipe velocity", a relational operator dropdown containing ">", a text input field containing "10", a unit dropdown containing "m/sec", and a checkbox labeled "Use modulus".
- Condition 3**: A dropdown menu containing "None", a relational operator dropdown, a text input field, and a checkbox labeled "Use modulus".
- Logical Operators**: Radio buttons for "And" (selected) and "Or".
- Colour**: A dropdown menu containing "Red".
- Buttons**: "New", "Delete", "OK", "Apply", and "Cancel".

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.

11.2 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**.

11.3 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 Pressure Envelope

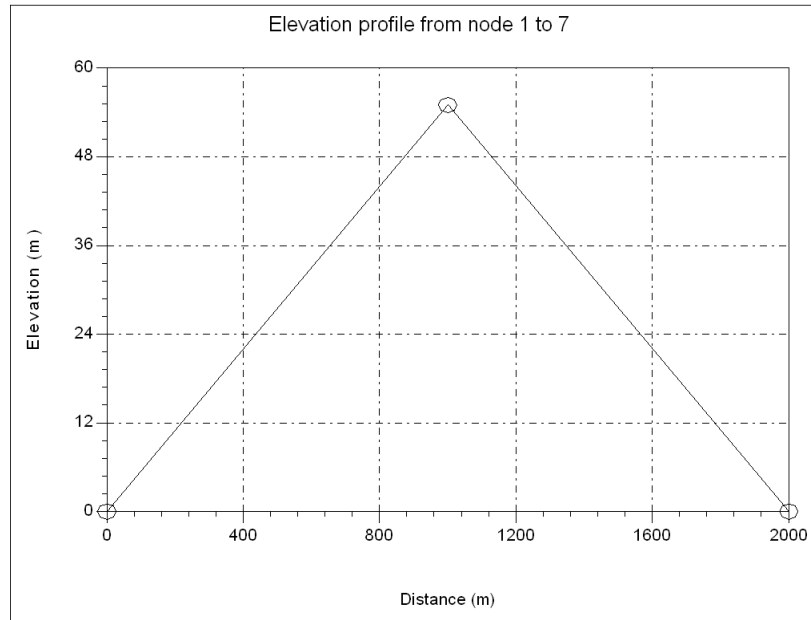
Part

12

12 Elevation Profile and Pressure Envelope

12.1 Elevation Profile Window

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

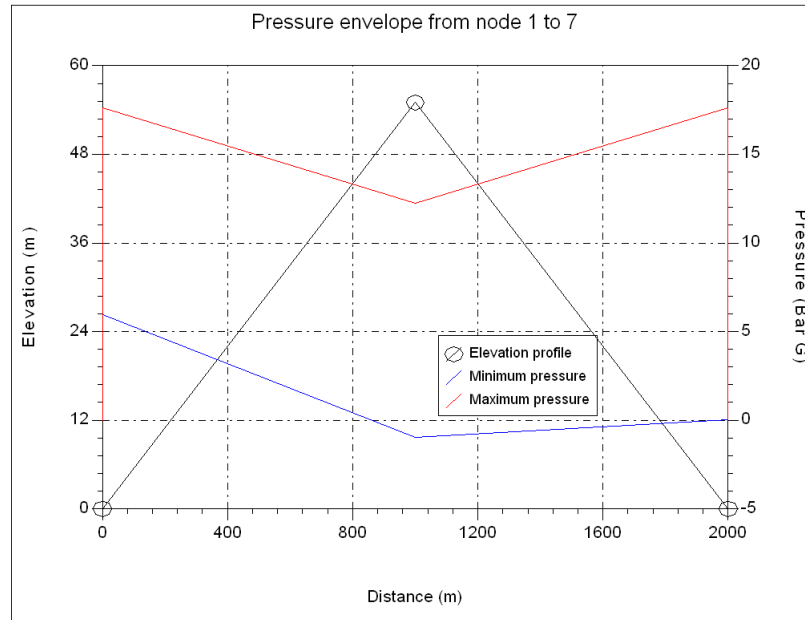
Right-clicking with the mouse displays a popup menu with the following options:

- 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

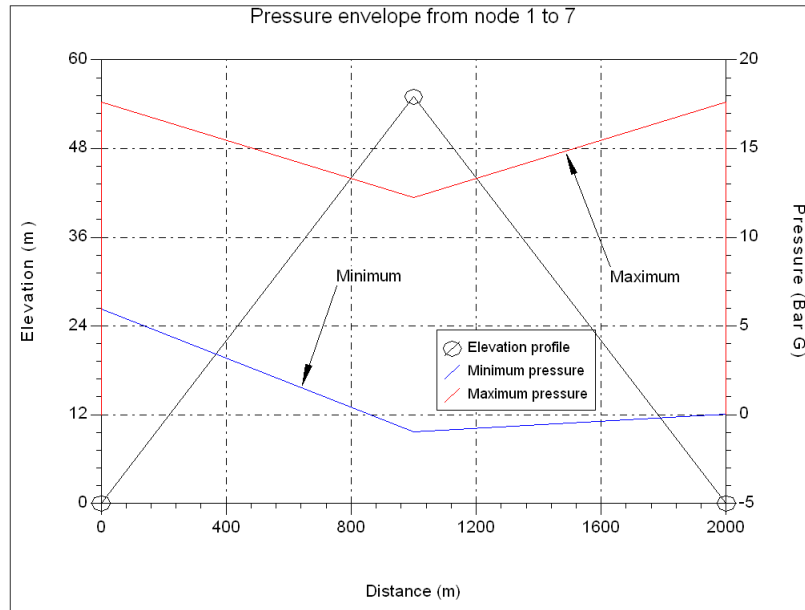
possible to edit the title, labels, styles and the axes. These properties can be saved as a template.

Pressure Envelope

If a Transient calculation has been performed the maximum and minimum pressures will be displayed for each node, as show in the following image:



The left-hand axis is the node elevation as in the previous figure and the right-hand axis is the pressure. Note the Title for the graph now indicates that this a pressure envelope. Since there is more than plot a legend is provided, this legend may be moved by selecting the legend with the right-hand mouse button and selecting the option Move from the pop-up menu. It is also possible via the pop-up menu to delete the legend or edit the legend. The following image is the same as the above but includes two arrowed text items:



Add multiple pipes

Part

13

13 Add multiple pipes

13.1 Add Multiple Pipes

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 <input type="text" value="Pipes"/>		
Tag	<NO TAG>		No.	Distance (m)	Elevation (m)
Diameter	Unset	m	1	1	0
Length	1	m	2	2	0
Elevation	0	m			
Roughness	Unset	m			
Additional K-factor	0				
Status	Normal				
Type	No pipe type				

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 be 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
5	5	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, and the units must agree with those in the column headings. [Undo/redo](#) is available with this facility.

Part

14

14 Libraries

14.1 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,:

- pipe schedules
- fittings
- pumps
- valves
- nozzles
- linings
- general pressure losses

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.

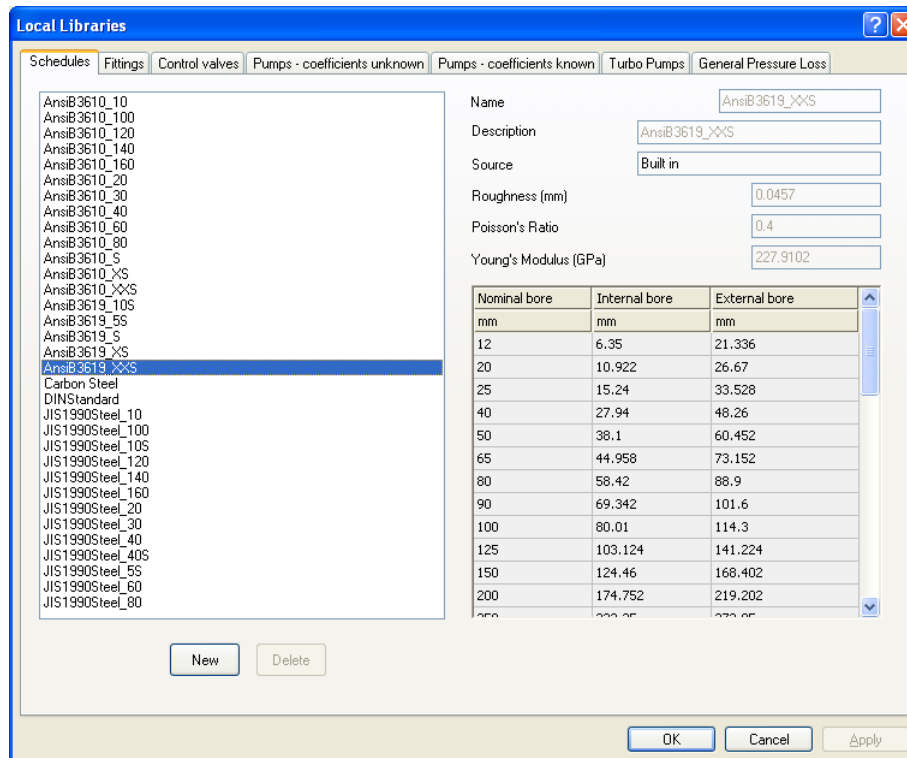
1. Create a new project.
2. Go to the File menu and select the option Import library.
3. Use the file dialog to select the library to open, that is a .PDF, .PMP, etc. file.
4. Open the file - the file is imported into the local user library.
5. Repeat steps 2 and 3 for each old style library to be imported.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

14.2 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 Standard 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 are three fields containing the roughness, Poisson's ratio and Young's Modulus. Below these is 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. Right-click within the sizes grid to display an option to copy the grid contents to the clipboard.

Nominal diameters are greyed-out indicating that their value is fixed, however by scrolling

down to the end of the grid 10 user-defined sizes are revealed, for these entries both the nominal and internal bores can be edited.

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

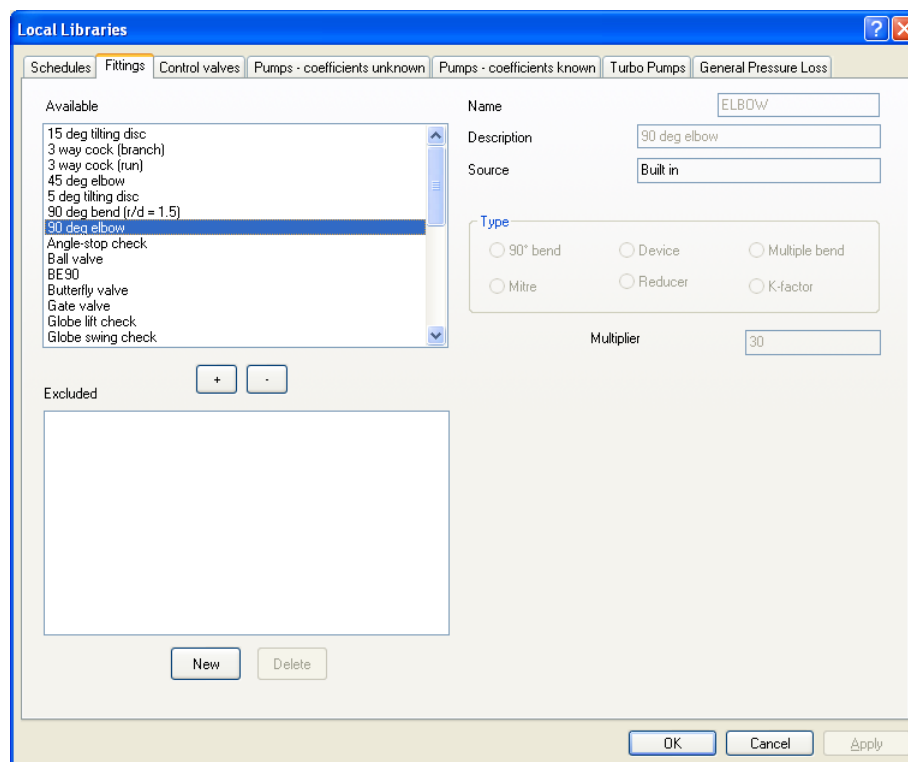
Select the schedule to be edited from the left-hand window, enter the desired changes and then select the **Apply** button to accept the changes.

Deleting an existing schedule

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

14.3 Fittings

Fittings are added to the library via the following dialog:



Making fittings available or unavailable

In the top left-hand window is a list of the available fittings and in the bottom left-hand corner a list of those fittings which are to be excluded from selection in the [Fittings window](#). To make a fitting in the top window unavailable for selection highlight the fitting

in the upper window and select the - button. To make an unavailable fitting in the bottom window available for selection, highlight the fitting in the lower window and select the + button.

Adding a fitting

To add a new fitting proceed as follows:

1. Select the **New** button
2. Enter the name of the fitting (see below)
3. Enter an option long description for the fitting
4. Select the type of fitting from those listed
5. Supply the Parameters (dependent on fitting type) in the one, two or three edit boxes provided. Note the number of edit boxes provided and the label for each box will change according to the fitting type selected
6. Select the **Apply** button to apply add the fitting.

Note that the rules for defining fitting names are as follows:-

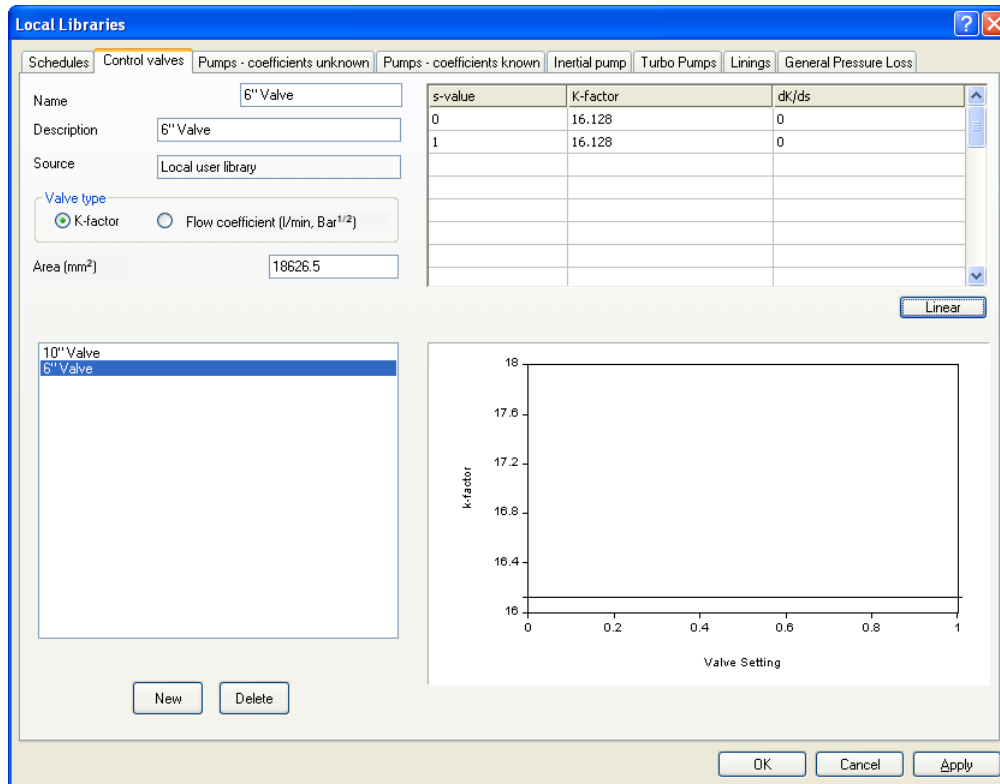
1. For **K-factor** type fittings an alphanumeric name may be specified
2. For all other fitting types the first two characters of the name are defined by the fitting type, for example "BE" for a bend and "DE" for a device, and so on. The Rest of the name is supplied by the user and must be a positive number in the range 1 to 999.

Deleting a fitting

Highlight the desired fitting in the top left-hand window and select the **Delete** button; button is disabled for built-in fittings.

14.4 Control Valves

This page is used for displaying and editing library control valves.



To the bottom left is a list of available control valves, both built-in and user-defined, selecting any item in this list displays the properties of the control valve. At the top left of the dialog there are the following attributes:

1. the name of the control valve 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 control valve, which may be one of the following:
 - Built-in control valve provided with the Standard module. Currently there are no built-in control valves
 - 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
4. Valve type K-factor or Flow coefficient.
5. Area of valve

At top right there is a grid into which can be entered up to eight (minimum of two) defining points. Below the data grid is a plot of the entered points.

Selecting the **Linear** button (immediately below the grid) will fill in the values for dK/ds or dCv/ds using a linear gradient between each two successive points.

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.

Adding a new control valve

To add a new control valve 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 control valve name. When all of the data has been entered click the **Apply** button to accept the new control valve or the **Cancel** button to abort.

Editing an existing control valve

Select the control valve 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 control valve

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

14.5 Pumps - Coefficients Unknown

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

Flow rate	Head (input)	Head (curve)	Head (calculation)
m ³ /h	Bar fluid (reference)	Bar fluid (reference)	Bar fluid (current)
0	7.06	7.06	7.831
1000	6.266	6.266	6.95
2000	5.472	5.472	6.069
3000	4.943	4.943	5.482
4000	4.502	4.502	4.993
5000	3.266	3.266	3.622

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. If units of pressure are being used (that is Pascals, bar, psi or kgf/cm²) then provide a reference fluid density, water with a density of 998.2345 kg/m³ being the default
6. Provide a minimum and maximum flow rate
7. Provide the two degeneration factors (defaults 0.0) for a Quadratic curve that modify the curve slope outside the working range
8. Provide a minimum of three points for the curve in the bottom left-hand corner window
9. Select **Apply** to add the pump to the library
10. 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.

For more information on curve types and degeneration factor see the [modelling section for simple pumps](#).

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.

14.6 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 'Name' field contains 'aaa'. The 'Description' field contains 'aaa'. The 'Source' is 'Local user library'. The 'Curve type' is 'Quadratic'. The 'Ref. Fluid Density (kg/m³)' is '998.2343'. The 'Flowrate' is 'm³/hour' and 'Head' is 'bar'. The 'Working range' has 'Min' flowrate of 5000 and 'Max' flowrate of 10000, with 'Min' head of 5 and 'Max' head of 7.5. The 'Curve coefficients (reference fluid)' are: Constant 5, Linear 0.001, Quadratic -1e-007, and Cubic 0. The table below shows the resulting pump curve data:

Flow rate m³/h	Head (input) Bar fluid (reference)	Head (curve) Bar fluid (reference)	Head (calculation) Bar fluid (current)
5000		7.5	7.5
5250		7.494	7.494
5500		7.475	7.475
5750		7.444	7.444
6000		7.4	7.4
6250		7.344	7.344
6500		7.275	7.275
6750		7.194	7.194
7000		7.1	7.1

The graph shows Pressure (Bar fluid (reference)) on the y-axis (ranging from 4.8 to 7.8) and Flow Rate (m³/h) on the x-axis (ranging from 4800 to 10800). The curve is a downward-sloping parabola.

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. If units of pressure are being used (that is Pascals, bar, psi or kgf/cm²) then provide a reference fluid density, water with a density of 998.2345 kg/m³ being the default
5. Provide a minimum and maximum flow rate
6. Provide the coefficients
7. Select **Apply** to add the pump to the library
8. 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.

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.

14.7 Pumps Turbo

The Turbo Pump is defined by its rated quantities (values of pressure, flow rate, speed and torque at maximum operational speed), the total moment of inertia of the pump and by a pair of Suter characteristic curves for pressure head and torque.

The screenshot shows the 'Local Libraries' dialog box with the 'Turbo Pumps' tab selected. The 'Rated Quantities' section contains the following values:

- Pressure (m fluid): 48.00141
- Flowrate (m³/h): 7499.988
- Speed (revs/sec): 12.25
- Torque (Nm): 14848
- Moment of Inertia (kg m²): 331.7

The 'Name' field is '910P01', 'Description' is '910P01', and 'Source' is 'Local user library'. The 'Suter Curve Data' dropdown is set to 'Mixed Flow (NS=147)'. Below these are two tables for Suter curve data:

x	Head-Flowrate WH(x)
0	-0.69
0.0714	-0.599
0.1428	-0.512
0.2142	-0.418
0.2856	-0.304
0.357	-0.181
0.4284	-0.078
0.4998	-0.011
0.5712	0.032
0.6426	0.074
0.714	0.13
0.7854	0.19

x	Torque-Flowrate WB(x)
0	-1.42
0.0714	-1.328
0.1428	-1.211
0.2142	-1.056
0.2856	-0.87
0.357	-0.677
0.4284	-0.573
0.4998	-0.518
0.5712	-0.38
0.6426	-0.232
0.714	-0.16
0.7854	0

Buttons for 'New', 'Delete', 'OK', 'Cancel', and 'Apply' are visible at the bottom of the dialog.

Three built-in Suter curves are provided : Radial, Mixed Flow and Axial having specific speeds 25, 147 and 261 (SI units rpm, m³/s, metres) respectively. Alternatively the user can define the Suter curves by selecting user-defined in the Suter curve data field and entering a number of points. For details of how to transform turbo pump data for pressure head and torque into Suter characteristic curve data x , $WH(x)$ and $WB(x)$ see the [modelling details](#). The bottom left-hand window is for entry of the pump head - flow rate curve data, and the bottom right-hand window for the pump torque - flow rate data.

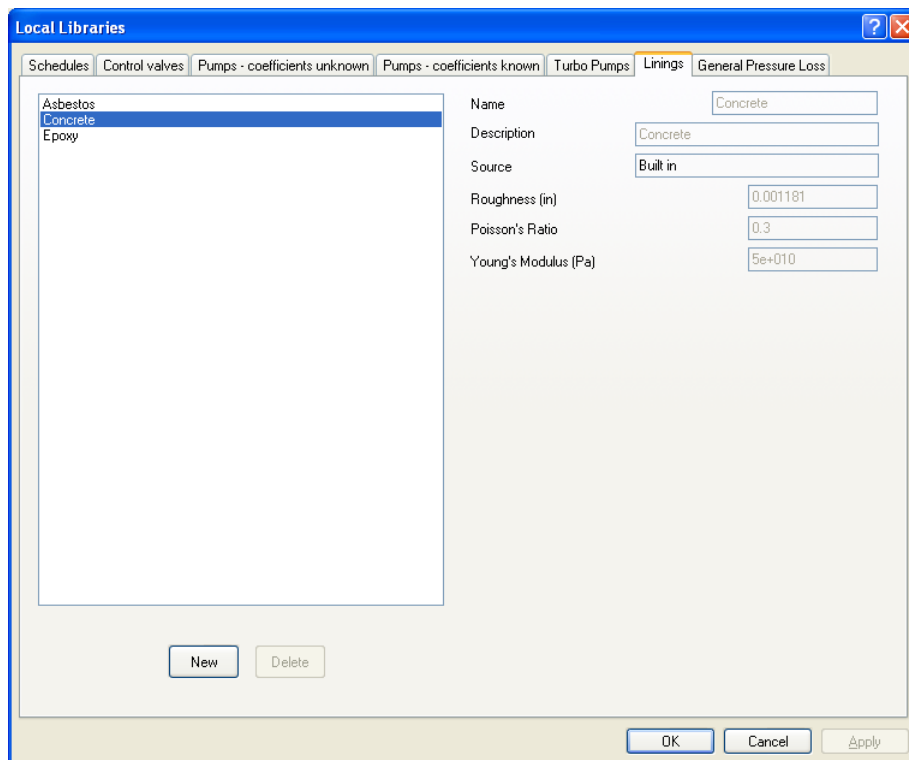
For the moment of inertia, enter the total rotary moment of inertia of the pump impeller port and the pump motor port.

Right-click within the head-flow rate grid area to display an option to copy the head grid contents to the clipboard; right-click within the torque-flow rate grid area to display an option to copy the torque grid contents to the clipboard

14.8 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:

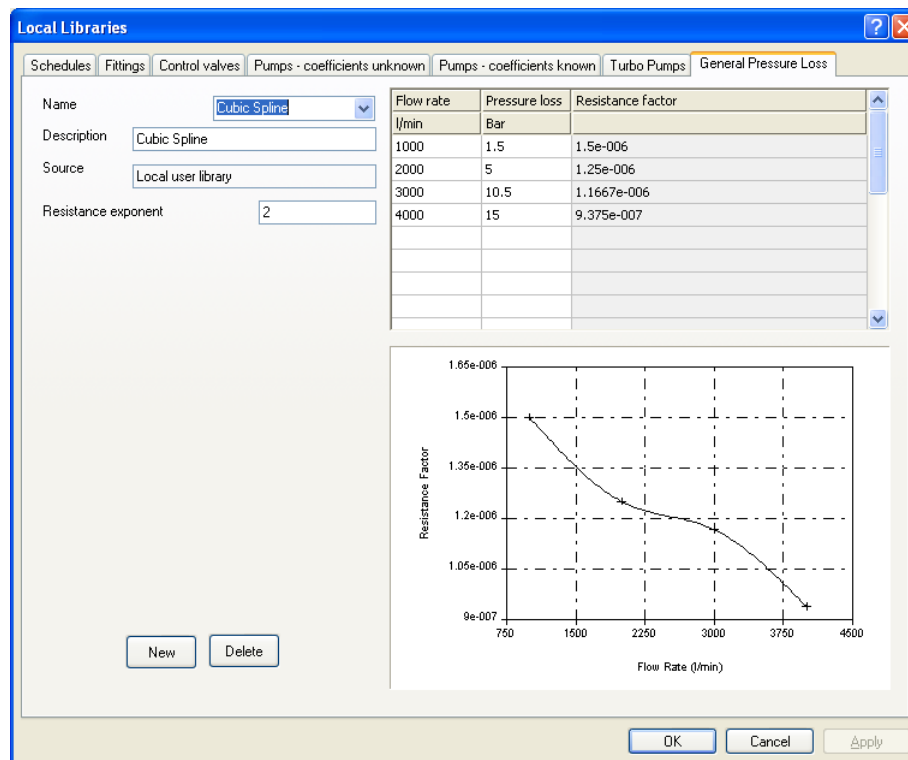
1. Select the **New** button
2. Provide a unique name for the lining and an optional description
3. Enter the values for roughness, Poisson's ratio and Young's modulus
4. Select the **Apply** button to add the lining to the library.



Note that linings are only applicable when the [Spray option](#) has been selected.

14.9 General Pressure Losses

This page is for the display and editing of general pressure losses:



Creating a new Pressure Loss

1. select the **New** button and provide a name and optional description
2. provide a resistance factor (see the [modelling section](#))
3. in the grid on the left-hand side of the dialog provide a series of data points - if two points are provided a linear curve will be used; three points a quadratic will be used to fit the points, and if more than three points are provided a cubic spline will be used.
4. select **Apply** to add the pump to the library
5. the resistance factor coefficients are calculated and displayed, along with the pump curve.

The flow rate range in the graph will be 0 to 120% of the maximum flow.

Editing an existing pressure loss

1. Select the pump from the drop-down list presented in the pressure loss name box (top right-hand corner)
2. make any changes required to the pressure loss parameters;
3. select **Apply** to commit the changes.

Deleting a pressure loss

1. Select the pressure loss from the drop-down list presented in the pump name box;

2. Select the **Delete** button.

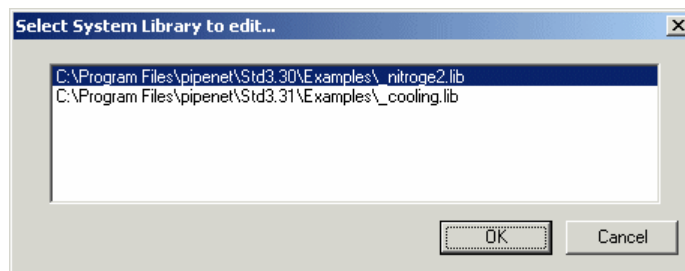
14.10 Editing System Libraries

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

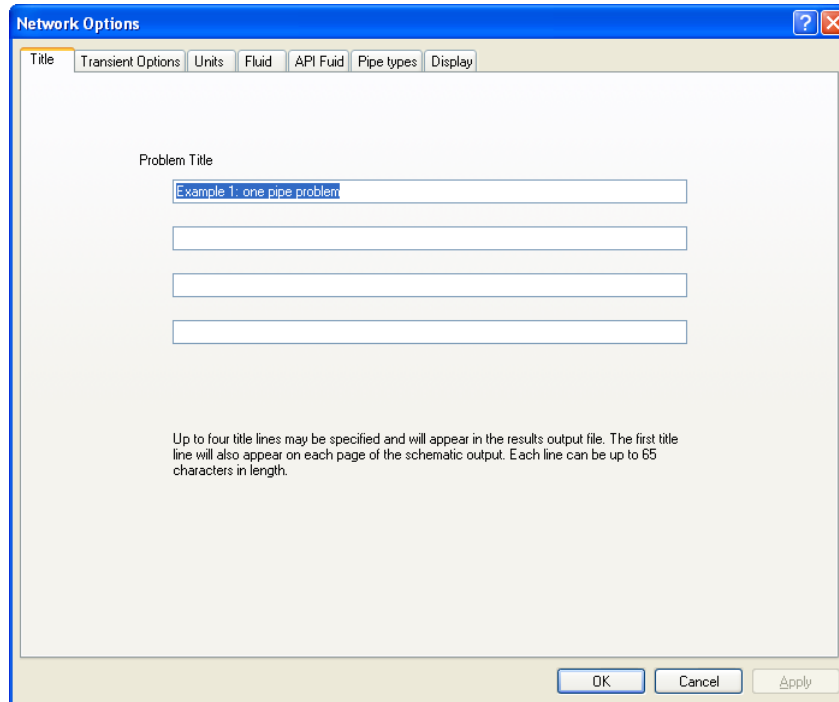
Part

15

15 Specifying Options

15.1 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.



15.2 Transient options

The options page provides control over a number of modelling and calculation options for the Transient module. The dialog is the same for both the Standard and Spray sub-types, except for the Pressure model and Design rules sections. The dialog for the Standard sub-type is:-

and for the Spray sub-type it is:-

15.3 Units

A wide selection of unit systems are provided, including:

- SI
- Metric

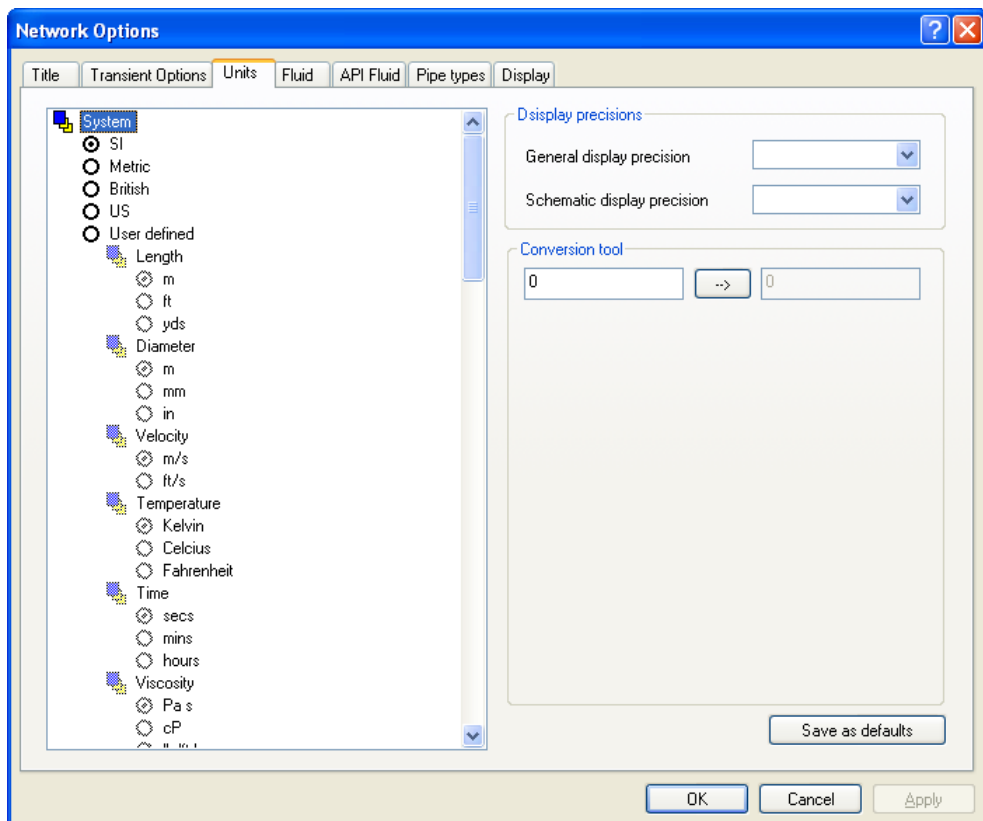
- US
- Imperial
- User Defined

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.

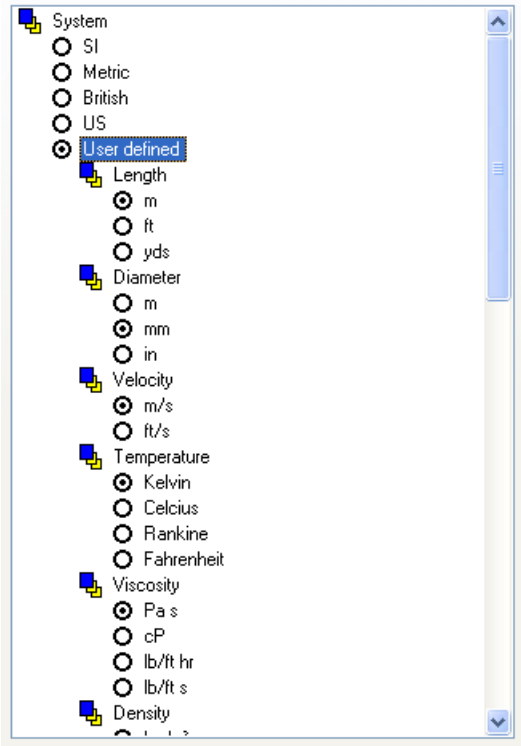
The Units dialog

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:

The image shows two sections of a software dialog box. The top section, titled 'Display precisions - Length', contains two dropdown menus. The first is labeled 'General display precision' and the second is 'Schematic display precision'. Both dropdowns are currently set to '0.123'. The bottom section, titled 'Conversion tool - Length', features two input fields, both containing the number '0', separated by a right-pointing arrow button. Below these fields are two columns of radio buttons for unit selection. The left column has radio buttons for 'm', 'ft', and 'yds', with 'm' selected. The right column also has radio buttons for 'm', 'ft', and 'yds', with 'm' selected.

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).

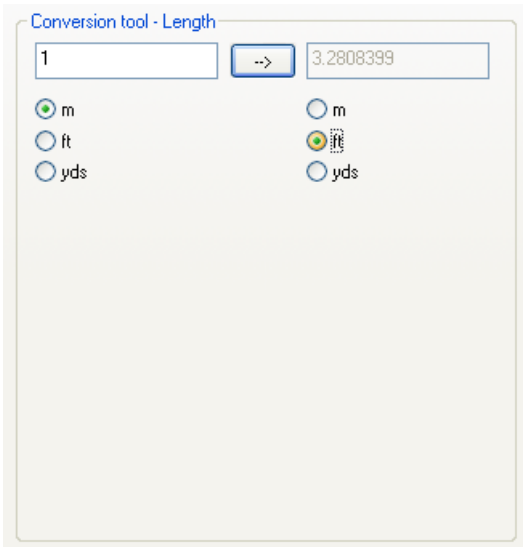
Selecting the arrow to the right of each field produces a drop-down showing that the number of decimal places can be selected to between zero and eight.:

This image is similar to the previous one but shows the dropdown menu for the 'General display precision' field open. The menu lists various precision options: 0, 0.1, 0.12, 0.123 (which is highlighted in blue), 0.1234, 0.12345, 0.123456, 0.1234567, and 0.12345678. The 'Conversion tool' section remains visible below.

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.



15.4 Fluid

Via this dialog the user can specify the properties of the fluid in use, the properties being the fluid density, fluid viscosity, temperature, bulk modulus and the vapour pressure. An alternative method of specifying a fluid is provided via the American Petroleum Institute ([API dialog](#)) described in the following section.

The screenshot shows the 'Network Options' dialog box with the 'Fluid' tab selected. The 'Fluid name' dropdown is set to '<User-defined Fluid>'. The 'Fluid state' dropdown is set to 'Unsatrated state'. The following parameters are entered in their respective text boxes:

Parameter	Value
Pressure (Bar G)	
Temperature (°C)	
Density (kg/m ³)	998.2
Viscosity (Pa s)	0.001
Temperature (°C)	20
Bulk modulus (GPa)	2.2
Vapour pressure (Bar G)	-0.977

At the bottom of the dialog, there are buttons for 'Use API fluid', 'Calculate', 'Refresh', 'OK', 'Cancel', and 'Apply'. A text box labeled 'Messages' is also present.

The five fluid parameters are either entered directly for a User-defined fluid, selected from Fluid name combo box), or they can be calculated for water and steam by selecting Water-Steam as the fluid name: and selecting the fluid state (second combo box) as one of:

- Unsaturated; a pressure and temperature must be provided
- Saturated liquid; a pressure or a temperature must be provided
- Saturated vapour; a pressure **or** a temperature must be provided

The screenshot shows the 'Network Options' dialog box with the 'Fluid' tab selected. The 'API Fluid' sub-tab is active. The 'Fluid name' is set to 'Water-Steam' and the 'Fluid state' is 'Unsaturated state'. The following parameters are entered in the input fields:

Parameter	Value
Pressure (Bar G)	0
Temperature (°C)	0
Density (kg/m³)	998.2
Viscosity (Pa s)	0.001
Temperature (°C)	20
Bulk modulus (GPa)	2.2
Vapour pressure (Bar G)	-0.977

Below the input fields is a 'Messages' area. At the bottom of the dialog are buttons for 'Use API fluid', 'Calculate', 'Refresh', 'OK', 'Cancel', and 'Apply'. A text box on the right side of the dialog provides instructions: 'For a named fluid the pressure and/or temperature (depending on the fluid state) are entered. Selecting the Calculate button calculates the five fluid parameters. For user defined fluids, the five values are entered directly.'

Selecting the **Calculate** button will calculate the five fluid parameters based on the settings of the Pressure and/or the temperature values, and place display them in the dialog. If the calculation of the fluid parameters succeeds the fluid state will be displayed in the Messages area, otherwise the Messages area will contain an indication of why the calculation could not be completed.

By default the fluid is assumed to be defined by the settings of this dialog, however, if it is desired to use the properties of the fluid defined by the [API dialog](#) then select the **Use API fluid** button - the text of the button changes to **Use direct fluid**. Selecting the button again will revert to using the direct fluid specification provided by this dialog.

15.5 API Fluid

This dialog can also be used to specify the fluid in use in accordance with the American Petroleum Institute (API) criteria. Based on the data entered parameters (left-hand side of the dialog) and on selecting the Calculate button the properties of the fluid are calculated and displayed on the right-hand side of the dialog. Currently only the five parameters, fluid density, fluid viscosity, temperature, bulk modulus and the vapour pressure are used in the calculation.

Network Options

Title Transient Options Units Fluid **API Fluid** Pipe types Display

Basic parameters

Input temperature (°C) Unset

Input pressure (Bar A) Unset

API gravity Unset

Specific gravity Unset

Watson characterisation Unset

Mean average boiling point (°C) Unset

Viscosity correction

Not specified

Ref. viscosity (Pa s) Unset

Ref. temperature (°C) Unset

Critical viscosity (Pa s) Unset

Molecular weight

Not specified

Molecular weight Unset

Reference density

Not specified

Ref. density (kg/m³) Unset

Ref. temperature (°C) Unset

Results

Parameter	Units	Value
Viscosity	Pa s	
Density	kg/m ³	
Secant bulk modulus	Pa	
Vapour pressure	Bar A	
The following parameters are not used in the calculation		
Heat capacity	J/kg K	
Heat capacity - alt. solution	J/kg K	
Atmospheric heat capacity	J/kg K	
Atmospheric heat capacity - alt. solution	J/kg K	
Thermal conductivity	W/(m.K)	
Atmospheric thermal conductivity	W/(m.K)	
Surface tension	N/m	
Atmospheric density	kg/m ³	
Compressibility	1/Pa	
Molecular weight		

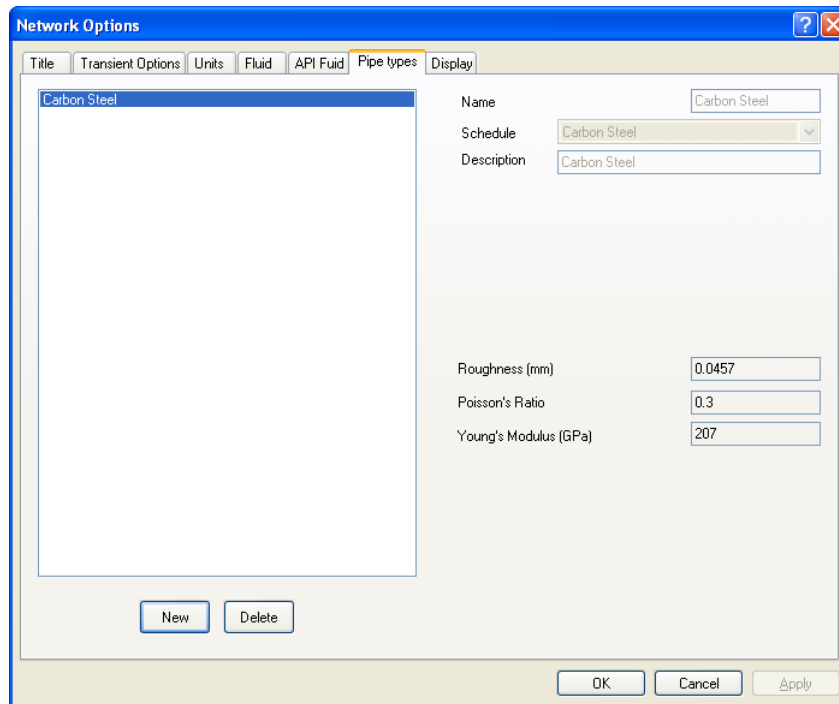
Use API fluid Calculate

OK Cancel Apply

By default the fluid is assumed to be directly specified using the [Fluid dialog](#) described in the previous section, however, if it is desired to use the properties of the fluid defined by this dialog then select the **Use API fluid** button - the text of the button changes to **Use direct fluid**. Selecting the button again will revert to using the direct fluid specification provided by the Fluid dialog. For further information on API fluids refer to the [Modelling chapter](#).

15.6 Pipe types

This dialog is used to provide a pipe type:



In the Transient module all pipes in a network can be entered directly (that is not using pipe types) or all pipes in a network must be of a specified type. It is not possible to have some pipes in a network using pipe types and some pipes not. If after entering some pipes you define a first pipe type then a warning will be issued giving the user the opportunity to ensure that all pipes have a pipe type.

A pipe type will be associated with a pipe schedule, and this [schedule](#) must already exist before the pipe type can be created.

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 - name and description fields are completed;
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 add the pipe type to the library.

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 use.

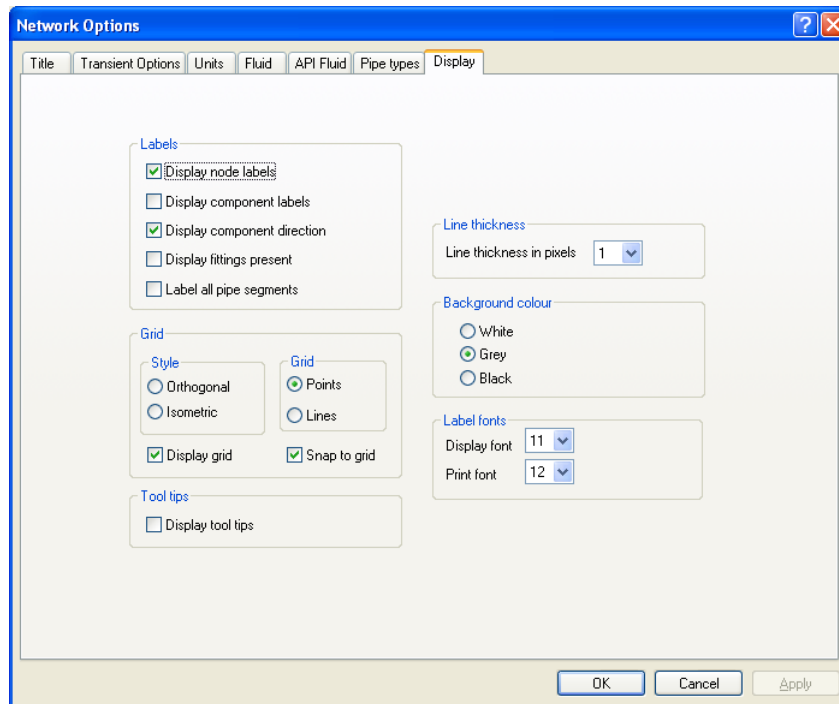
Pipe type extrema

On completion of a calculation the maximum and minimum pressures found for each pipe type can be displayed in the Data window by selecting the **Tabular results** page and then the **Pipe Type Extrema** option from the combo at the head of the page.

Pipe Types Extrema									
Pipe type	Max. pressure	Time	Pipe id	Position	Min. pressure	Time	Pipe id	Position	
	Bar G	sec		m	Bar G	sec		m	
Carbon Steel	27.117011	0.13	2	Outlet	24.585209	0.34	BEND	Outlet	

15.7 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.



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

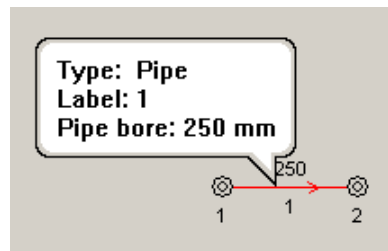
the horizontal and a third at 150 degrees to the horizontal).

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 pipe length..

Performing a Calculation

Part

16

16 Performing a Calculation

16.1 Transient calculations

Transient generally operates in the same manner as the Standard and Spray modules. However, there are a number of additions, primarily to do with the selection of tables and graphs and with the display of graphical results. Running a transient calculation does not produce a simple set of results that can be displayed in a grid. Instead the user selects the results they want and on completion of the calculation the requested results are stored in a results file. Results can be selected for display using a separate Graph Viewer program.

16.2 Selecting graphs for components

Selecting results

Components for which results (graphs) have been selected are displayed in green using the default colouring scheme. To request results for a particular component, right-click on the component and select the Select Results option. This is the same method that was used in previous versions of PIPENET Transient. For non-pipe components this will display a list of available variables - select the single result required, for example **Pressure**, or **All** to accumulate results for all attributes.

Removing results

To remove results for a component right-click on the component and select the **Remove Results** option.

Selecting and removing results in the Tabular Grid

A summary of which results have been selected can be viewed on the [Tabular grid](#) - select the **Graphs** tab when all selected results are displayed, each result being in one of two categories: variable against time (all components) or variable against distance (pipes only).

Component type	Component label	Variable
Variable against time		
Pipe	1	All variables
Pipe	2	All variables
Transient force	BEND	All variables
...		
Variable against distance		
Pipe	1	All variables
Pipe	2	All variables
...		

The first column provides the component type, the second the component label and the third the selected variable, only the last of these is directly editable. To add a new result in one of the two categories:

1. Select line containing ... in the first column the first blank line.
2. Select the component type from the drop-down list in the second column of the newly created row.
3. Select the component label in the third column.
4. Select the variable(s) required.
5. Finally select the symbol in the first column to commit the change.

A result may be deleted by right-clicking in the first column of a row and selecting the **Delete graph** option.

16.3 Defining Tables

Tables are defined using the **Tables** tab in the Tabular grid. When first selected the display might typically be as follows:

Column	Component type	Component label	Variable	Value
+	1	Pipe	1	
...	5			

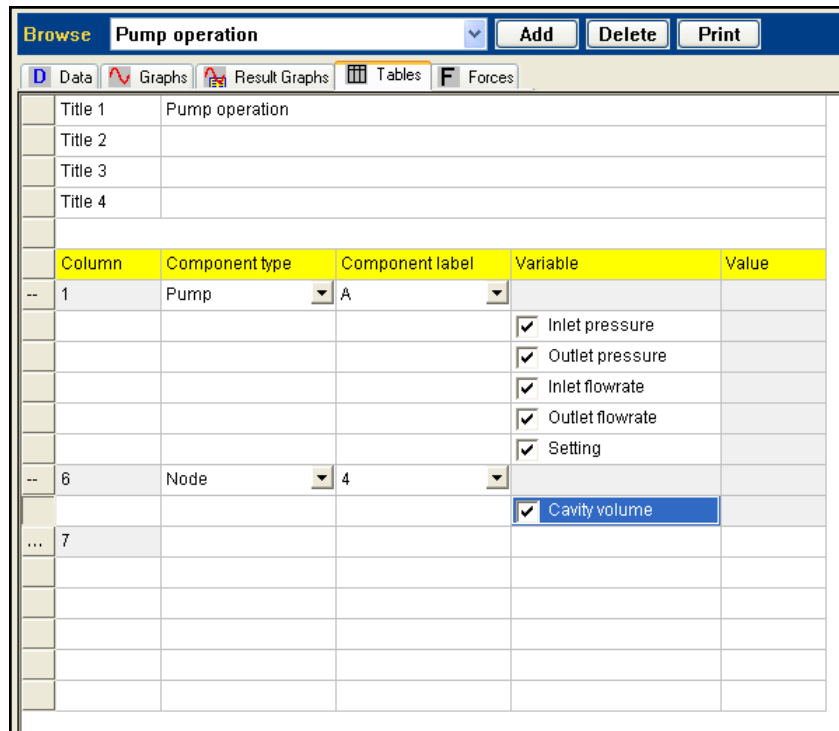
The table is divided into two parts; at the top are four title lines and below this the definition of the table. The drop-down box at the top indicates which table is currently displayed, in this case "Pump operation".

The table definition as initially displayed displays a line for each component, identified by its type and label together with a start column. The last row (with the symbol ... in the first column) indicates the next free column available (there are a maximum of eight columns allowed in a table). In the above example the pump with the label uses the first five columns of the table. To view which attributes have been selected for the table, select the + symbol in column one and the display expands to include a list of the variables:

Browse Pipe 1 - Reservoir to Bend					Add	Delete	Print	
D Data					Graphs	Result Graphs	Tables	F Forces
Title 1	Pipe 1 - Reservoir to Bend							
Title 2								
Title 3								
Title 4								
Column	Component type	Component label	Variable	Value				
-- 1	Pipe	1	<input checked="" type="checkbox"/> Inlet pressure <input checked="" type="checkbox"/> Outlet pressure <input checked="" type="checkbox"/> Inlet flowrate <input checked="" type="checkbox"/> Outlet flowrate <input type="checkbox"/> Pressure at...(m) <input type="checkbox"/> Flowrate at...(m) <input type="checkbox"/> Cavity vol at...(m) <input type="checkbox"/> Pressure at n point... <input type="checkbox"/> Flowrate at n points... <input type="checkbox"/> Cavity vol at n points... <input type="checkbox"/> Length of cavity volu... <input type="checkbox"/> Length of liquid colu... <input type="checkbox"/> Volume of vapour c... <input type="checkbox"/> Vapour cavity volum... <input type="checkbox"/> Liquid holdup					
... 5								

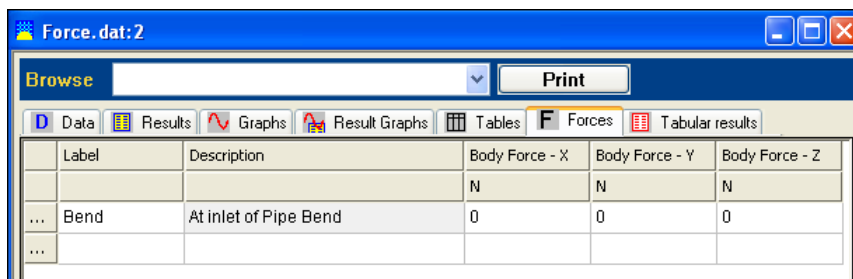
The + symbol now changes to a - symbol which if selected will once again hide the details of the attributes selected. De-selecting or selecting one or more of the available attributes will immediately update the number of columns used.

To add extra columns to the grid click on the last row (the one containing the ... symbol in column 1) and select from the list of available component types and component labels displayed. Finally select the attributes to be displayed:



16.4 Defining Forces

Forces are defined using the **Forces** tab in the Tabular grid. When first selected the display might typically be as follows:



This view shows the list of defined forces and a description, clicking on the ... symbol in the first column of one of the rows displays a dialog where full details of the force can be viewed and edited:

Force

Force label: BEND

Components in control volume

Pipe	1
Pipe	2
Operating valve	shutoff

Force type

Simple

Complex

Body force - X (N): 0

Body force - Y (N): 0

Body force - Z (N): 0

Show Interior Components

Control Area

Type	Label	Position	Distance from inlet m	Area mm ²	Normal vector			Sense	Elastic/Rigid
					X	Y	Z		
Pipe	1	Interior	9	8213	-1	0	0	Outlet to inlet (-1)	Elastic
Pipe	1	Outlet	10	8213	1	0	0	Inlet to outlet (+1)	n/a
Pipe	2	Inlet	0	8213	0	0	1	Outlet to inlet (-1)	n/a
Pipe	2	Interior	1	8213	0	0	-1	Inlet to outlet (+1)	Elastic

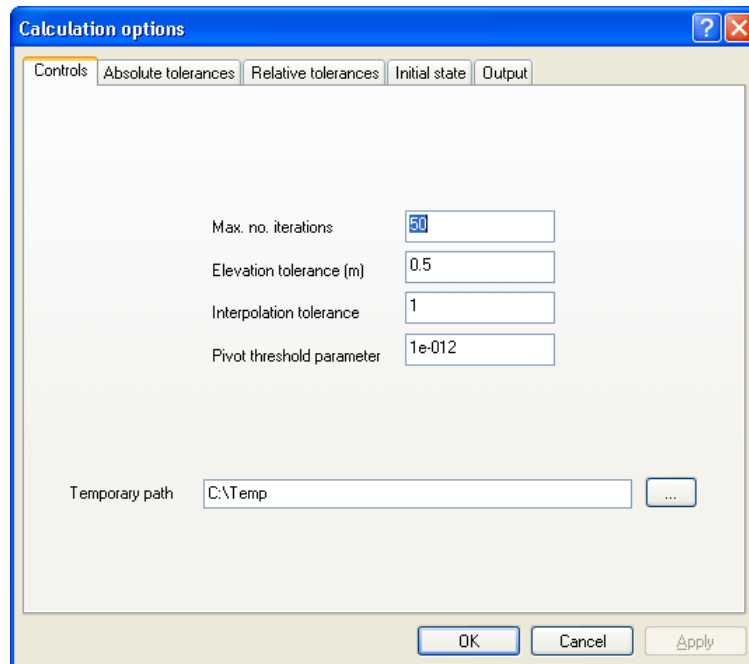
OK Cancel

16.5 Displaying Results

Once a calculation has been performed graphical results may be selected by display, either directly by right-clicking on a component and selecting the option **View Results**, or if the Graph View is already active, by selecting from the list of available graphs there. For further information on viewing results please see the section [Viewing the results for Tutorial example 1](#).

16.6 Controls

This dialog defines some general control for each calculation:



Maximum number of iterations

Almost all calculations will converge to a solution within the default setting of 50. However, if a calculation fails to converge within 50 iterations, try increasing the number.

Elevation tolerance

This specifies the tolerance to be applied when performing a height check, the default being 0.5 m.

Interpolation tolerance

Defines how complex the internal representation of the network should be. Decrease the default of 1.0 for a more accurate solution.

Pivot threshold parameter

Defines how a singular Jacobean matrix is detected, uses the default unless you are an experienced user, or if a change has been recommended by Sunrise Systems Limited.

Temporary path

This is the path where all temporary calculation files will be stored. Use either the default or select the ... button adjacent to the path name to browse for a rectory.

16.7 Absolute tolerances

These fields can be used to control the accuracy of the calculation. Advanced users can use these fields to fine tune the calculation process. The default settings are adequate for most situations.

Parameter	Value	Parameter	Value
Dimensionless variables	0.0001	Viscosity (Pa s)	0.001
Length (m)	0.001	Bulk modulus (GPa)	0
Diameter (mm)	1	Volume (l)	0.001
Area (mm ²)	1	Time (sec)	0
Pressure (Bar)	1e-005	Force (N)	0
Velocity (m/sec)	0	Mass (kg)	1e-006
Flowrate (l/min)	0.06	Temperature (°C)	0
Density (kg/m ³)	0.001		

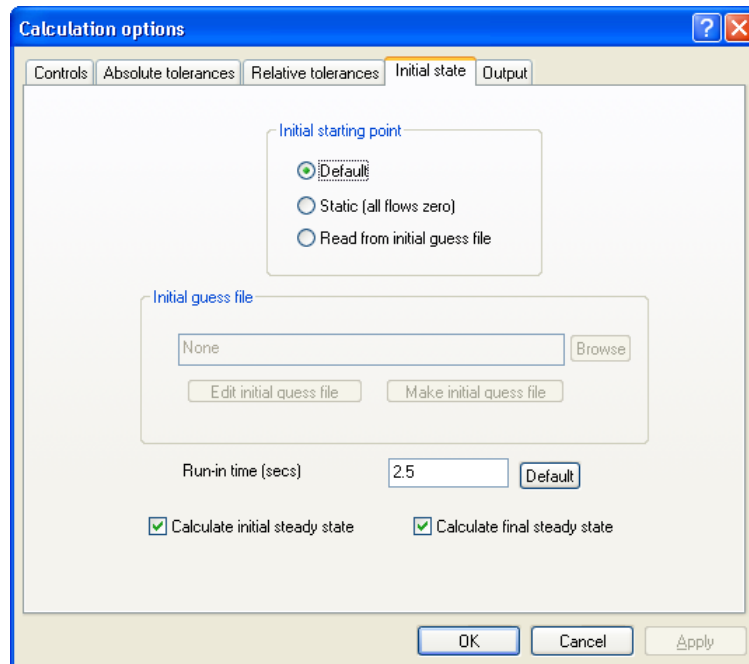
16.8 Relative tolerances

These fields can be used to control the accuracy of the calculation. Advanced users can use these fields to fine tune the calculation process. The default settings are adequate for most situations.

Parameter	Value	Parameter	Value
Dimensionless variables	0.001	Viscosity	0.001
Length	0.001	Bulk modulus	0.001
Diameter	0.001	Volume	0.001
Area	0.001	Time	0.001
Pressure	0.001	Force	0.001
Velocity	0.001	Mass	0.001
Flowrate	0.001	Temperature	0.001
Density	0.001		

16.9 Initial state

The initial state dialog allows you to set the initial state of the simulation and to create/edit an [Initial guess file](#):



The starting point for the simulation is selected from the available options, the default being that the starting point will be generated by PIPENET. Alternatively Static or [Read from an initial guess file](#) can be selected.

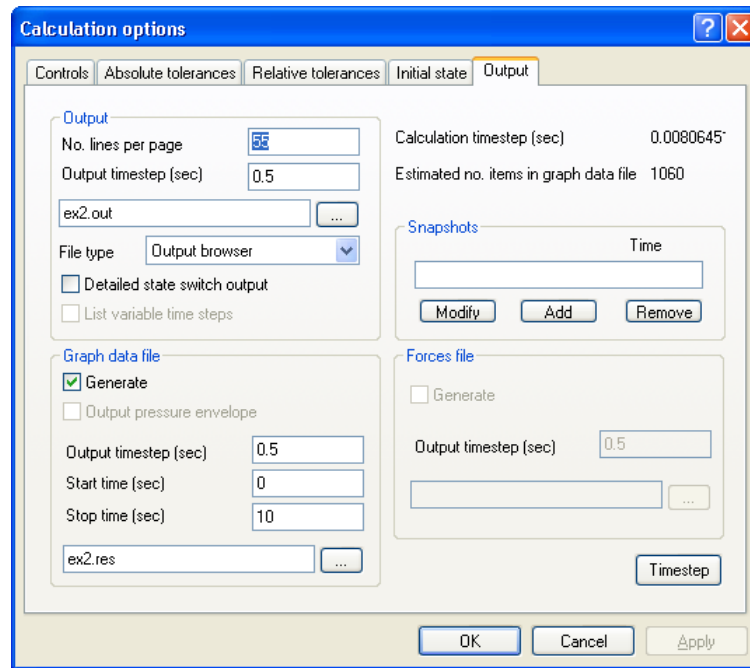
If an initial guess file is to be used then one is created by supplying a file name and selecting Make initial guess file. An existing initial guess file may be edited by entering the file name and selecting Edit initial guess file. It should be noted that a new initial guess file must be created if the network is changed in any way.

The run-in time can be entered, or the default button selected to assign a run-in time of one quarter of the total simulation time.

The calculation of initial and final steady states is specified by checking the appropriate check boxes.

16.10 Transient Output

Transient Module has three types of output file. The program cannot run a calculation until an Output Report filename has been specified. The Graph and Forces Output are optional and contain Graph and Forces data respectively.



The Report File - .out

This is the file created by PIPENET to store all output details of the simulation. The file includes a listing of the network data file used, details of the valves, pumps and valves in the network, and tables of results requested by the User.

The Graph Output File - .res

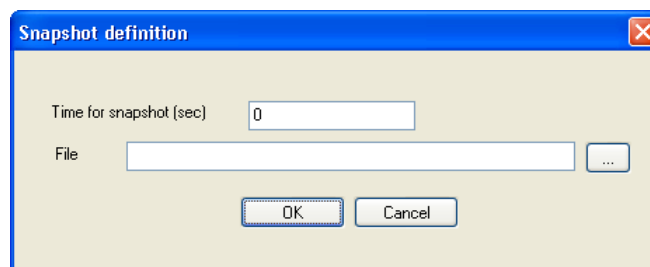
This file contains data for the Transient Module Graph Viewer. Output graphs are selected by the User before running the simulation and the graphical results can be displayed and edited in the Graph Viewer. A restricted time range for graphical output can be selected by specifying the start and stop times.

The Forces Output File - .frc

This file contains data for the force results.

Snapshots

A snapshot generates an [initial guess file](#) for the network at any given time during a simulation. This initial guess file can then be used to determine the initial state of a second simulation. Selecting the **Add** button will display the snapshot dialog box:

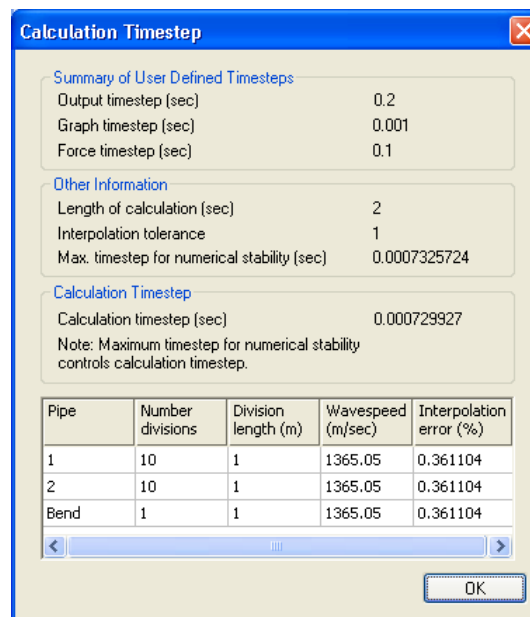


allowing entry of the snapshot time and the name of the snapshot file.

Selecting the **Modify** button will allow a user to edit an existing snapshot and selecting **Delete** will delete an existing snapshot.

Time step

Selecting the **Timestep** button will allow a user to examine the value the simulation will use for the calculation, and to identify which pipes may cause the simulation to run for a long time. Information on various factors which affect the choice of time step is displayed:



A list of pipes is provided, each pipe displaying the number of sections into which the pipe has been divided, the length of each section and the wavespeed. Also displayed is the interpolation error, which is a guide to how well the network has been divided.

16.11 Maximum and minimum pressure

Maximum/minimum pressure compares the instant calculation results and determines the maximum and minimum pressures occurring in every single pipe, valve, pump, caisson, tank and bursting disc. Every component has maximum and minimum pressure value at inlet and outlet respectively. Apart from that, the pipe has the pressure extremum occurring in the interior pipe, and the two node caisson has the air pressure extremum inside caisson. The time values when the extremum pressures happened are also recorded.

One of the reasons why this feature is provided is the following. Sometimes the weakest part of the system is not the place where the maximum pressure surge occurs. For example, if the system is made of carbon steel as well as GRP pipes, while the maximum pressure may occur in a carbon steel pipe a lesser pressure could damage the GRP pipe. So it is important to know the maximum pressure in the GRP part of the system. In a cross country pipeline in mountainous regions, the pipe schedule may vary depending on the operating pressure at a point. In such a case, the global maximum pressure may not be the cause of potential damage. A weaker pipe experiencing a lower pressure may be more

vulnerable.

Display in the output report

The maximum/minimum pressure results are listed in the output report file with the format shown in the next page. All the components are grouped according to component type, and the component type names are displayed on the top of each group. For the pipe, middle maximum/minimum pressure means the maximum/minimum pressure occurring in the pipe interior, and for the caisson, middle maximum/minimum pressure means the maximum/minimum air pressure occurring inside caisson.

[Untitled] Maximum/minimum pressure report													Output Page 14			
													23050221			
MAXIMUM/MINIMUM PRESSURE																
Component	Inlet max.		Inlet min.		Outlet max.		Outlet min.		Middle max.		Middle min.		Time	pressure	Time	
	pressure	Time	pressure	Time	pressure	Time	pressure	Time	pressure	Time	pressure	Time				
	bar A	seconds	bar A	seconds	bar A	seconds	bar A	seconds	bar A	seconds	bar A	seconds	bar A	seconds	bar A	seconds
PIPE																
1	0.110E+02	0.500E+01	0.101E+01	0.000E+00	0.111E+02	0.538E+01	0.101E+01	0.740E-01	0.111E+02	0.538E+01	0.101E-01	0.000E+00				
2	0.111E+02	0.538E+01	0.101E+01	0.740E-01	0.111E+02	0.538E+01	0.101E+01	0.810E-01	0.111E+02	0.538E+01	0.101E-01	0.740E-01				
3	0.111E+02	0.538E+01	0.101E+01	0.810E-01	0.111E+02	0.530E+01	0.101E+01	0.155E+00	0.111E+02	0.530E+01	0.101E-01	0.114E+00				
4	0.111E+02	0.530E+01	0.101E+01	0.155E+00	0.112E+02	0.523E+01	0.101E+01	0.229E+00	0.112E+02	0.523E+01	0.101E+01	0.229E+00				
5	0.530E+01	0.204E+01	0.151E+01	0.229E+00	0.229E+01	0.200E+01	-0.142E+01	0.266E+00	0.530E+01	0.204E+01	-0.142E-01	0.266E+00				
CAISSON																
1	0.214E+01	0.100E+02	0.199E+01	0.155E+00	0.101E+01	0.000E+00	0.101E+01	0.000E+00	0.102E+01	0.100E+02	0.101E-01	0.177E+00				
2	0.215E+01	0.100E+02	0.199E+01	0.810E-01	0.101E+01	0.000E+00	0.101E+01	0.000E+00	0.102E+01	0.465E+01	0.101E-01	0.353E+00				
PUMP																
1	0.112E+02	0.523E+01	0.101E+01	0.229E+00	0.530E+01	0.204E+01	0.151E+01	0.229E+00								
2	0.111E+02	0.530E+01	0.101E+01	0.155E+00	0.214E+01	0.100E+02	0.199E+01	0.155E+00								
3	0.111E+02	0.538E+01	0.101E+01	0.810E-01	0.215E+01	0.100E+02	0.199E+01	0.810E-01								
4	0.111E+02	0.538E+01	0.101E+01	0.740E-01	0.151E+01	0.100E+02	0.151E+01	0.900E-02								
VALVE																
1	0.229E+01	0.200E+01	-0.142E+01	0.266E+00	0.101E+01	0.000E+00	0.101E+01	0.000E+00								

Display in the user interface

Minimum and maximum pressure information is also displayed for each component in the [Properties window](#) and in the Results page of the [Data window](#).

Part

17

17 Graph Viewer

17.1 Graph Viewer introduction

Opening the Graph Viewer

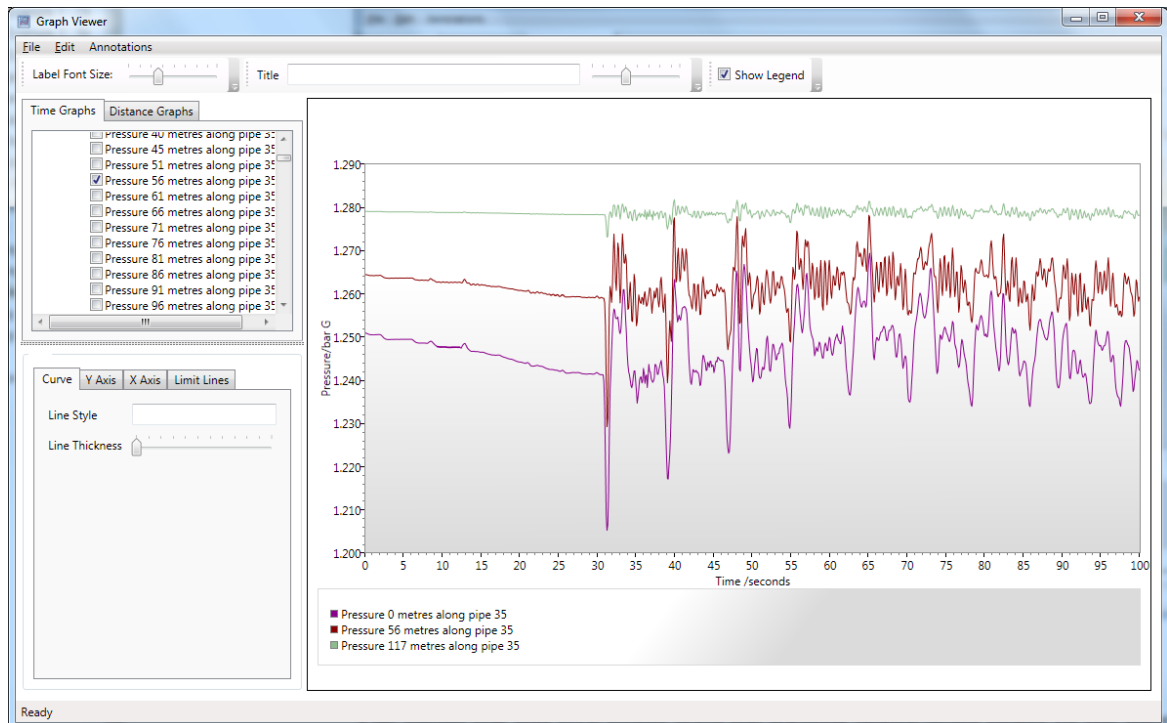
The Graph Viewer is opened from the Transient Module User Interface by selecting **Output | Graphs**. If graphical output has been selected the results file is automatically opened by the program.

Graph result files have a .res extension. Once the Graph Viewer has been opened it can be used to open any other .res file.

The Graph Viewer Desktop

The Graph Viewer desktop is shown below. The application window is split into three panes. The graph window, the graph tree control and the final pane contains the tabs that enable the user to alter various properties of the currently selected graph.

There are three toolbars across the top of the main application window. One is for controlling the size of the annotation text, one for creating a chart title and the other enables you to hide the chart legend.



The menu bar has three items. The three menus are summarized below, and described in

more detail later in
this section:

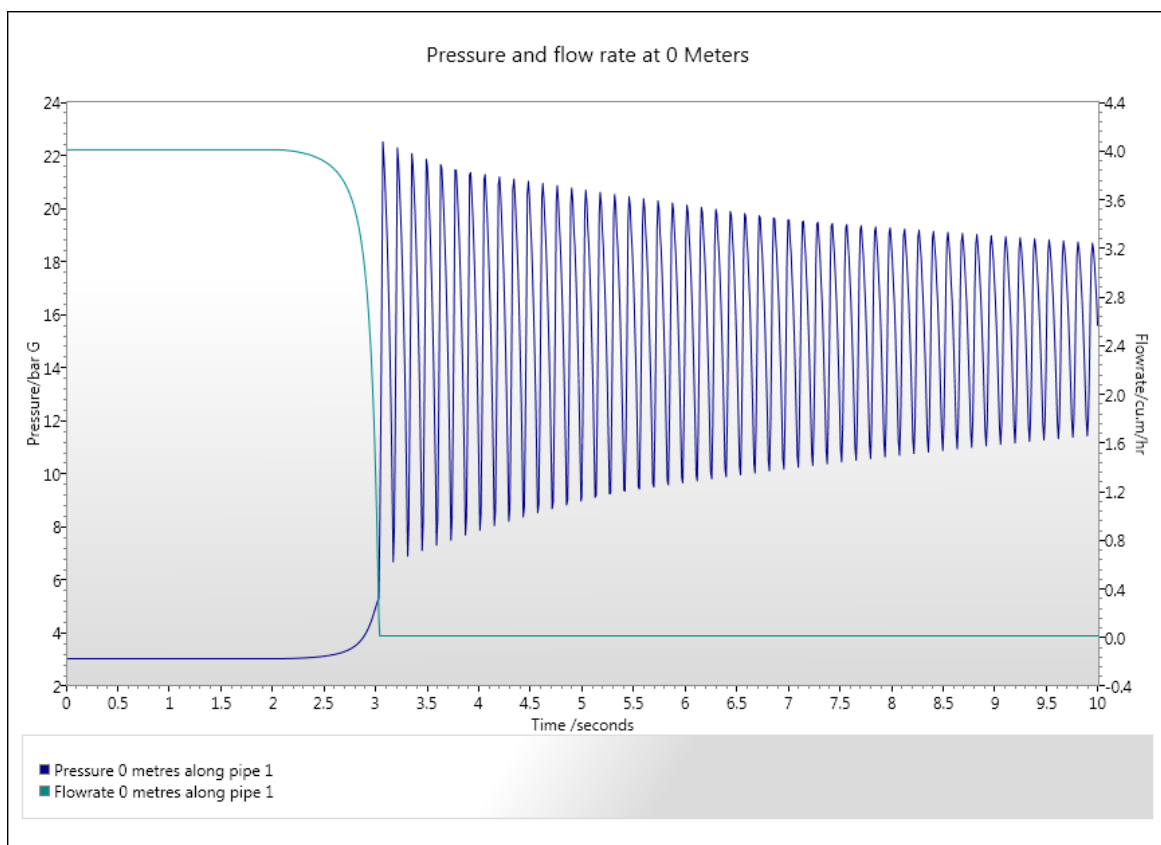
File	Opens graph .res files, prints and exports graph as an image.
Edit	Copy graph.
Annotations	Adds labels and arrows to the graph

File menu

Open	Opens a graph .res file.
Show Movie	Launches the movie window.
Export	Export graph window as an image. The supported formats are XPS, PNG and JPEG.
Print	Print the graph.
Exit	Closes the Graph Viewer.

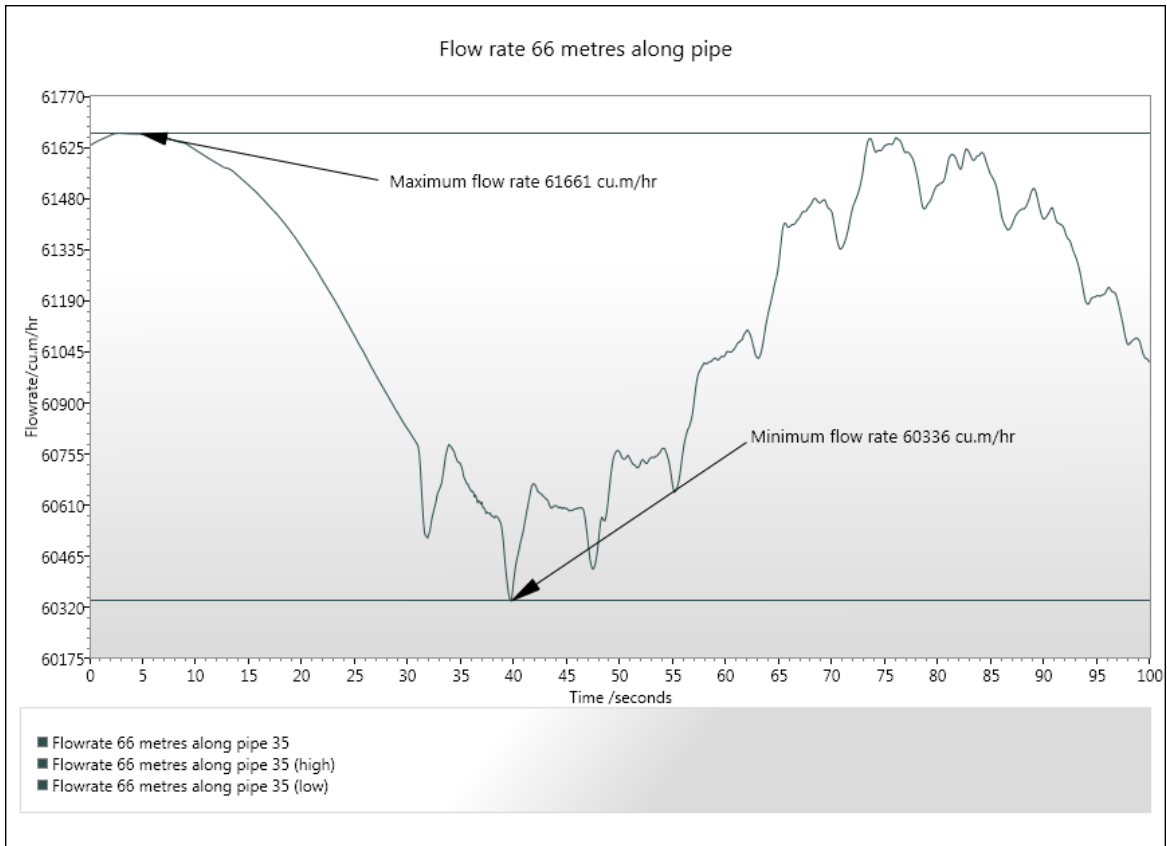
Edit menu

Copy	Copies the graph to the clipboard.
-------------	------------------------------------



Annotation

Labels	Insert/Delete a text label on the graph.
Arrows	Insert/Delete an arrow on the graph.

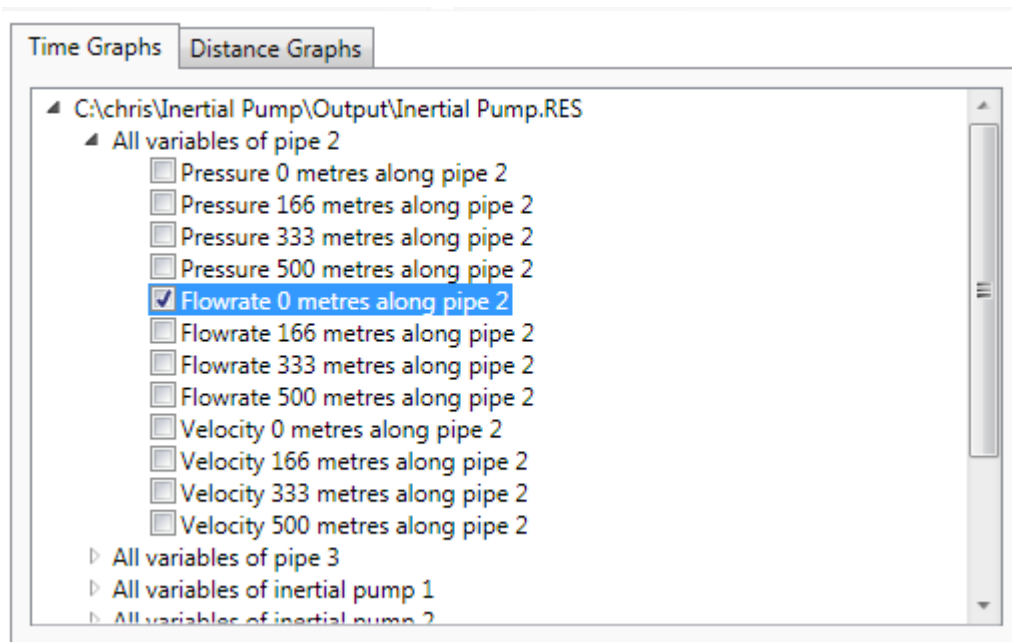


17.2 Different graph types

The Graph Viewer allows three different types of graph to be drawn.

Variable against Time

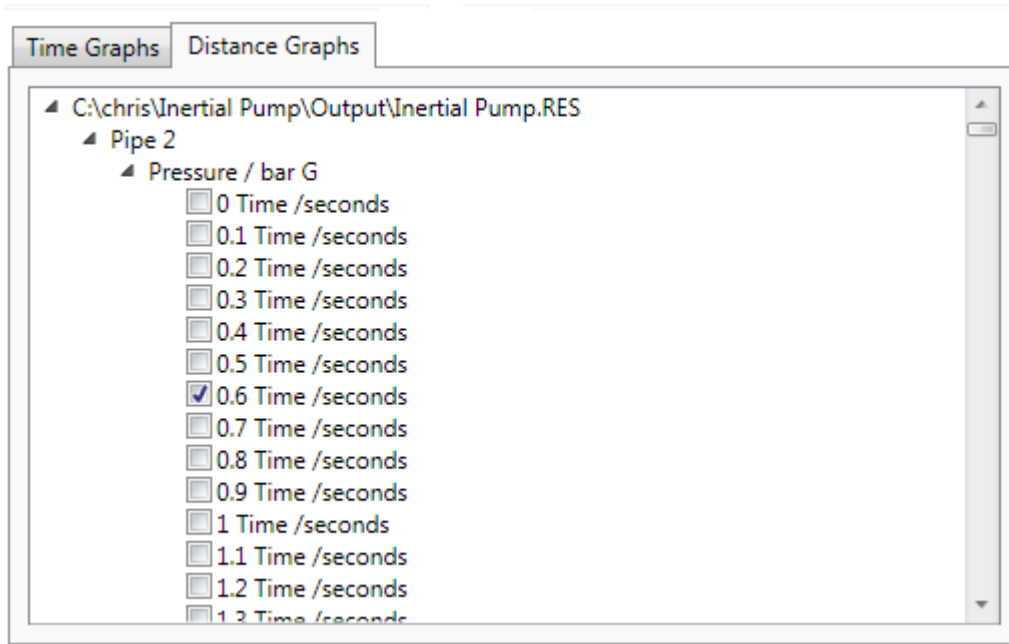
Draws a graph of component variable against time. The time interval is that selected in the transient "Calculate Output" dialog (by default the total simulation time). To display a particular graph click the check box next to the required graph. When the graph has been drawn its properties can be altered by using the property tabs.



Pipe Variable against Distance

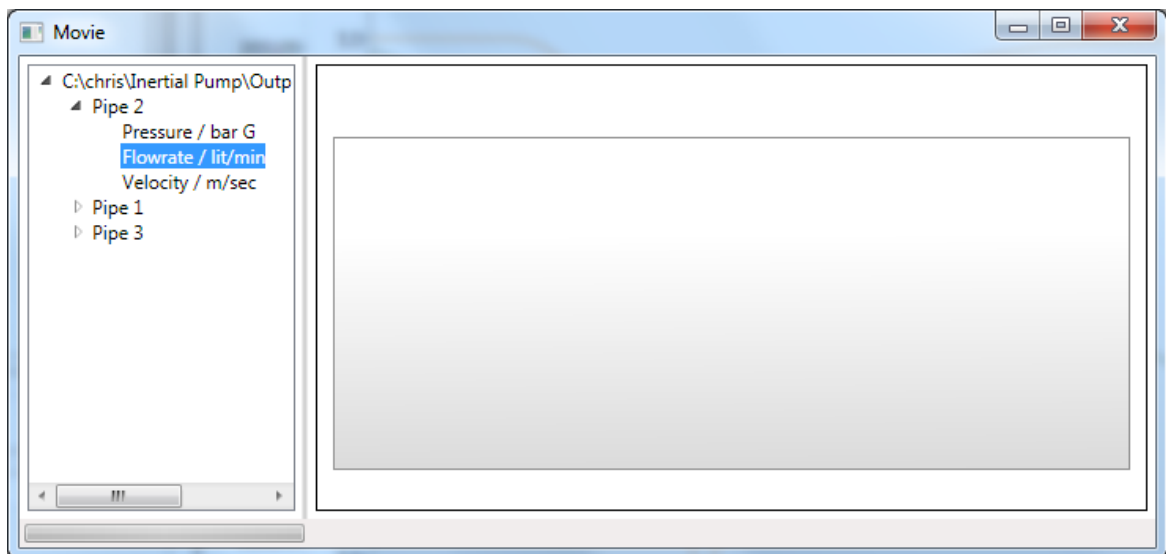
Draws a graph of pipe variable against distance along the pipe, at a specified time. The Distance Graphs tab contains a list of pipes selected for graphical output.

The appropriate graph can be selected by expanding the tree control and clicking the check box next to the graph title. When the graph has been drawn its properties can be altered by using the property tabs.



Pipe Variable Movie

Shows a 'Movie' display of pipe variable against distance along the pipe. Selecting File->Show Movies displays the following window.



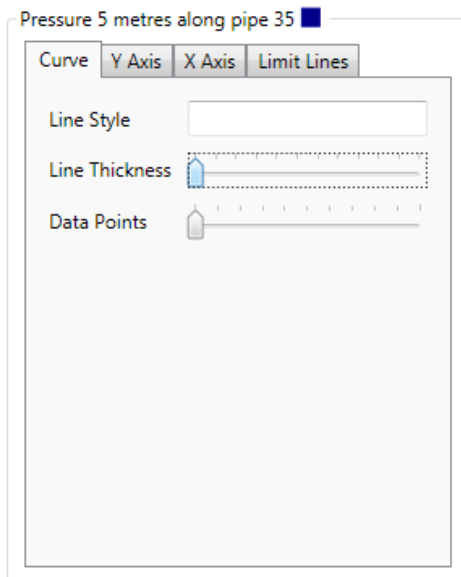
Expand the tree, and right click on the graph to be played. The window closes automatically when the movie finishes.

17.3 Using the property tabs

Once a graph has been drawn, its appearance can be changed by using the property tabs. The property pages for a particular graph are selected by left-clicking on the graph in the graph tree control.

In the following sections the 4 different property tabs are described in detail.

17.4 Curve property tab



Set the property options for the curve

Line Style

A series of numbers which are interpreted as a stroke dash pattern for the curve. Each number in the list specifies the length of a dash or gap relative to the Thickness of the line of the cure. For example, a value of 1 creates a dash or gap that has the same length as the thickness of the pen (a square).

The first item in the collection, index 0, specifies the length of a dash; the second item, which is located at index 1, specifies the length of a gap.

Objects with an even index value specify dashes; objects with an odd index value specify gaps. eg a Line style of 4 1 will result in the following line
The dash is 4 times the size of the line thickness and the gap is the same size as the line thickness.



Line Thickness

A Slider control changes the thickness of the currently selected curve.

Data Points

Moving the Slider control cause the curve's data points to fade into and fade out of view.

17.5 Y Axis property tab

Pressure 166 metres along pipe 2 ■

Curve Y Axis X Axis Limit Lines

Axis Title

Show Grid Lines

Axis Position

Hide Axis

Font Size

Range

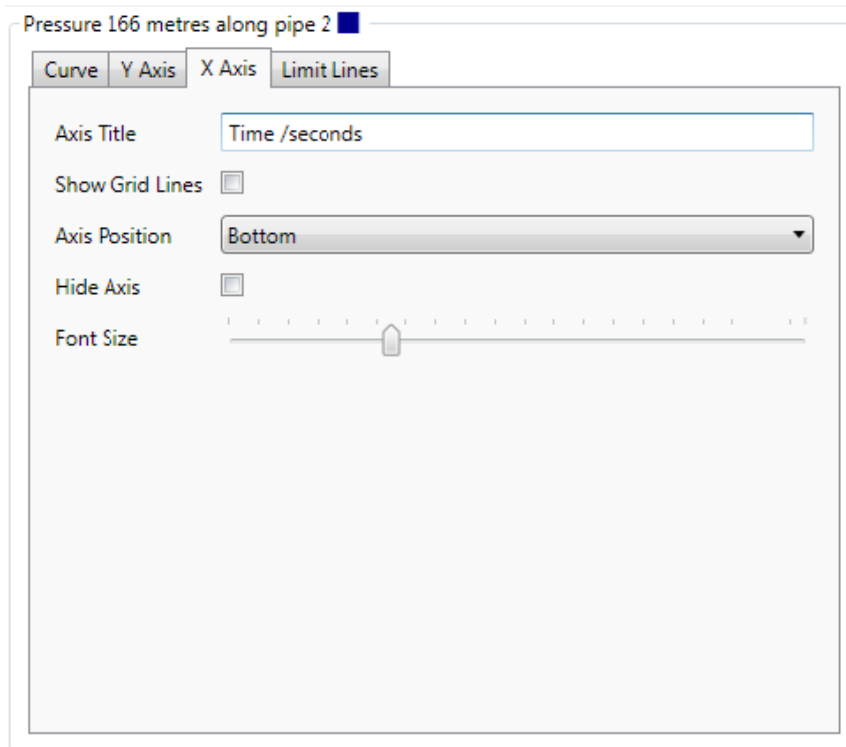
Begin

End

Interval

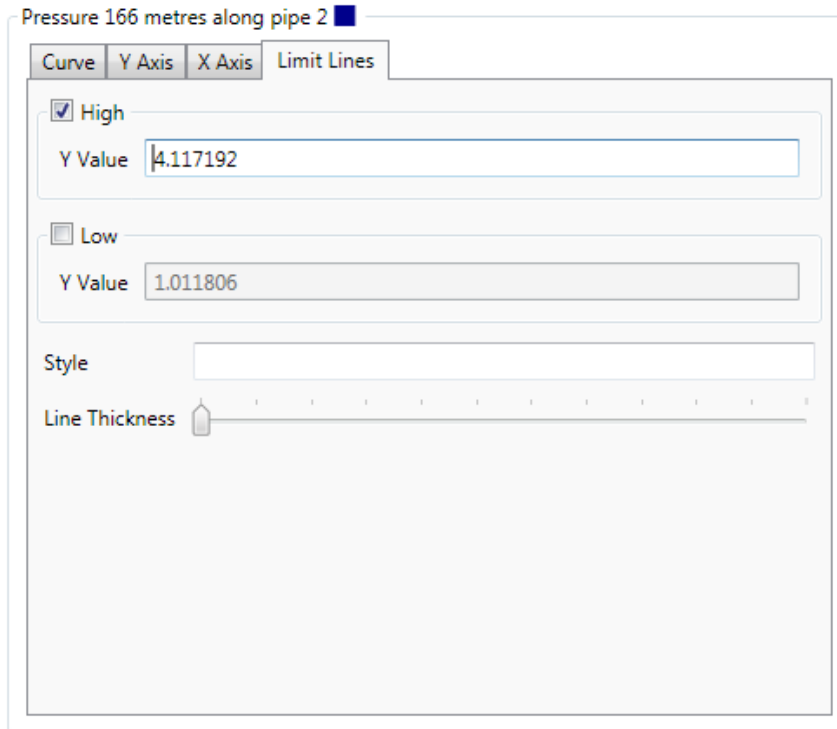
Axis Title	The axis title can be changed.
Show Grid Lines	Show/hides the grid lines along the Y (vertical) axis.
Axis Position	The Y (vertical) axis can be moved from the left hand side of the chart to the right hand side.
Hide Axis	The Y (vertical) axis can be hidden
Font Size	The size of the axis title text can be changed by moving the slide control.
Range	The begin, end and interval properties of the Y axis can be changed.

17.6 X Axis property tab



Axis Title	The axis title can be edited.
Show Grid Lines	Show/hides the grid lines along the X (horizontal) axis.
Axis Position	The X (horizontal) axis can be moved from the bottom of the chart to the top of the chart.
Hide Axis	The X (horizontal) axis can be hidden.
Font Size	The size of the axis title text can be changed by moving the slider.

17.7 Limit Lines properties tab



To display the maximum and minimum lines for the selected graph, click the appropriate check box.

Y Value

The default value is the maximum Y value from the graph's data points. This value can be changed by the user.

Style

A series of numbers which are interpreted as a stroke dash pattern for the curve. Each number in the list specifies the length of a dash or gap relative to the Thickness of the line of the curve. For example, a value of 1 creates a dash or gap that has the same length as the thickness of the pen (a square).

The first item in the collection, specifies the length of a dash; the second item, which is located at index 1, specifies the length of a gap.

Objects with an even index value specify dashes; objects with an odd index value specify gaps. eg a Line style of 4 1 will result in the following line The dash is 4 times the size of the line thickness and

the dash is the same size as the line thickness.



Line Thickness

Moving the slider changes the thickness of the limit line.

17.8 Labels and Arrows

The graphs can be annotated by adding labels and arrows to the chart. Labels and arrows can be added either via the annotations menu, or via the chart's context menu (right click within the chart area).

Select Annotation->Labels->Create.

"Label text" will appear in the chart. Click on the text to edit the label. Move the mouse just to the side (top, bottom, left, right) of the text and a blue rectangle surrounds the label and the cursor changes to indicate the text can be moved.

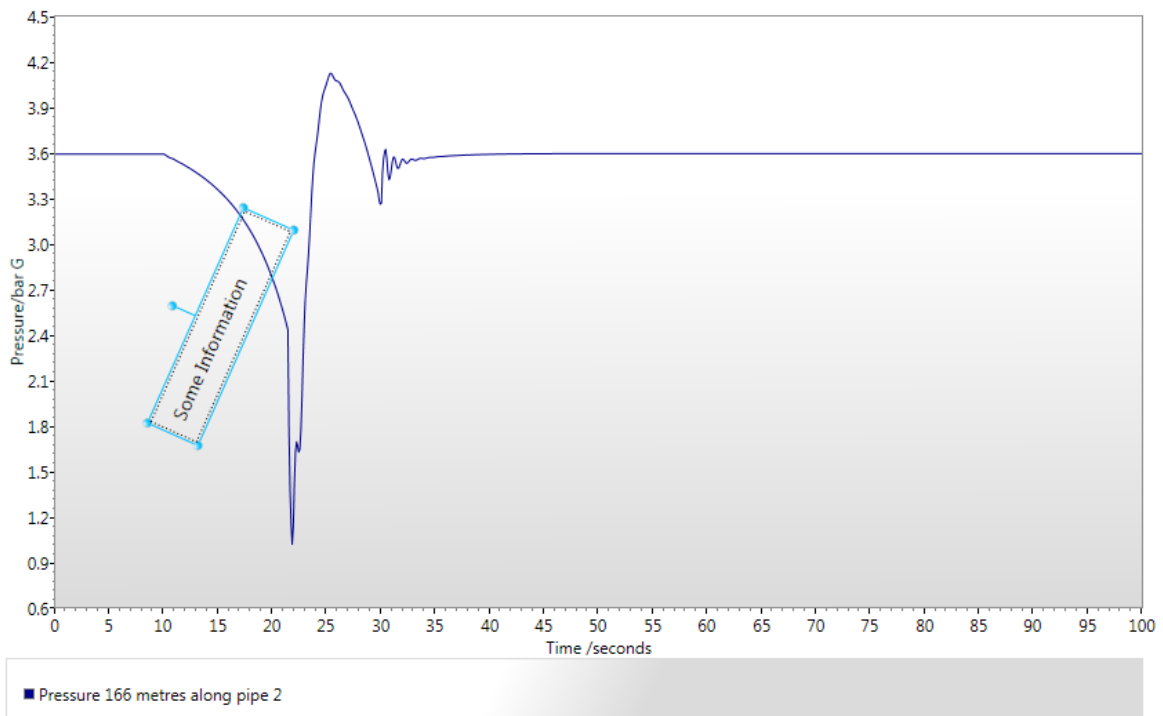
Double clicking on the text will display the resize/rotation control (see below).

The text box can be resized by dragging the corners, and can be rotated by clicking on the circle on the top of the control.

To delete a label, double click on the label to reveal the resize/rotate control.

Right click on the chart and select Labels->Delete.

Inserting, resizing and deleting arrows, works in the same way as described above.



17.9 Hot spot

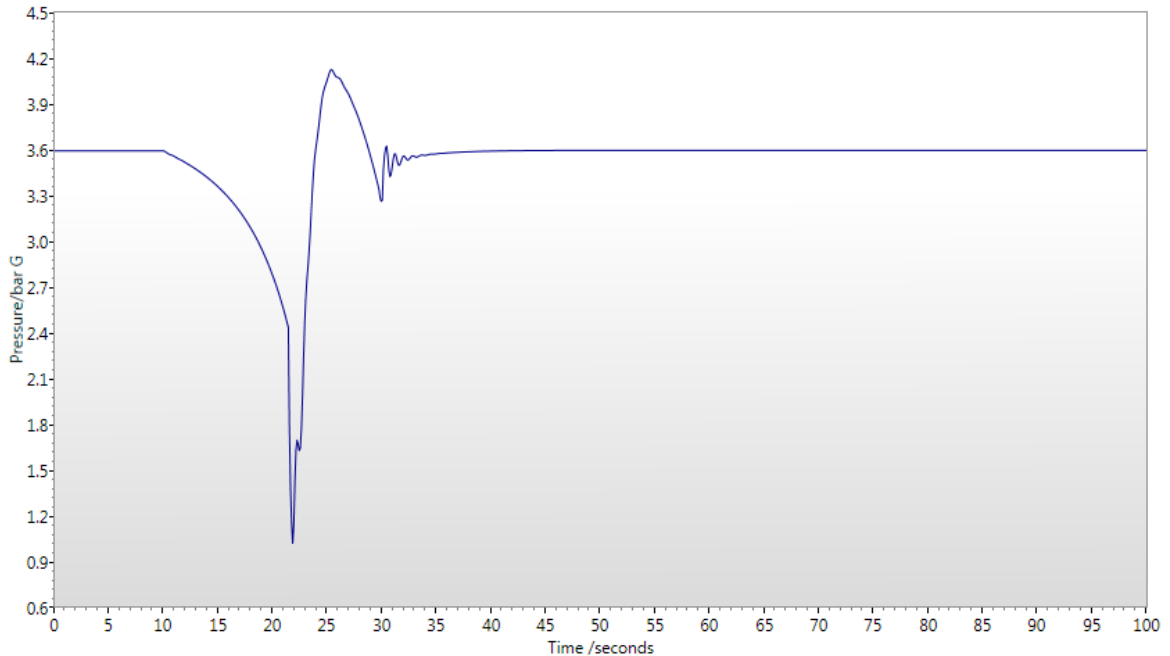
Trace the curve with the mouse, when a data point is encountered, a tool tip will appear displaying the coordinates of that data point.

The data points can be made visible via the Curve properties tab.

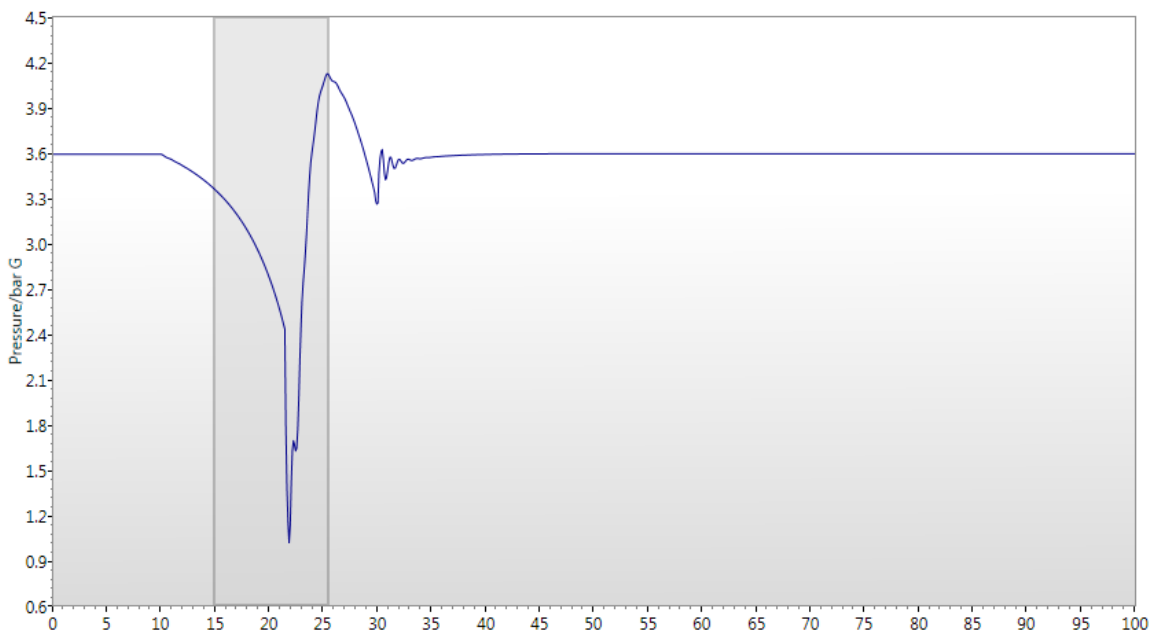


17.10 Zooming

It is possible to zoom the graph to an area of interest.

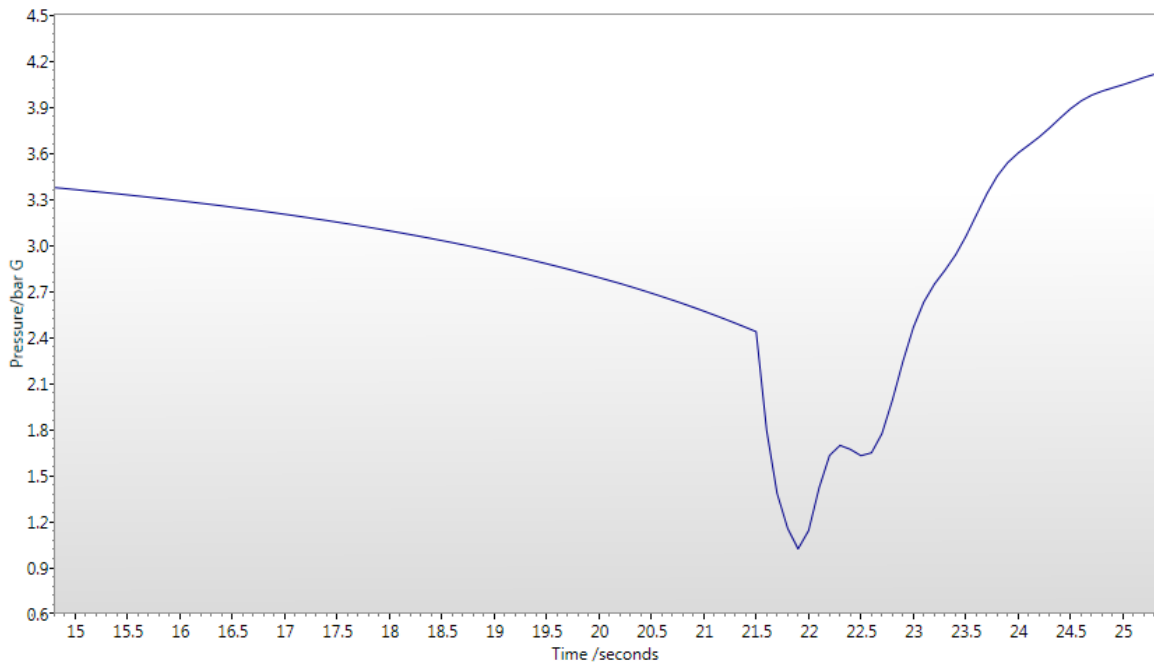


Hold down the Ctrl key, click and hold down the left mouse button and drag the mouse to highlight the area to be zoomed.



Release the mouse button and the Ctrl key, the graph redraws to show the zoomed area.

The zoomed graph can be zoomed further by repeating the above steps.



To zoom out a zoomed graph, hold down the Alt key and click inside the chart with the left mouse button.

Modelling

Part

18

18 Modelling

18.1 Standard and Spray options

On running the Transient Module the user has the choice between two modes of operation.

- The STANDARD option is the default general use option.
- The SPRAY option is for the analysis of fire protection systems in accordance with either NFPA or FOC rules.

There are a number of important differences between the two options, listed in the table below:

	Standard	Spray
Pipe Schedules	Thirty two built-in schedules, or user-defined	Eight built-in schedules, or user-defined
Pipe Linings	Not available	Three built-in linings or user-defined
Pipe Fittings	Converted to a k-factor	Converted to an equivalent length of pipe
Pressure Drop Equation	Bernoulli equation	Bernoulli or Hazen-Williams equation

18.2 HazenWilliams

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

18.3 Coulson and Richardson

The Coulson and Richardson equation is a theoretical equation which gives the pressure in pipes, ignoring frictional effects. By comparing the theoretical results obtained using the Coulson and 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 ($Re = uD/\nu$ where ν is the fluid dynamic 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 ($Re < 2000$):

$$f = \frac{16}{Re}$$

Transitional flow ($2000 < Re < 3000$):

f is found by interpolating between laminar value for $Re = 2000$ and turbulent value at $Re = 3000$.

Turbulent flow ($Re > 3000$):

$$\frac{1}{\sqrt{f}} = -1.768 \ln \left(\frac{0.27r}{D} + \frac{1,252}{Re\sqrt{f}} \right)$$

18.4 Pipe C-factor

There are two possible methods for calculating pressure drop in pipes - the [Coulson and Richardson](#) method and the [Hazen-Williams](#) method.

The Coulson and 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.

18.5 Linings

When operating in the [Spray mode](#), 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..

18.6 Fluids

Fluids are defined in the Transient module by a set of five parameters:

- fluid density,
- fluid viscosity,
- temperature,
- bulk modulus
- vapour pressure.

These parameters can be provided in one of three ways:

1. They can be entered directly by the user via the [Fluid dialog](#)
2. For water and steam they can be calculated, based on a given temperature and/or pressure, again via the [Fluid dialog](#)
3. They can be calculated according to American Petroleum Institution criteria via the [API Fluid dialog](#) using a number of parameters provided by the user. Note that this dialog will calculate a number of results but only the 5 parameters specified above are used in the calculation.

API Fluids

An alternative method of defining fluids is via the use of an API (American Petroleum Institute) fluid function. This function specifies the fluid in use on the basis of the petroleum characterization correlation methods provided by API Technical Data Book. The current version can calculate the following properties: vapour pressure, heat capacity, atmospheric heat capacity, thermal conductivity, atmospheric thermal conductivity, viscosity, surface tension, density, atmospheric density, secant bulk modulus, molecular weight, and compressibility. However, the properties used in the current simulation are only vapour pressure, viscosity, density, and secant bulk modulus.

To calculate the API fluid properties a number of parameters must be provided, some of these parameters are mandatory, whilst others are optional. The four mandatory parameters are:

- Input temperature;
- Input pressure;
- API or specific gravity;
- Watson characterization or mean average boiling point.

The optional parameters are:

- Viscosity correction - specified as a reference viscosity and temperature, or as the critical viscosity;
- Molecular weight;
- Reference density - supplied as a reference density and temperature.

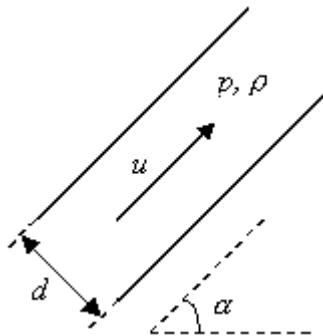
Calculation of the API results will involve a number of checks and the following error conditions can be detected:

- Error 1 - the input data are not valid for API heat capacity calculation - the heat capacities cannot be calculated.
- Error 2 - the reference data are not valid for API liquid viscosity calculation method. The default reference data are applied in the calculation.
- Error 3 - the input data are not valid for API heat capacity calculation - the heat capacities cannot be calculated. The reference data are not valid for API liquid viscosity calculation method - the default reference data are applied in the calculation.

- Error 4 - the input data are not valid for API liquid viscosity calculation - the liquid viscosities cannot be calculated.
- Error 5 - the input data are not valid for API heat capacity calculation - the heat capacities cannot be calculated. The input data are not valid for API liquid viscosity calculation - the liquid viscosities cannot be calculated.
- Error 6 - The input data are not valid for API fluid calculation.- the API fluid properties cannot be calculated.

18.7 Fluid Equations

Transient Module solves water hammer equations in the pipes.



Momentum and continuity equations are solved numerically for pressures and flow rates. Other component modelling equations are described in subsequent sections of this chapter.

Momentum Equation

Applying Newton's second law to a control volume of fluid, and allowing for friction and gravity, one obtains:

$$\frac{1}{\rho} \frac{\delta p}{\delta x} + \frac{\delta u}{\delta t} + g \sin \alpha + \frac{4f}{d} \frac{u|u|}{2} = 0$$

Continuity Equation

The rate of increase of mass of a control volume of fluid is equal to the net mass flow rate into the volume. i.e.

$$\frac{1}{\rho A} u \frac{\delta \rho A}{\delta x} + \frac{1}{\rho A} \frac{\delta \rho A}{\delta t} + \frac{\delta u}{\delta x} = 0$$

where:

p is the pressure in the pipe.

u is the fluid velocity along the pipe.

x is the distance along the pipe.

t is the time.

A is the cross-sectional area of the pipe.

d is the diameter of the pipe.

ρ the fluid density.

α the angle the pipe makes to the horizontal.

f is a friction factor.

18.8 Wave Speed

The wavespeed in a pipe is the speed at which pressure surges are propagated along the pipe. It depends on a number of factors including the material and diameter of the pipe, and the bulk modulus of the fluid. For Transient Module to make an automatic calculation of the wavespeed the User must provide a pipe schedule and define the fluid bulk modulus. If this information is unavailable a default wavespeed can be specified.

Details of wavespeeds for each pipe in the network are listed in the output report file, along with the rest of the pipe data.

The wavespeed is calculated using:

$$a = \sqrt{\frac{K/\rho}{1 + \left(\frac{KD}{Et}\right)c_1}}$$

where:

a is the wavespeed.

K is the fluid bulk modulus.

D is the internal pipe diameter.

E is the Young's Modulus for the pipe material.

t is the pipe wall thickness.

c_1 is the pipe distensibility (restraint factor).

ρ the fluid density.

The pipe distensibility is calculated assuming the pipe is anchored at both ends against longitudinal movement:

$$c_1 = \frac{2t}{D}(1+\mu) + \frac{D(1-\mu^2)}{D+t}$$

where μ is the Poisson ratio for the pipe material.

If a pipe is not anchored at both ends the pipe wavespeed should be specified by the User as a [user-defined wavespeed](#). The earlier formula for the wavespeed should be used, with pipe distensibility given by:

$$c_1 = \frac{2t}{D}(1+\mu) + \frac{D(1-\mu/2)}{D+t}$$

for a pipe anchored at the upstream end only, and:

$$c_1 = \frac{2t}{D}(1+\mu) + \frac{D}{D+t}$$

for a pipe having expansion joints throughout its length.

18.9 Specifications

Specifications are used to describe behaviour of valves and pumps in the network and boundary value pressures and flow rates. A Pressure or Flow rate specification is required at all input and output nodes. Note that:

1. PIPENET uses the convention that flow rate is positive into, and negative out of, the system
2. There must be at least one pressure specification somewhere in the network.
3. Only one specification is allowed on each node.
4. No pressure or flow rate specification is allowed on an interior node.

An Information specification is used to control the operation of a pump or valve. The specification value should lie between 0 and 1 (from 0 for a fully closed valve / shut down pump to 1 for a fully open valve / full speed pump). The information specification should be attached to an information node having no other connected components.

User Input Parameters

Specification Type - Pressure or flow rate for I/O nodes, information for information nodes.

Node - Node to which specification applies.

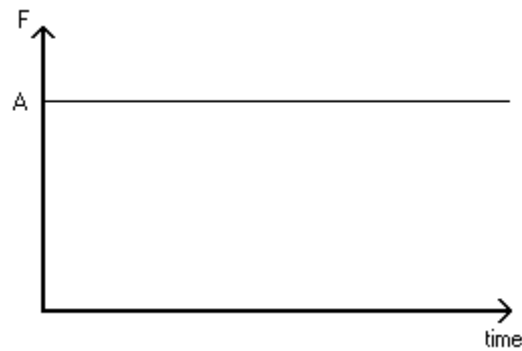
Time function - Choice of time function and associated parameters.

Modelling Equations

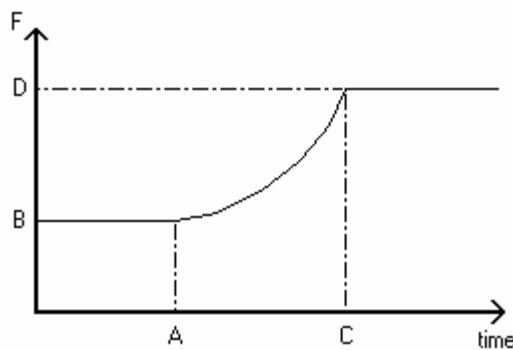
The User has the choice of nine basic types of specification. In each case the program calculates the specification value at each time.

Built-in Functions

Nine built-in functions are available. These are described below.

Constant

Gives a constant specification value throughout the simulation.
Only the constant value has to be specified.

Power Ramp

Gives a change in the specification value over a specified time period.
The specification value is assumed constant before and after the power ramp.

$$F = \begin{cases} B & (t < A) \\ |F|^E & (A < t < C) \\ D & (C < t) \end{cases}$$

where:

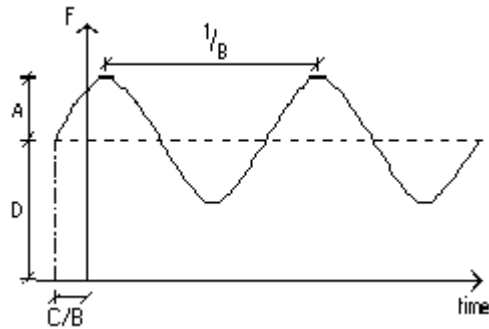
$$F' = \frac{(C-t) \left(\frac{B}{|B|} \right) (|B|)^{\frac{1}{E}} + (t-A) \left(\frac{D}{|D|} \right) (|D|)^{\frac{1}{E}}}{C-A}$$

and:

- A = power ramp start time.
- B = initial specification value.
- C = power ramp stop time.
- D = final specification value.
- E = exponent.

Sine Wave

Gives an oscillating specification value.



$$F = D + A \sin 2\pi B(t + C)$$

where:

A = amplitude.

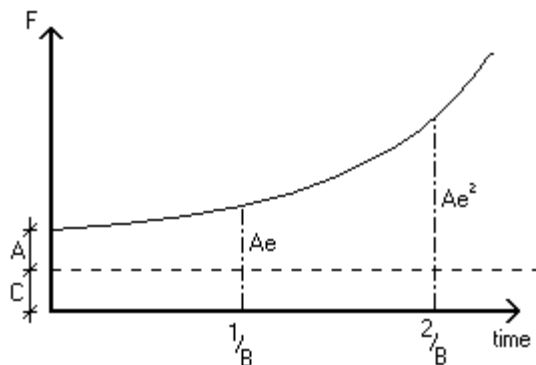
B = frequency.

C = constant time lag.

D = additive constant.

Exponential

Gives an exponentially increasing or decreasing specification value.



$$F = Ae^{Bt} + C$$

where:

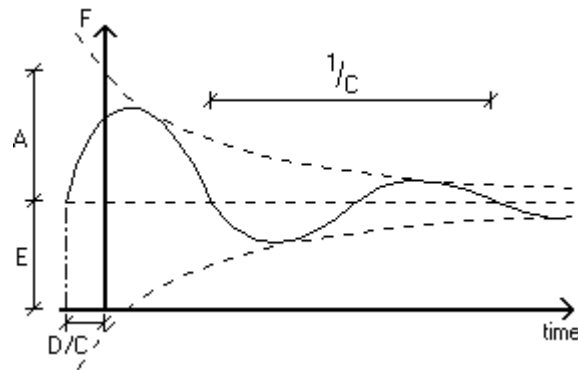
A = amplitude.

B = growth factor.

C = additive constant.

Damped Sine Wave

Gives a damped oscillating specification value throughout the simulation.



$$F = Ae^{Bt} \sin 2\pi C(t + D) + E$$

where:

A = amplitude at time $t=0$.

B = decay factor.

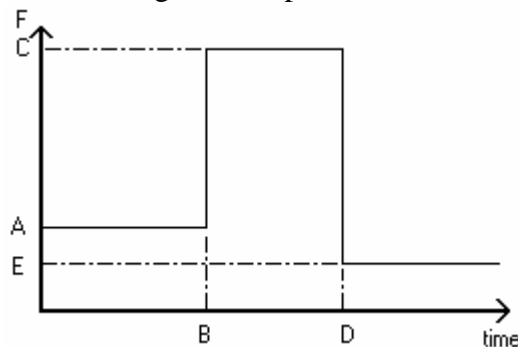
C = frequency.

D = constant time lag.

E = additive constant.

Asymmetric Pulse

Gives a sudden change in the specification value over a specified time period.



$$F = \begin{cases} A & (t < B) \\ C & (B < t < D) \\ E & (D < t) \end{cases}$$

where:

A = specification value before pulse.

B = pulse start time.

C = specification value during pulse.

D = pulse end time.

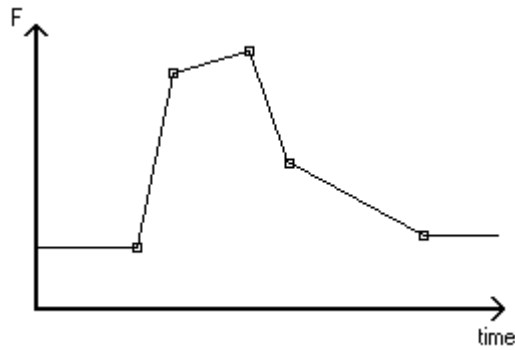
E = specification value after pulse.

Profiles

Three types of profile are available and are described below. The repeat option ensures that the specified profile points are repeated over the remainder of the time period. A maximum of 30 profiles can be modelled by any one network.

Linear Profile

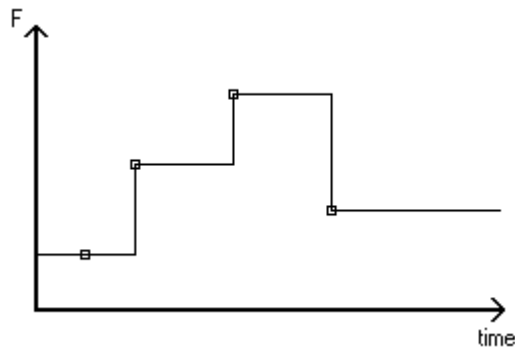
This is a more general form of the power ramp function:



Up to 40 data points can be used to define the curve and linear interpolation is used between the points.

Step Profile

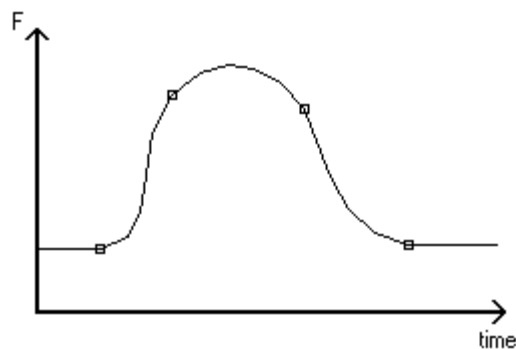
This is a more general form of the asymmetric pulse function:



Up to 40 data points can be used to define the curve and the function is assumed constant between the points.

Cubic Profile

This is a more general form of the power ramp function.



Up to 40 data points can be used to define the curve and cubic interpolation is used between the points.

18.10 Initial Guess File and the initial state

The initial state of the system will generally have a critical bearing on how the system reacts during a simulation, and thus it is important to give some consideration to starting the simulation correctly. The Transient Module obtains the initial state of the system in three stages, all of which are under user control:

Stage 1 - A starting point from which to begin is found. This can be specified by the user to be either a state in which all the fluid is at rest (i.e. static), or some other user-defined state using an initial guess file. If an initial guess file is not provided the Transient module will supply its own guesses.

Stage 2 - The starting point can be evolved by performing a number of transient steps with the clock turned off - that is without any changes in valve or pump settings etc. This period is also called the run-in time.

Stage 3 - The Transient module will, if required, calculate the state of the system under conditions of steady flow, that is the steady state of the system. This calculation is made using an iterative method starting from the state obtained at the completion of Stage 2.

Frequently, all three stages will not be required. For example, in order to start the system in a state defined by the user (using an initial guess file), stages 2 and 3 may not be required.

The run-in time will tend to move the initial guesses towards the steady state of the system, and can be useful for initial guesses that are a good approximation to the true initial state, but which have small internal inconsistencies.

Not all networks will have a steady state solution. However, the use of a longer run-in time will make it easier for the Transient module to determine the steady state of the system. If the run-in time is long enough then the system will usually settle into a steady state anyway, eliminating the need for the initial guess calculation.

Initial guess file

In systems without a unique steady-state solution, or a complex history that makes the steady-state solution an inappropriate starting point, the user can specify an initial guess file. The Transient module automatically generates a template for this file based on the network under study. The template file simplifies the procedure for inputting the initial guesses since:

1. It is already in the correct format to be read in.
2. It provides a complete list of all variables that have to be specified.
3. It provides suggested values for each variable, generated automatically by the Transient module, suitable for use as defaults.

The initial guess file can be edited with the Initial Guess File Editor which is accessible via the [Initial State dialog](#).

18.11 Elastic Pipe

Pipes are assumed to be of uniform circular cross-section. The properties of the pipe and bulk modulus of the fluid determine the fluid [wavespeed](#) (i.e. the speed of propagation of pressure transients) in the pipe.

In the [Spray option](#) the pipe type list contains the built-in schedules and the available fittings are the 9 [built-in](#) fittings. If the [Hazen-Williams](#) equation is being used then the roughness is replaced by C-factor.

1. User Input Parameters.
2. Diameter - Pipe internal diameter.
3. Length - Pipe length.
4. Net Height Change - When using a constant gradient pipe, this is the difference in height of the pipe outlet from the pipe inlet (must be no greater than length).
5. Roughness or C-Factor - internal surface roughness in the [Standard option](#), C-factor in the [Spray option](#).
7. Type - Select whether the pipe is of constant gradient or a long section.
8. Pipe Type Select a user-defined (or in the Spray option [built-in](#)) pipe type.
9. Fittings - Lists the available [fittings](#). Insert or Delete fittings from the pipe as required.
10. Available VHL - Specify an additional k-factor on the pipe ([Standard option](#) only).

Modelling Equations

Fluid Modelling Equations

The [momentum and continuity equations](#) are solved for pressures and flow rates along the pipe:

Momentum equation

$$\frac{1}{\rho} \frac{\delta p}{\delta x} + \frac{\delta u}{\delta t} + g \sin \alpha + \frac{4f}{d} \frac{u|u|}{2} = 0$$

Continuity equation

$$\frac{1}{\rho A} u \frac{\delta \rho A}{\delta x} + \frac{1}{\rho A} \frac{\delta \rho A}{\delta t} + \frac{\delta u}{\delta x} = 0$$

Frictional Pressure Loss in Standard option

In the [Standard option](#) the pressure drop due to friction effects is given either by the Coulson-Richardson (Darcy) equation, or by the Colebrook-White equation.

Coulson & Richardson

This equation is derived by comparing experimental results for pressure in pipes with theoretical results obtained using the [Coulson and Richardson](#) equation (which ignores frictional effects).

The equation is

$$P_f = \frac{2fL}{D} \rho u^2$$

where:

P_f is the pressure drop.

u is the fluid velocity.

f is the Fanning friction factor.

L is the pipe length.

D is the pipe internal diameter.

ρ is the fluid density.

The Fanning friction factor is a function of the Reynolds number and the relative pipe roughness. PIPENET uses the following empirical formulae for the friction factor:

Laminar flow ($Re < 2000$):

$$f = \frac{16}{Re}$$

Turbulent flow ($Re > 3000$)

$$\frac{1}{\sqrt{f}} = -1.7681 \ln \left(\frac{0.27r}{D} + \frac{1.252}{Re\sqrt{f}} \right)$$

Transitional flow:

f is found by interpolating between the laminar value at $Re = 2000$ and the turbulent value at $Re = 3000$.

Colebrook-White equation

PIPENET can also use an alternative formulation of friction loss using the Colebrook-White equation:

$$1/\sqrt{f} = -4 \log(r/3.7D + 1.256/Re\sqrt{f})$$

The pressure drop caused by the difference in elevation of the two ends of the 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

Frictional Pressure Loss in SPRAY option

In the [Spray option](#) the User has the choice of the [Coulson and Richardson](#) method above or the [Hazen-Williams method](#) for calculating pressure loss. The Hazen-Williams equation is an empirical formula giving the following expression for frictional pressure loss:

SI units

$$P_f = \frac{6.05 \times 10^5 L Q^{1.85}}{C^{1.85} D^{4.87}}$$

where

P_f is the frictional pressure drop.

Q is the volumetric fluid flow rate.

L is the pipe length.

D is the pipe internal diameter.

C is the C-factor (or Hazen-Williams constant) for the pipe.

Imperial units

$$P_f = \frac{4.52 L Q^{1.85}}{C^{1.85} D^{4.87}} ;$$

where:

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 UK gallons per minute.

D is the pipe diameter in inches.

C is the Hazen-Williams constant (or C-factor) for the pipe.

18.12 Short Pipe

The Short Pipe is used to model incompressible flow through a rigid pipe. Using a short pipe as a replacement for an elastic pipe can offer great savings in computational time since the pipe length determines the [calculation time step](#).

Pressure transients are assumed to travel across the short pipe instantaneously. Consequently the short pipe should only be used to replace an elastic pipe if the user wishes to ignore transient behaviour in the pipe.

User Input Parameters

1. Diameter - short pipe internal diameter.
2. Length - short pipe length.
3. Elevation - short pipe elevation (must be no greater than length).
4. Roughness or C-Factor - internal surface roughness in the [Standard](#) option or [Spray](#) option, C-factor in the Spray option.

Modelling Equations

The short pipe uses a 'lumped inertia' model for the fluid in the pipe. Pressure transients then travel across the pipe instantaneously, and the momentum equation for fluid in the short pipe may be written:

$$\frac{dQ}{dt} = \frac{A}{\rho L} (P_1 - P_2) - \frac{2f}{AD} Q|Q| - Ag \sin \alpha$$

where:

- Q is the flow rate through pipe.
- P_1 is the inlet pressure.
- P_2 is the outlet pressure.
- A is the cross-sectional area.
- L is the length.
- D is the diameter.
- α is the angle inclination from horizontal.
- f is the friction factor.
- g is the acceleration due to gravity,
- ρ is the fluid density.

18.13 Pipe Bundle

The Pipe bundle model simulates a bundle of parallel pipes with the same diameters and lengths. Generally it is used to model tube-type heat exchangers and condensers, including both normal and emergency (break or leak) states. The tube side is liquid and shell side is gas or vapour. The pressure in the tube side is higher than the pressure in the shell side. The input additional K factor considers the resistance of the fittings on the tubes (e.g. bends, valves etc) except that the entrance and exit which have been included in the model default.

User Input Parameters

1. Number of tubes:
2. Inside diameter of each tube:
3. Length of each tube
4. Net height change - this is the difference in height of the pipe outlet from the pipe inlet (must be no greater than length)
5. Roughness or C-factor: roughness in Standard option or Spray option, C-factor only in Spray option.
6. Additional K factor: the resistance of the fittings on the tubes except the entrance

and exit.

7. Status: normal, break or leak

Normal status: no extra parameters required.

Leak status:

- Leak area:
- Leak time: the time starts to break.
- Developing time: the period between when the leak starts and becomes fully developed.
- Back pressure: the outside pressure of the tubes, e.g. shell side pressure.

Break status:

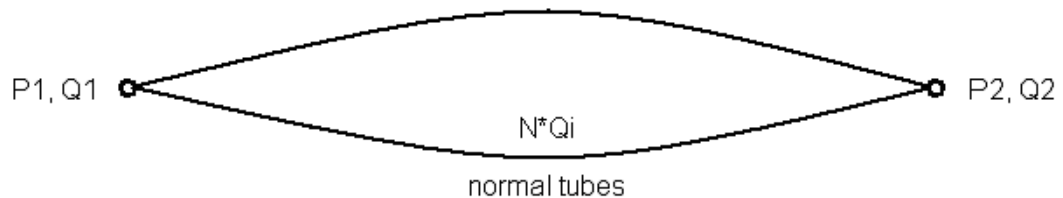
- Break number: a broken tube (break number is one) has two break sections.
- Break time: the time starts to break.
- Developing time: the period between the break starts and fully developed.
- Back pressure: the outside pressure of the tubes, e.g. shell side pressure.

Block status:

- Block number:
- Block time: the time to block.
- Developing time: the period between when the block starts and when it becomes fully developed.

Modelling equations

Normal status



Momentum equation:

$$P_1 - P_2 = \rho g z + \frac{\rho L}{A} \frac{dQ_i}{dt} + \frac{2f\rho L}{DA^2} Q_i |Q_i|$$

$$P_0 = 0.5(P_1 + P_2)$$

Flow-rate equation:

$$Q_1 = Q_2$$

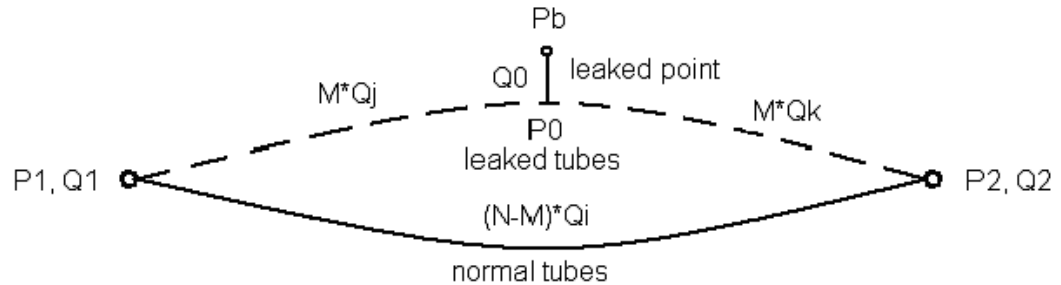
$$Q_1 = NQ_i$$

$$Q_j = Q_i$$

$$Q_k = Q_i$$

$$Q_0 = 0$$

Leak status (the leak position is assumed at the middle of the tube)



Momentum equation:

$$P_1 - P_2 = \rho g z + \frac{\rho L}{A} \frac{dQ_i}{dt} + \frac{2f\rho L}{DA^2} Q_i |Q_i|$$

$$P_1 - P_0 = 0.5 \left(\rho g z + \frac{\rho L}{A} \frac{dQ_j}{dt} + \frac{2f\rho L}{DA^2} Q_j |Q_j| \right)$$

$$P_0 - P_2 = 0.5 \left(\rho g z + \frac{\rho L}{A} \frac{dQ_k}{dt} + \frac{2f\rho L}{DA^2} Q_k |Q_k| \right)$$

$$P_0 - P_b = \frac{k\rho Q_0 |Q_0|}{2(SMA)^2}$$

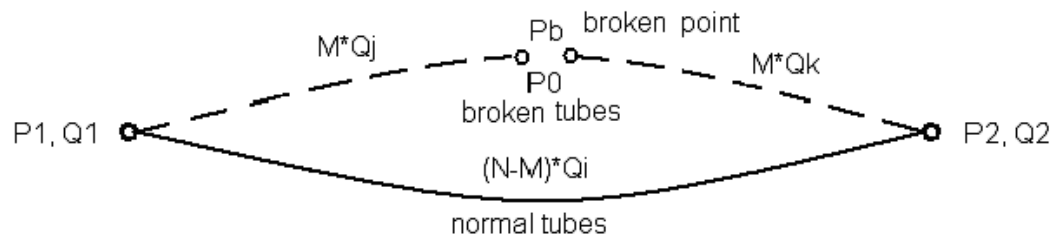
Flow-rate equation:

$$Q_1 = Q_2 + Q_0$$

$$Q_1 = (N - M)Q_i + M Q_j$$

$$Q_2 = (N - M)Q_i + M Q_k$$

Break status (the break position is assumed at the middle of the tube)



Momentum equation:

$$P_1 - P_2 = \rho g z + \frac{\rho L}{A} \frac{dQ_i}{dt} + \frac{2f\rho L}{DA^2} Q_i |Q_i|$$

$$P_1 - P_b = 0.5 \left(\rho g z + \frac{\rho L}{A} \frac{dQ_j}{dt} + \frac{2f\rho L}{DA^2} Q_j |Q_j| \right)$$

$$P_b - P_2 = 0.5 \left(\rho g z + \frac{\rho L}{A} \frac{dQ_k}{dt} + \frac{2f\rho L}{DA^2} Q_k |Q_k| \right)$$

$$P_0 - P_b = 0$$

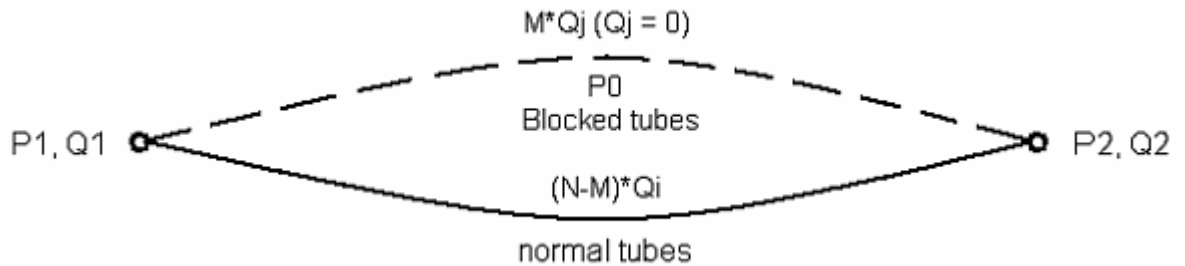
Flow-rate equation:

$$Q_1 = Q_2 + Q_0$$

$$Q_1 = (N - M)Q_i + MQ_j$$

$$Q_2 = (N - M)Q_i + MQ_k$$

Block Status



Part block

Momentum equation:

$$P_1 - P_2 = \rho g z + \frac{\rho L}{A} \frac{dQ_i}{dt} + \frac{2f\rho L}{DA^2} Q_i |Q_i|$$

$$P_1 - P_2 = \rho g z + \frac{\rho L}{A_j} \frac{dQ_j}{dt} + \frac{2f\rho L}{D_j A_j^2} Q_j |Q_j|$$

$$P_0 = 0.5(P_1 + P_2)$$

Flow rate equation:

$$Q_1 = Q_2$$

$$Q_1 = (N - M)Q_i + MQ_j$$

$$Q_k = Q_j$$

$$Q_0 = 0$$

Full block

Momentum equation:

$$P_1 - P_2 = \rho g z + \frac{\rho L}{A} \frac{dQ_i}{dt} + \frac{2f\rho L}{DA^2} Q_i |Q_i|$$

$$P_0 = 0.5(P_1 + P_2)$$

Flow rate equation:

$$Q_1 = Q_2$$

$$Q_1 = (N - M)Q_i$$

$$Q_j = 0$$

$$Q_k = 0$$

$$Q_0 = 0$$

where:-

A	is the cross section area
A_j	is the cross section area during pipe block
D	is the diameter
D_j	is the diameter during pipe block
f	is the friction factor
g	is gravity acceleration
L	is the length
M	is the number of the broken/leaked tubes
N	is the total number of the tubes
P₀	is the inner pressure at the broken/leaked point
P₁	is the inlet pressure
P₂	is the outlet pressure
P_b	is the outer/back pressure at the broken/leaked point
Q₀	is the fluid flow rate escaped
Q₁	is the inlet flow rate
Q₂	is the outlet flow rate
Q_i	is the flow rate in the normal tubes
Q_j	is the inlet flow rate in the broken/leaked tubes

Q_k	is the outlet flow rate in the broken/leaked tubes
t	is the time
z	is the net height change
ρ	is the fluid density

18.14 Compressible Pipe

A Compressible Flow Pipe is a pipe initially containing air at ambient temperature. It is typically used to simulate the first path of a shock wave caused by a sudden increase in pressure at the inlet of the pipe. Such a situation occurs when a pipe is attached to a pressure relief valve, which can open very rapidly.

This can be simulated by setting the pressure at the inlet of the Compressible Flow Pipe to that at the outlet of the pressure relief valve.

It is often of interest to compute the forces acting on the pipe due to the shock wave. Note that a compressible flow pipe can have bends. The flow is assumed to be frictionless, so forces on straight parts of the pipe are zero.

The Compressible Flow Pipe differs from all other PIPENET models in a number of ways:

- The maximum number of Compressible Flow Pipes is one.
- It must be connected to pressure specifications at both ends.
- The pressure specified at the inlet must be greater than the pressure specified at the outlet.
- It is assumed that there is a step change in the inlet pressure at time $t = 0$. The inlet pressure is stepped from the value specified at the outlet at $t = 0$ to the value specified the inlet at $t = 0$. After this step change the pressure at both the inlet and outlet are kept constant at those values, irrespective of changes in the pressure specifications. Therefore, only constant pressure specifications should be given at both ends.
- An analytical solution of the first path of a shock wave is computed.
- The simulation automatically stops when the shock wave reaches the outlet.

In addition to the pressure specification at the inlet, the temperature must also be specified at this point. This is done using the Compressible Flow Pipe Dialog. The temperature at the outlet of the pipe and in the pipe at the start of the simulation is assumed to be the ambient temperature, which can be set.

User Input Parameters

1. Diameter - compressible flow pipe internal diameter.
2. Length - compressible flow pipe length.
3. Number of sections - required for definition and resolution of output graphs and forces.

4. Temperature - temperature at the inlet of the compressible flow pipe.

Modelling Equations

The flow in the pipe is a solution of the Rankine-Hugeniot relation, assuming adiabatic frictionless flow.

18.15 Fittings and equipment items

Pipe fittings such as bends, tee-pieces, valves and filters cause an additional frictional pressure drop across the pipe. Because PIPENET treats fittings as being associated with individual pipes a large number of fittings can easily be incorporated into a network.

Fittings in the [Spray option](#) are required to comply with fire protection standards and so are modelled differently to fittings in the [Standard option](#).

User Input Parameters

Standard option

- Insert or Delete any number of the available fittings on a particular pipe.
- Specify an additional k-factor for the pipe in Additional k-factor.

Spray option

For an equipment item, specify the following:

- Description (up to 10 characters).
- Equivalent length of the pipe.

Modelling Equations

Standard option

In the [Standard option](#) the frictional pressure drop across a fitting is given by:

$$P = \frac{1}{2} k \rho u^2$$

where:

P is the pressure drop.

k is the k-factor for the fitting.

ρ is the density of the fluid.

u is the fluid velocity through the fitting.

[Approximate k-factor values](#) for some common fittings types are given in the [Reference section](#).

Spray option

In the [Spray option](#) fittings are incorporated as equivalent lengths of straight pipe. Adding a fitting to a particular pipe is equivalent to extending the pipe's length by an amount that depends on the fitting type. However the inclusion of an equivalent length does not alter the actual length of the pipe and the time for a

pressure wave to travel the pipe length remains the same. Equivalent lengths are provided for the nine [built-in](#) Spray fittings, for both [FOC](#) and [NFPA](#) rules.

Placement of Tees in the Spray option

For the placement of crosses and tees the NFPA guidelines [\[NFPA\]](#) pages 13-213 to 13-216 provides the following information

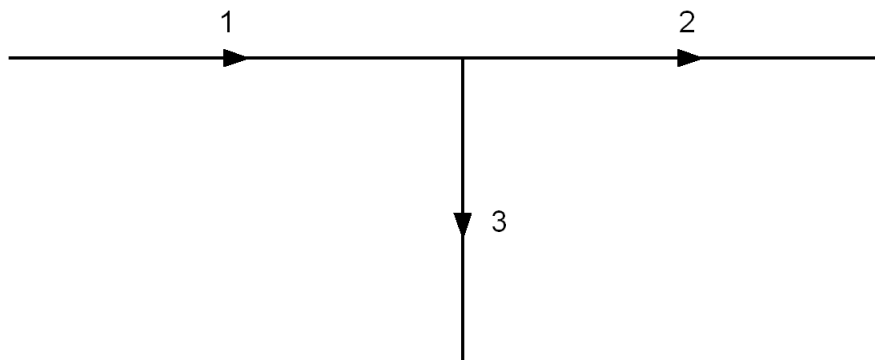
- 22.4.4.7 (3) Calculate the loss for a tee or a cross where flow direction change occurs based on the equivalent pipe length of the piping segment in which the fitting is included.
- 22.4.4.7 (4) The tee at the top of a riser nipper shall be included in the branch line, the tee at the base of a riser 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.

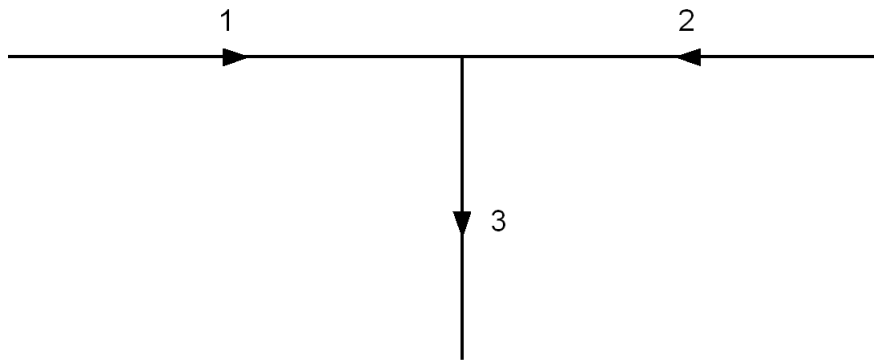
Placement of Tees in the Standard option

There is no hard-and-fast rule to govern the placement of fittings on pipes which meet at a tee-junction, but many engineers would consider it good practice to add a tee (branch) where the flow changes direction, and a tee (run) where the flow direction is unchanged. This is illustrated below for each of the cases in which three flows meet at a tee-junction.

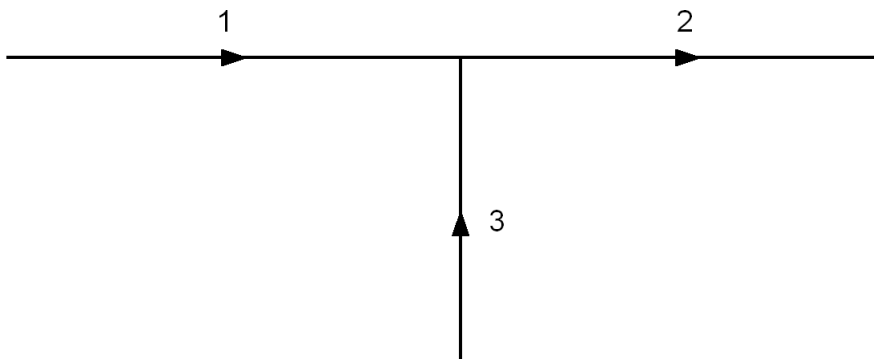
Example 1: tee(run) on pipe 2; tee(branch) on pipe 3.



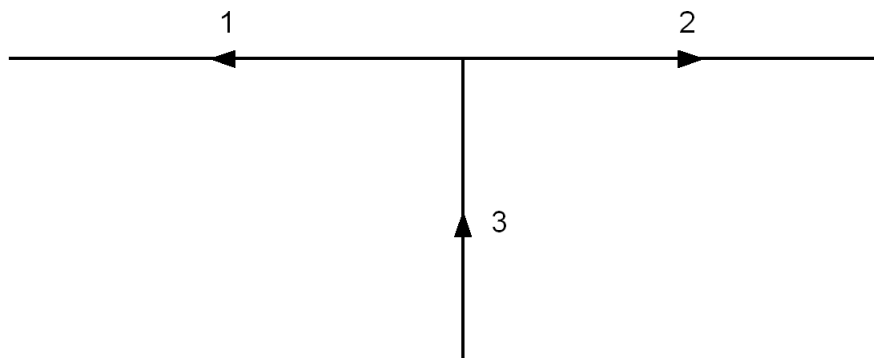
Example 2: tee(branch) on pipe 1; tee(branch) on pipe 2.



Example 3: tee(run) on pipe 1; tee(branch) on pipe 3.



Example 4: tee(branch) on pipe 1; tee(branch) on pipe 2.



18.16 Simple Pump

A simple 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 simple pumps (either with [unknown coefficients](#) or [known coefficients](#)).

Modelling Equations

The pressure increase produced by the pump is:

$$\Delta P = A Q^3 / s + B Q^2 + C s Q + D s^2$$

where:

s is the pump setting.

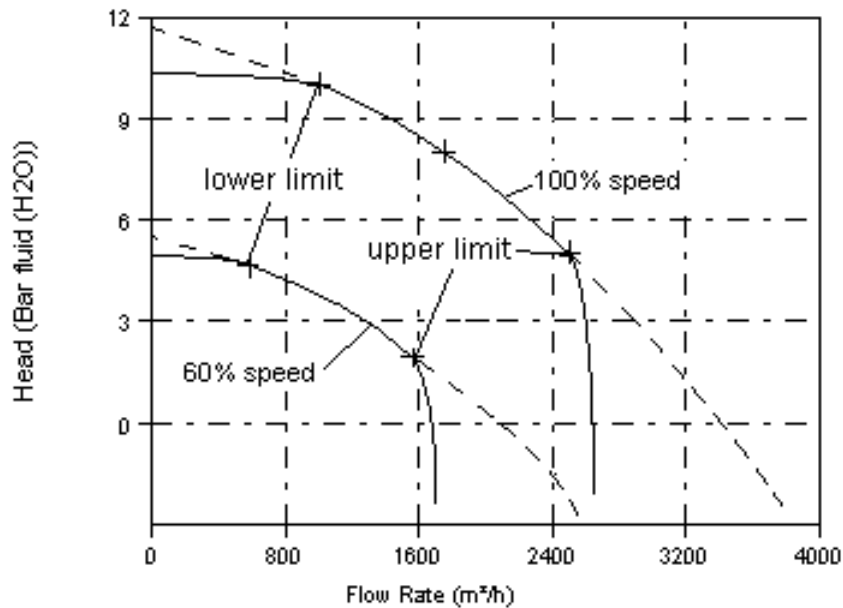
Q is the flow rate through the pump.

A, B, C, D are pump performance coefficients.

The pump performance coefficients A, B, C and D 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:

$$P = A Q^2 + B s Q + C s^2$$

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:

$$0 \leq n \leq 10$$

$$\alpha = 2A Q_{LL} + B s$$

$$\beta = A Q_{LL}^2 + B s Q_{LL} + C s^2$$

$$Q_{LL} = Q_{min} s$$

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..

$$\Delta P = A Q^2 + B s Q + C s^2 - (2^m - 1) |A| (Q - Q_{UL})^2$$

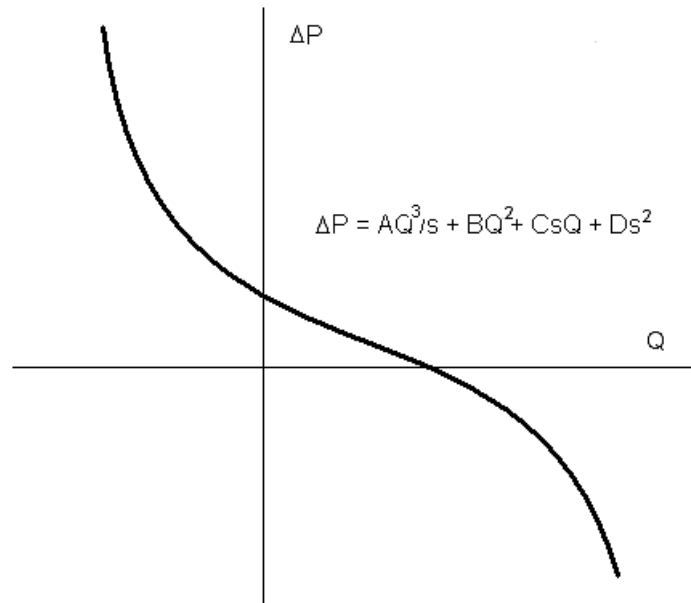
where:

$$0 \leq m \leq 10$$

$$Q_{UL} = Q_{max} s$$

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.

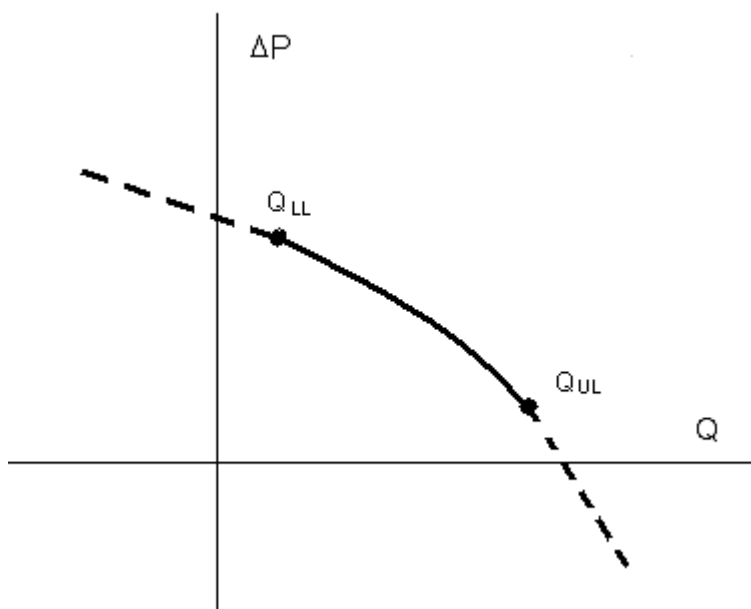


There is a potential problem with the cubic curve in that it may lead to multiple solutions. Therefore, the above function must satisfy the following condition:

$$B^2 - 3AC \leq 0$$

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:

$$Q_{UL} = Q_{max} s$$

$$\alpha = 3A_n Q_{UL}^2 / s + 2B_n Q_{UL} + C_n s$$

$$\beta = A_n Q_{UL}^3 / s + B_n Q_{UL}^2 + C_n s Q_{UL} + D_n s^2$$

18.17 Turbo Pump

The Turbo Pump provides an increase in pressure and can also handle the 'spin down' due to pump failure.

User Input Parameters

1. **Pump Type** - selected from a list of defined [library](#) turbo pumps.
2. **Pump Failure** - define a time at which the pump fails and starts to spin down.

Modelling Equations

Suter Characteristic Curves

The Turbo Pump is defined by a pair of Suter Characteristic Curves. The curves define pressure head and torque as a function of flow rate and pump speed over all possible operating conditions. To derive the Suter curves the pressure head, flow rate, speed and

torque are first non-dimensionalised with respect to the rated quantities (the values when the pump is operating at full speed).

$$h = \frac{H}{H_R} = \frac{P}{P_R}$$

$$v = \frac{Q}{Q_R}$$

$$\alpha = \frac{N}{N_R}$$

$$B = \frac{T}{T_R}$$

where:

H is the piezometric head across the turbo pump.

P is the pressure increase across the turbo pump.

Q is the flow rate through the turbo pump.

N is the speed of the turbo pump.

T is the applied torque.

The subscript R denotes the rated quantity of the variable, i.e. the values of H, P, Q, N and T at the point of best efficiency. The head-flow rate and torque-flow rate curves for the turbo pump are then transformed to Suter characteristic curves of the form.

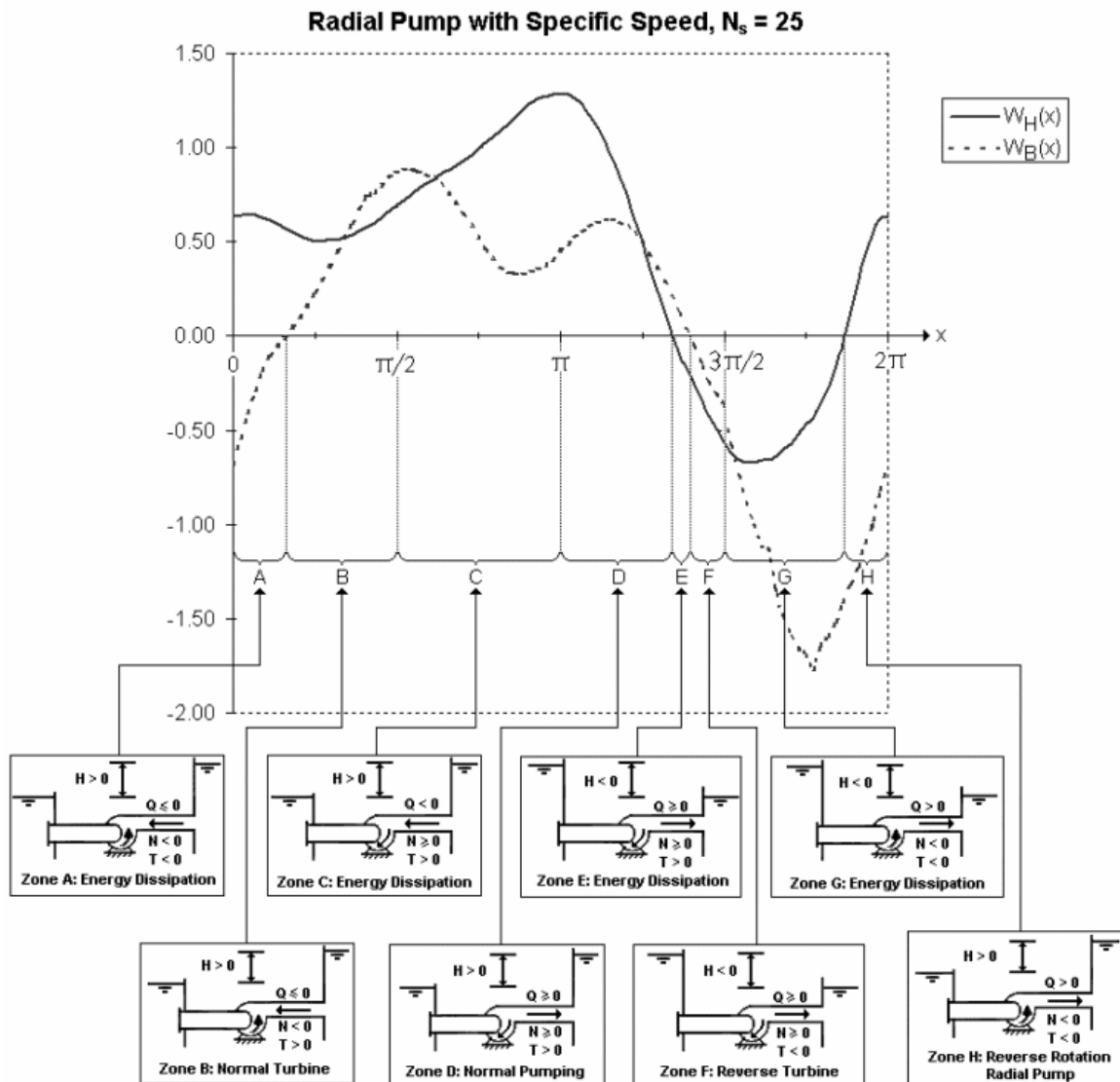
$$W_H(\chi) = \frac{h}{\alpha^2 + v^2} \quad (\text{head curve})$$

$$W_B(\chi) = \frac{B}{\alpha^2 + v^2} \quad (\text{torque curve})$$

where:

$$\chi = \pi + \tan^{-1} \frac{v}{\alpha}$$

A typical plot of the Suter curves is shown below:



The figure shows typical Suter curves for a Radial Pump. The regions referred to in the figure are termed as Zones and Quadrants. Each quadrant is of length $\pi/2$ and the zones lying therein are split at zero head-flow rate and torque-flow rate values. There are eight possible zones of pump operation: four occur during normal operation and four are abnormal zones. During a transient event, a pump may enter most, if not all, regions in the figure depending on the appropriate circumstances.

Similar Turbo Pumps are characterized by the specific speed. This is defined as:

$$N_s = \frac{N_R \sqrt{Q_R}}{H_R^{3/4}}$$

Built-in Turbo Pumps

Three built-in Turbo Pumps are available having specific speeds 25 (Radial), 147 (Mixed) and 261 (Axial) for SI units: N_R the rpm, Q_R in m^3/s , H_R in metres.

If the pump specific speed is different to one of the three built in Turbo Pump types then

the Suter curve data should be derived from the head and torque curves.

The Suter curve data (x , $WH(x)$) and (x , $WB(x)$) can be entered in the [turbo pump libraries](#). Each point on the head-flow rate and torque-flow rate is transformed to a point on the $WH(x)$ - x and $WB(x)$ - x curve respectively, using the previous formulae. A reasonable number of points (e.g. over 30) should be entered and data taken from curves from many different pump speeds to give an accurate representation of the pump behaviour.

Pressure Head

From the Suter relations, the pressure increase across the turbo pump is given by:

$$P = P_R h = P_R (\alpha^2 + v^2) W_H(X)$$

Moment of Inertia

If the moment of inertia is not obtainable from the manufacturers, the following equations may be used to provide an estimate for the moment of inertia [\[ARDT\]](#):-

Pump impeller inertia

$$I_P = 1.5 \times 10^7 \times \left(\frac{P_R}{N_R^3} \right)^{0.9556}$$

Pump motor inertia

$$I_M = 118 \times \left(\frac{P_R}{N_R} \right)^{1.48}$$

Entrained fluid inertia

$$I_W = \frac{\pi}{2} \rho H_P R_R^4$$

Total moment of inertia

$$I = I_P + I_M + I_W$$

where

- H_P width of pump impeller chamber, m
- I total moment of inertia, including pump, entrained fluid and motor, kg.m²
- I_M motor inertia, kg.m²
- I_P pump and the entrained fluid inertia, kg.m²
- I_W entrained fluid inertia, kg.m²
- N_R rated rotate speed, rpm

P_R	rated shaft power, kW
RP	rotating radius of pump impeller, m
ρ	liquid density, kg/m ³

Pump Failures

The user can specify a trip time at which the turbo pump loses all power and spins down through the remainder of the simulation. The specification associated with the turbo pump information node is overridden for all times after the trip time.

As the turbo pump spins down a torque is applied to the drive shaft. The equation for turbo pump spin down is therefore:

$$T = -I \frac{dw}{dt}$$

$$I = MR^2$$

$$w = 2\pi N$$

where:

- I is the moment of inertia of the turbo pump.
- M is the total mass of the rotating parts and entrained fluid.
- R is the radius of gyration of the rotating mass.
- N is the speed of the turbo pump.
- w is the angular velocity.

The modelling equations presented here are based on a linear regression analysis of data presented in [\[ARDT\]](#)

18.18 Inertial Pump

The starting and stopping of a pump are two important sources of transients in a pipeline network. The [turbo pump model](#) can predict transient behaviour during pump trip for all four quadrants, however, Suter curves are not available for most pumps. With the [simple pump model](#) the pump's performance $H-Q$ (head-flowrate) curve is required, but the start-up/trip time has to be estimated and this can be unreliable and inaccurate. The **inertial pump** seeks to address these problems.

The inertial pump model is designed to predict a pump's steady/transient behaviour in the first quadrant, i.e. the working region. The required parameters are fewer and more accessible compared with the turbo pump. In summary, the inertial pump works like a simple pump at the steady state but can simulate transient behaviour during start-up and stopping.

Inertial pump library

Before they can be used an inertial pump must first be defined as a pump library item via the **Libraries | Inertial pumps** menu option. As with other library items this will result in the display of a dialog, but because there are data to be entered for three curves the dialog contains three tabs on the right-hand side of the dialog. the first is for entering the H-Q and P-Q, the second for entering the T-S data and the third for displaying the calculated coefficients for the three curves. The following illustrates the first two of these tabs:

The screenshot shows the 'Local Libraries' dialog box for an 'Inertia Pump'. The dialog is divided into several sections:

- Name:** Inertia Pump
- Description:** Inertia Pump
- Source:** Local user library
- Motor characteristics:**
 - Model type: Both
 - Moment of inertia (kg m²): 11530
 - Rated speed (rpm): 370
 - Motor synchronous speed (rpm): 375
- Fluid property:**
 - Ref. Fluid Density (kg/m³): 998.2343
- Working range - flow rate:**
 - Min: 0, Max: 20
 - Degeneration factor: 0
- Units:**
 - Flowrate: m³/sec, Head: Pascals, Power: W, Torque: Nm

A table displays the following data:

Flow rate	Head	Efficiency	Power	Head (curve)	Head (curve)
m ³ /sec	Pa fluid (reference)	%	W	Pa fluid (reference)	Pa fluid (reference)
0	799930	0	5006750	799930	799930
1.389	756668	20.502	5126504	756668	756668
2.778	715400	38.005	5229270	715400	715400
4.167	670137	51.003	5475072	670137	670137
5.556	620835	62.01	5562545	620835	620835
6.944	585575	70.993	5727626	585575	585575

A graph plots Pressure / Pa fluid (reference) and Power / W against Flow Rate / m³/sec. The graph shows a smooth curve for the H-Q relationship and a smooth curve for the P-Q relationship. The H-Q curve starts at 799930 Pa at 0 m³/sec and decreases to 585575 Pa at 6.944 m³/sec. The P-Q curve starts at 5006750 W at 0 m³/sec and increases to 5727626 W at 6.944 m³/sec.

The dialog also includes tabs for 'Head and power data', 'Torque and relative speed data', and 'Calculated coefficients'. The 'Head and power data' tab is currently selected.

Relative speed	Rotation speed rpm	Motor Torque Nm
1	370	150000
0.9	333	170000
0.8	296	188000
0.7	259	204000
0.6	222	218000
0.5	185	230000
0.4	148	240000

On the left-hand side of the dialog the following information must be provide:-

- **Pump name**
- **Pump description**
- **Model type** - select from **Trip**, **Start-up** or **Both**
- **Moment of inertia** - the total rotation moment of the pump, motor and entrained fluid
- **Rated speed** - the recommended working speed
- **Motor synchronous speed** - motor driving torque is zero at synchronous speed, only required for a Start-up type pump
- **Reference. fluid density** - the fluid density that the pump curves are based on.
- **Min/max flowrate** - the recommended working range.
- **Degeneration factor** - used to correct pump performance curve (flow rate vs. head) beyond the working range.
- **Units** - select from the available options for the units to be used in defining the curves; Flow rate, Head, Power and Torque (torque unit is required only for start-up type pump).

Via the **Head and power data** tab provide:-

- The flow rate and head values defining the H-Q curve
- The pump input shaft power, defining the P-Q curve. If power data are not available, the power curve can be deduced from flow rate – efficiency curve

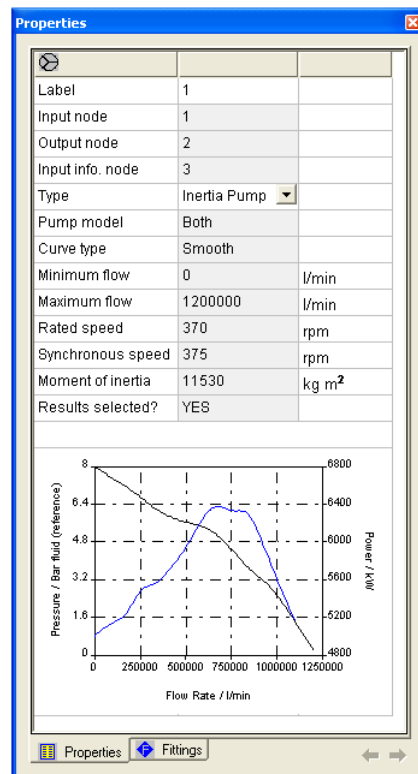
except for the power at zero flow rate, this must be entered manually. To select which is entered power (default) or efficiency use the **Known** combo box in the bottom right-hand corner of the dialog.

Via the **Torque and relative speed** tab provide (not required for the Trip model pump):-

- The torque and relative speed data defining the T-S curve. Relative speeds are relative to the rated speed entered on the left-hand side of the dialog. If only the actual speeds are known then these can be entered instead by selecting the **Rotation speed** option in the **Known** combo box in the right-hand corner of the dialog,

Inertial pump properties

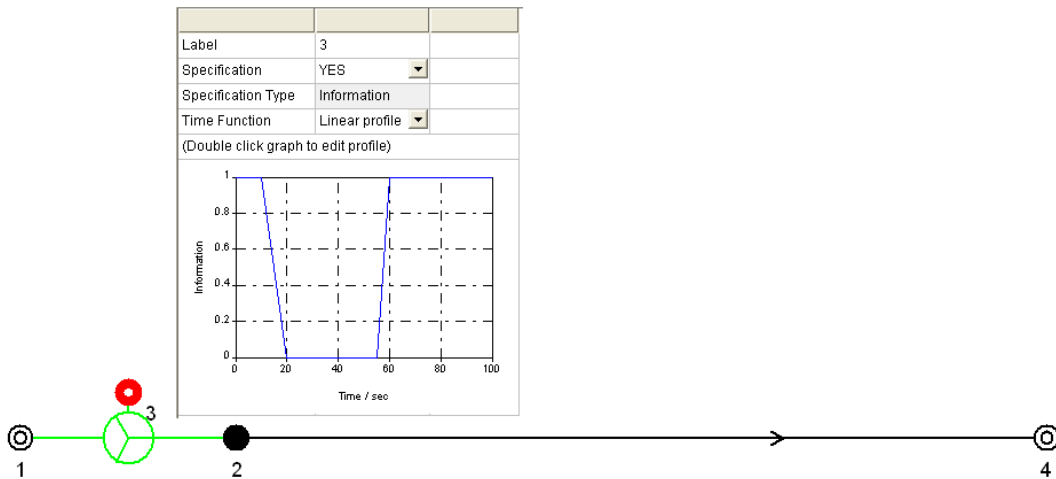
When one or more library inertial pumps have been defined, selecting an inertial pump in the schematic window displays the following pump properties;_



Pump control

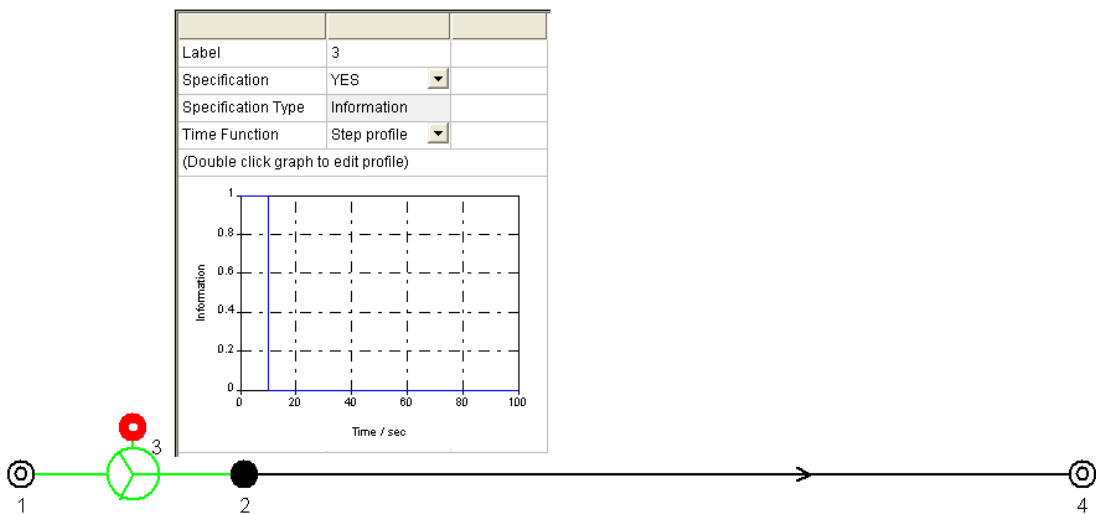
Normal state

The inertial pump works like simple pump at normal state. the pump speed is controlled via the information node of the inertial pump.



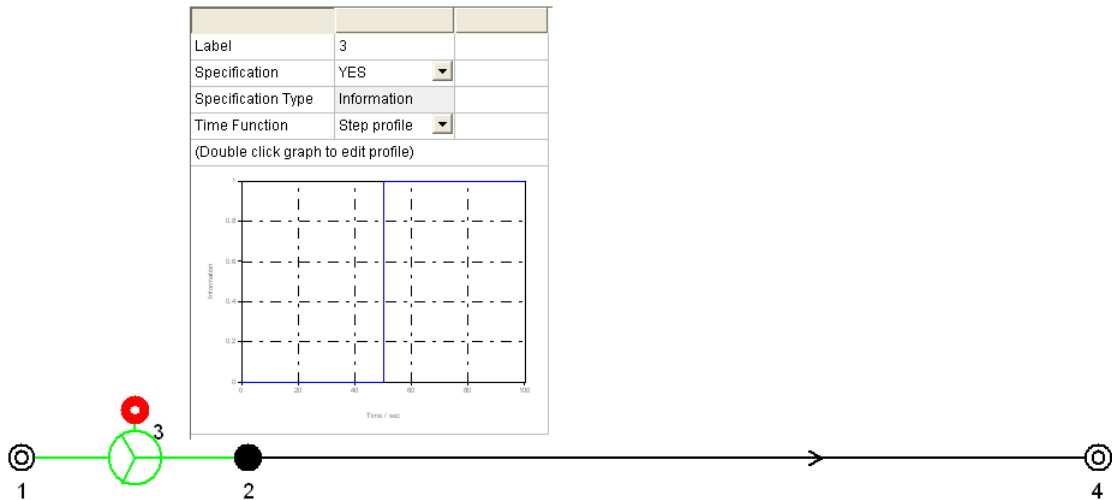
Pump trip

The inertial pump can also simulate pump trip like the turbo pump does. The difference is that the turbo pump sets trip time in pump property window whilst the inertial pump sets trip time using step type time function via the information node and the final setting must be zero.



Pump start-up

The pump model type must be Start-up or Both in pump library. As same as pump trip, inertial pump sets start time using step type time function via the information node and the final setting must be 1.0.



Modelling equations

Normal state

The fluid head increase produced by the pump is:-

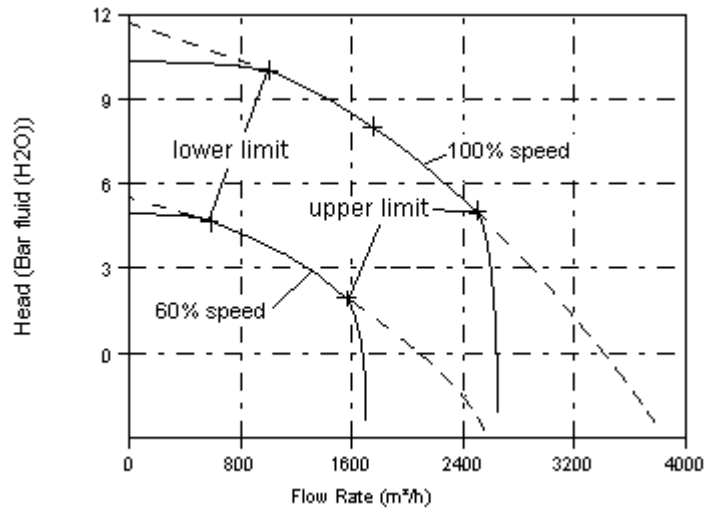
$$H = A Q^3 / s + B Q^2 + C s Q + D s^2$$

where:

- H** is pressure increase (head).
- s** is pump setting.
- Q** is flow rate through the pump.
- A, B, C, D** are pump performance coefficients.

The pump performance coefficients **A**, **B**, **C** and **D** 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 valve setting, **s**, must be between 0 and 1. A setting of $s = 0$ represents a shut down pump, while $s = 1$ represents a pump at full rated speed.



Pump Trip

The pump spin down rate can be calculated based on power curve.

$$P = -I \omega \frac{d\omega}{dt}$$

where

- P** is pump input shaft power.
- I** is total inertia moment.
- ω is angular velocity.

If pump power curve P-Q is unavailable but pump efficiency curve H-Q is available, the P-Q curve can be deduced from the H-Q curve

$$P = \frac{\rho_{ref} g H Q}{\eta_P}$$

where

- P** is pump input shaft power.
- H** is pressure increase (head).
- Q** is flow rate through the pump.
- ρ_{ref} is reference fluid density.
- η_P is pump efficiency.

Pump Startup

The pump spin-up rate during start-up can be calculated based on the difference of

motor driving torque and pump resistance torque.

$$T_D - T_R = I \frac{d\omega}{dt}$$

$$T_R = \frac{P}{\omega}$$

where

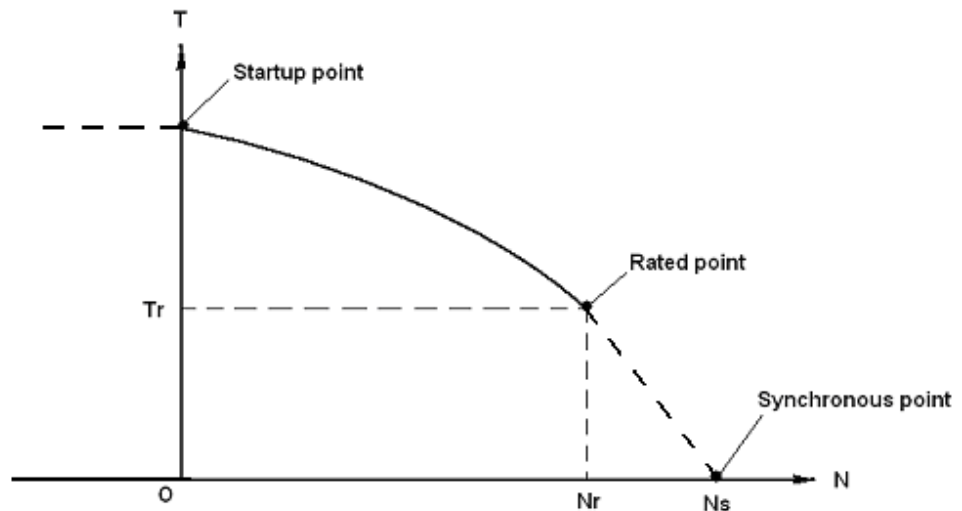
T_D is motor driving torque, is zero at synchronous speed.

T_R is resistance torque from pump.

P is pump input shaft power.

I is total inertia moment.

ω is angular velocity.



Moment of Inertia

If the moment of inertia is not obtainable from the manufacturers, the following equations may be used to provide an estimate for the moment of inertia [\[ARDT\]](#):-

Pump impeller inertia

$$I_P = 1.5 \times 10^7 \left(\frac{P_R}{N_R^3} \right)^{0.9556}$$

Pump motor inertia

$$I_M = 118 \left(\frac{P_R}{N_R} \right)^{1.48}$$

Entrained fluid inertia

$$I_W = \frac{\pi}{2} \rho H_P R_R^4$$

Total moment of inertia

$$I = I_P + I_M + I_W$$

where

H_P	width of pump impeller chamber, m
I	total moment of inertia, including pump, entrained fluid and motor, kg.m ²
I_M	motor inertia, kg.m ²
I_P	pump and the entrained fluid inertia, kg.m ²
I_W	entrained fluid inertia, kg.m ²
N_R	rated rotate speed, rpm
P_R	rated shaft power, kW
R_R	rotating radius of pump impeller, m
ρ	liquid density, kg/m ³

18.19 Valves

Valves are components that have a variable resistance to flow, causing a pressure drop across the valve. In PIPENET a wide range of valves are available to the user:

- [Operating Valves](#)
- [Non-return Valves](#)
- [Check Valves](#)
- [Fluid Damped Check Valve](#)
- [Liquid Surge Relief Valve](#)
- [Regulator Valve](#)
- [Inertial Check Valve](#)
- [Pressure surge relief valve](#)

The pressure drop across the valve as fluid flows through it depends on a number of factors including the flow rate through the valve, the valve characteristic, the valve setting and the physical properties of the valve.

In PIPENET, the valve setting is represented by a number 's' which varies between zero (for a fully closed valve) and one (for a fully open valve), for example:

- s = 0 Valve is fully closed
- s = 0.25 Valve is quarter open
- s = 0.667 Valve is two thirds open
- s = 1 Valve is fully open

Depending on the type of valve the setting may be specified by the User, set by a [control system](#) or calculated based on flow conditions and the physical properties of the valve.

Valve Characteristics

Valve characteristics can be defined in terms of K-Factor or Flow Coefficient.

K-Factor Valve

The pressure drop across an open valve is given by:

$$\frac{1}{2} k \rho Q |Q| = S^2 A^2 (P_1 - P_2)$$

where:

- P_1 is the inlet pressure.
- P_2 is the outlet pressure.
- Q is the flow rate.
- $k(s)$ is the k-factor.
- s is the valve setting.
- A is the valve port area.
- ρ is the fluid density.

The value $k(s)$ is the k-factor of the valve and represents the resistance to flow. The valve characteristic is the curve of k against s and can be defined by the User or as one of the built-in types.

Flow Coefficient Valve

The pressure drop across an open valve is given by:

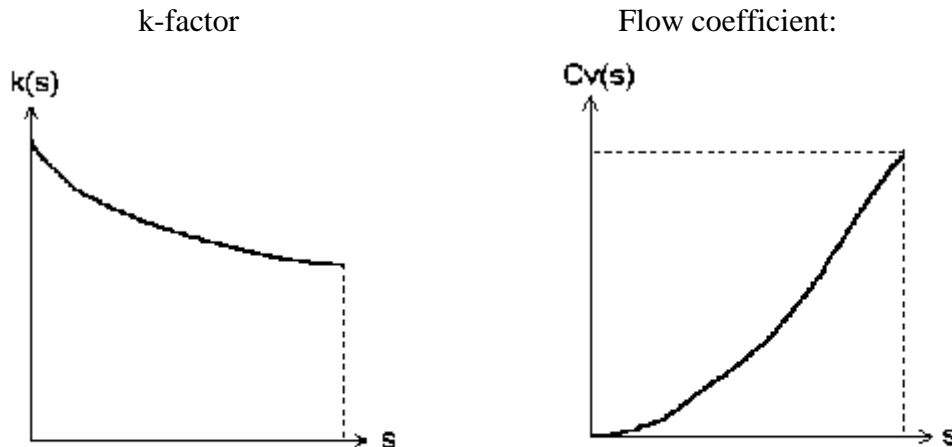
$$P_1 - P_2 = \frac{\rho Q |Q|}{\rho_0 C_v^2}$$

where:

- $C_v(s)$ is the C_v value.
- ρ is the density of the fluid.
- ρ_0 is the density of water at standard conditions (1000 kg/m³).

The value $C_v(s)$ is the flow coefficient of the valve and represents its capacity for flow. The valve characteristic is the curve of C_v against s and can be defined by the User or as one of the built-in types.

Typical representations of the two curves are shown below:



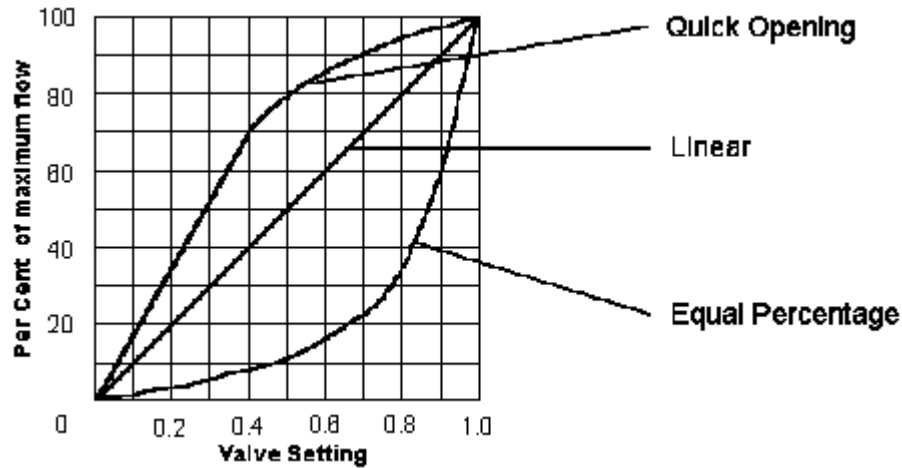
Specifying a Valve Type

Defining a Library Valve Characteristics

The User can specify a valve characteristic curve as a series of data points on the curve in [valve libraries](#). The characteristic curve can be defined for k-factor or C_v valves. Valve characteristics can be entered and saved in a library file. The valve characteristic is then selected in the valve dialog box.

Built-in Valve Characteristics

Three built-in valve characteristics are available to the User. These can be chosen from the valve dialog box as k-factor or C_v valve.



Linear Opening Valve

Flow through the valve is proportional to the valve setting. For example, at a valve setting of 0.8 the flow rate through the valve is 80% of the maximum flow.

Equal Percentage Opening Valve

Equal increments of valve plug travel produce equal percentage increases in the existing flow. Changes in flow rate are small at smaller values of s , but gradually increase as the valve opens.

Quick Opening Valve

Maximum changes in flow rate occur at smaller values of s . As the valve setting increases the changes in flow rate are drastically reduced.

18.20 Operating Valve

Operating Valves are general purpose valves whose setting is specified by the User as an information node specification.

User Input Parameters

Valve Type - select from a list of defined [library](#) valves, or one of the [built-in types](#) (Linear, Equal Percentage or Quick Opening).

If the valve is a built-in type the following information is required:

1. **K-factor Valve** - k-factor value for fully open valve and fully open valve area.
2. **Flow Coefficient Valve** - fully open C_v value.

Modelling Equations

1. Valve Equations

Open k-factor valve in SI units ($s \geq 0$)

$$\frac{1}{2} k \rho Q |Q| = S^2 A^2 (P_1 - P_2)$$

Open flow coefficient (C_v) valve ($s \geq 0$)

$$P_1 - P_2 = \frac{\rho Q |Q|}{\rho_0 C_v^2}$$

Fully closed valve ($s = 0$)

$$Q_1 = 0$$

2. 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.

s is the specified valve setting.

A is the fully-open cross section.

ρ is the density of the fluid.

ρ_0 is the density of water at standard conditions.

18.21 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

1. Valve Equation

Open valve

$$P_1 = P_2$$

Fully closed valve

$$Q_1 = 0$$

2. 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.

18.22 Check Valve

A check valve is a swing-gate type [non-return valve](#). Closure of the valve is not instantaneous but depends on flow and pressure near the valve and the valve's physical characteristics.

User Input Parameters

Valve Type - Select from a list of defined [library](#) valves, or one of the built-in types (Linear, Equal Percentage or Quick Opening)

If the valve is a built-in type the following information is required:

K-factor Valve - k-factor value of fully open valve and fully open valve area.

Flow Coefficient Valve - fully open Cv value.

Physical properties of the check valve (see modelling equation)

Close/Open Time

Reference Setting

Pressure or Flow rate

Trigger

Modelling Equations

1. Valve Equation

Open k-factor valve in SI units ($s \geq 0$)

$$\frac{1}{2} k \rho Q |Q| = S^2 A^2 (P_1 - P_2)$$

Open flow coefficient (C_v) valve ($s \geq 0$)

$$P_1 - P_2 = \frac{\rho Q |Q|}{\rho_0 C_v^2}$$

Fully closed valve ($s = 0$)

$$Q_1 = 0$$

2. 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.

s is the specified valve setting.

A is the fully-open cross section.

ρ is the density of the fluid.

ρ_0 is the density of water at standard conditions.

Check Valve Modelling Equation

The physical characteristics of the check valve are determined by the pressure and flow rate triggers, its reference setting and the close/open time. The pressure trigger P_t is the pressure difference required to keep the valve at its reference setting. Similarly the flow rate trigger Q_t is the flow rate required to keep the valve at its reference setting. The Close/Open Time is a typical response time for the valve clapper to move from closed to fully open. Note that a very small Close/Open time may cause the calculation to fail. This problem can usually be overcome by reducing the calculation time step accordingly or by using the [variable time step facility](#), specified via the [calculation output dialog](#).

The actual check valve setting is given by:

$$\tau \frac{ds}{dt} + s = \bar{s}$$

where:

$$\bar{s} = s_r \frac{P_1 - P_2}{P_r}$$

\bar{s} is the setting of the ideal check valve.

s_r is the reference setting.

P_r is the reference trigger pressure.

τ is the close/open time.

t is the time.

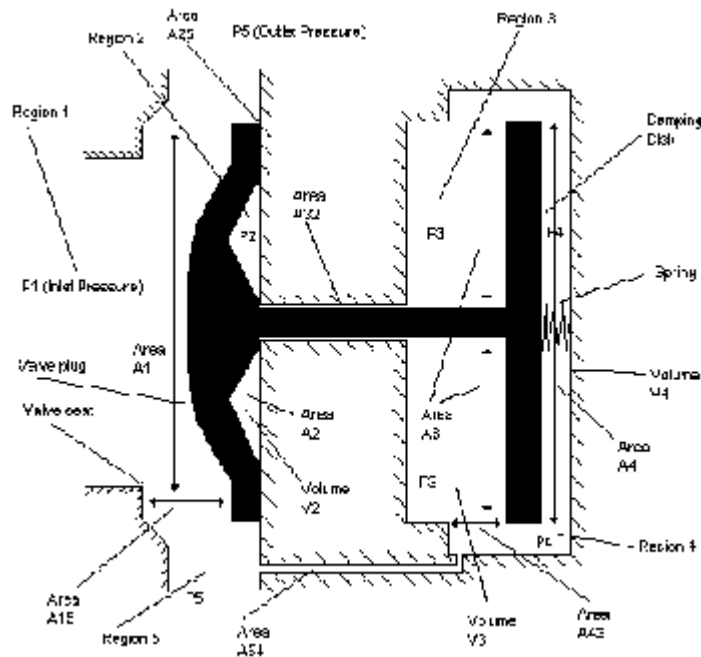
s is the actual setting.

18.23 Fluid Damped Check Valve

A fluid damped check valve is a type of [check valve](#) consisting of a translating disk and plug. The valve setting is determined by hydrodynamic forces acting on the valve plug and damping disk, as well as the elastic damping of the spring situated behind the damping disk.

User Input Parameters

A schematic of the fluid damped check valve is given below:



The various regions are as follows:

1. Inlet.
2. Downstream of valve disk.
3. Upstream of damping disk.
4. Downstream of damping disk.
5. Outlet.

The following parameters are input by the User:

Valve Characteristics - details of the valve characteristic. Can be a [library](#) valve or one of the [built-in valve types](#).

Physical Properties of Valve - details of the physical properties of the valve. The various regions are as indicated in the schematic diagram above.

Modelling Equations

1. Valve Equation

Open k-factor valve ($s \geq 0$)

$$\frac{1}{2} k \rho Q |Q| = S^2 A^2 (P_1 - P_2)$$

Open C-factor valve ($s \geq 0$)

$$P_1 - P_2 = \frac{\rho Q |Q|}{\rho_0 C_v^2}$$

Fully closed valve ($s = 0$)

$$Q_1 = 0$$

2. 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.

s is the specified valve setting.

ρ is the density of the fluid.

ρ_0 is the density of water at standard conditions.

Modelling Equation

The forces due to the weight of the plug, the spring and the hydrodynamic forces govern the motion of the check valve plug. The force due to friction is considered negligible.

Applying Newton's second law we have:

$$M \frac{d^2 \chi}{dt^2} = Mg \sin \Theta - s_k \chi + F_H$$

where the three terms on the right-hand side represent gravity, spring factor and hydrodynamic term, and:

M is the total mass of the valve plug and disk.

s_k is the spring constant.

χ is the displacement.

F_H is the total hydrodynamic force on the valve plug and damping disk.

Θ is the inclination of the valve from the horizontal.

18.24 Liquid Surge Relief Valve

Liquid Surge Relief Valves are used to relieve pressure surges. The relief valve consists of a valve plug situated in front of a spring or equivalent device. When the pressure on the plug increases above a specified set pressure the valve starts to open. The valve becomes fully open when the pressure on the plug is equal to a specified wide-open pressure.

User Input Parameters

Valve Type - select from a list of defined [library](#) valves, or one of the [built-in types](#) (Linear, Equal Percentage or Quick Opening)

If the valve is a [built-in type](#) the following information is required:

K-factor Valve - k-factor value for fully open valve.

Flow Coefficient Valve - fully open Cv value.

Physical properties of the check valve (see modelling equation)

Set Pressure,

Wide Open Pressure

Fully Closed Pressure

Hysteresis

Modelling Equations

1. Valve Equation

Open k-factor valve in SI units ($s \geq 0$)

$$\frac{1}{2} k \rho Q |Q| = S^2 A^2 (P_1 - P_2)$$

Open flow coefficient (Cv) valve ($s \geq 0$)

$$P_1 - P_2 = \frac{\rho Q |Q|}{\rho_0 C_v^2}$$

Fully closed valve ($s = 0$)

$$Q_1 = 0$$

2. 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.

s is the specified valve setting.

A is the fully-open cross section.

ρ is the density of the fluid.

ρ_0 is the density of water at standard conditions.

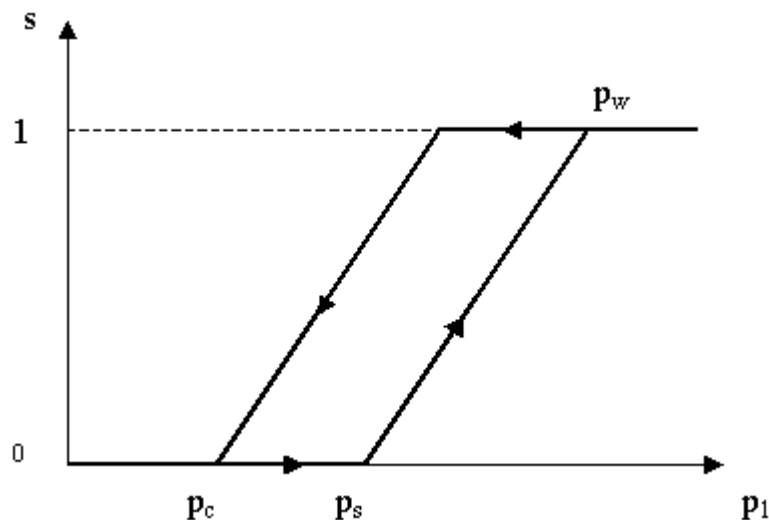
Modelling Equation

The valve remains closed until the inlet pressure is equal to the Set Pressure P_s . While opening, and for pressures between the Set Pressure P_s and the Wide Open Pressure P_w the valve setting is given by:

$$s = \frac{P_1 - P_s}{P_w - P_s}$$

For inlet pressures greater than the wide-open pressure the valve is fully open.

If the Hysteresis Check Box is ticked in the component properties, then the valve becomes fully closed again if the inlet pressure falls below the Fully Closed Pressure P_c . The valve retains its setting if the inlet pressure starts falling before it has reached the Wide Open Pressure P_w until it reaches the downward branch of the hysteresis loop. The graph below shows the setting of the valve as a function of the inlet pressure and the sign of its change.



Note that if a .DAT file of PIPENET classic Version 4.10 or earlier is loaded the Fully Closed Pressure P_c is set to the value of the Set Pressure P_s by default. This is for consistency with older versions but is not recommended as it means that there is no hysteresis. This can lead to high-frequency state switching.

18.25 Regulator Valve

The regulator valve is used to regulate the pressure in a network. It models a pressure control valve having an instantaneous response. A valve position is calculated so that a specified pressure upstream or downstream of the valve is maintained.

User Input Parameters

Valve Type - select from a list of defined [library](#) valves, or one of the [built-in types](#) (Linear, Equal Percentage or Quick Opening).

If the valve is a [built-in type](#) the following information is required:

K-factor Valve - k-factor for fully open valve value and fully open valve area.

Flow Coefficient Valve - fully open Cv value.

Physical properties of the valve (see modelling equations below).

Fully closed pressure - the maximum pressure required to maintain the valve as fully closed.

Fully open pressure - the minimum pressure required to maintain the valve as fully open.

Order of time response - determines the dynamics of the regulator valve:

zero order - no time delay.

1st order - first order time delay.

2nd order - second order of time delay and damping effect.

Time delay is modelled by transfer functions.

Time constant - entry only applicable if 1st or 2nd order time response selected.

Damping ratio - represents the damping effect of the transfer function on the input pressure - entry only applicable if 2nd order time response selected.

Modelling Equations

1. Valve Equation

Open k-factor valve in SI units ($s \geq 0$)

$$\frac{1}{2} k \rho Q |Q| = S^2 A^2 (P_1 - P_2)$$

Open flow coefficient (Cv) valve ($s \geq 0$)

$$P_1 - P_2 = \frac{\rho Q |Q|}{\rho_0 C_v^2}$$

Fully closed valve ($s = 0$)

$$Q_1 = 0$$

2. 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.

s is the specified valve setting.

A is the fully-open cross section.

ρ is the density of the fluid.

ρ_0 is the density of water at standard conditions.

Regulator valve modelling equations

Equation at instant response (zero'th order setting)

$$\bar{S} = \frac{P - P_{s0}}{P_{s1} - P_{s0}}, \quad P = \begin{cases} P_1, & \text{upstream} \\ P_2, & \text{downstream} \end{cases}$$

P_{s0} The pressure when the valve is fully closed

P_{s1} The pressure when the valve is fully open

The order of regulator dynamics

Zero'th order:

$$S = \bar{S}$$

where S is the actual valve setting.

First order:

$$\tau \frac{ds}{dt} + S = \bar{S}$$

where:

τ is the time constant.

t is the real time.

Second order:

$$\tau^2 \frac{d^2 s}{dt^2} + 2\tau r \frac{ds}{dt} + S = \bar{S}$$

where:

τ is the time constant.

r is the damping ratio.

Notes

1. If time constant τ is zero the above two formulae reduce to the first one, corresponding to the instant response.

18.26 Inertial Check Valve

An inertial check valve is a swing-type [check valve](#). The valve door is hinged at the top and its setting is determined by pressure and flow in the vicinity of the valve.

User Input Parameters

Valve Type - Select from a list of defined [library](#) valves, or one of the [built-in types](#) (Linear, Equal Percentage or Quick Opening)

If the valve is a [built-in type](#) the following information is required:

K-factor Valve - k-factor value at fully open.

Flow Coefficient Valve - fully open Cv value.

Physical properties of the inertial check valve (see modelling equation):

Mass of valve door

Spring constant

Radius of valve door

Angle of valve door when closed

Inclination

Damping

Exponent of velocity term.

Modelling Equations

1. Valve Equation

Open k-factor valve in SI units ($s \geq 0$)

$$\frac{1}{2} k \rho Q |Q| = S^2 A^2 (P_1 - P_2)$$

Open flow coefficient (Cv) valve ($s \geq 0$)

$$P_1 - P_2 = \frac{\rho Q |Q|}{\rho_0 C_v^2}$$

Fully closed valve ($s = 0$)

$$Q_1 = 0$$

2. 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.

s is the specified valve setting.

A is the fully-open cross section.

ρ is the density of the fluid.

ρ_0 is the density of water at standard conditions.

Modelling Equation

The motion of the valve door is governed by the inertia of valve door and the torque acting on the valve door.

The torque equation for such a system is

$$mr^2 \frac{d^2 \Theta}{dt^2} + k \left(\frac{d\Theta}{dt} \right)^n + T_w + T_s + T_h = 0$$

where:

θ is the angle of the valve door.

m is the mass of the valve door.

r is the radius of the valve door.

k is the damping (Note that the unit of the damping coefficient is (Unit of Torque/(rad/s)ⁿ), i.e. the unit given in the component properties is correct for the default case of $n = 1$, but has to be adjusted for other values of n).

n is the exponent of the angular velocity term.

T_w is the torque due to the weight of the door.

T_s is the torque due to the spring.

T_h is the torque due to the hydrodynamic forces.

18.27 Bursting disc

The bursting disc (or rupture disc) is placed in a system to protect the system from damage due to high pressures. During a simulation, if the rupture pressure of the bursting disc is reached it will open in the rupture time specified by the user. It will not close again during the simulation. The flow coefficient C_v , or k factor, is representative of the size of the rupture disc and the frictional losses in it.

User Input Parameters

Pressure drop model - select from C_v or k -factor and port area options.

K-factor valve - K -factor value and port area when fully open.

Flow Coefficient Valve - fully open C_v value.

Rupture pressure - pressure at which disc ruptures

Rupture time - time for disc to fully rupture; that is go from *fully closed* to *fully open*. The rupture time is zero for instantaneous rupture.

Modelling equations

During the simulation, at the very first time when we come across $(P_1 - P_2) = P_{rup}$ or $(P_1 -$

$P_2) > P_{rup}$, we record $t_0 = t$, indicating that the rupture process has just started. Before the simulation time reaches t_0 , i.e., $t < t_0$, $S = 0$, $Q = 0$ are used as the disc is still fully closed, implying that $(P_1 - P_2) < P_{rup}$. After that time point, i.e., when $t > t_0$, the following equation is always used regardless if $(P_1 - P_2) < P_{rup}$ or $(P_1 - P_2) \geq P_{rup}$ as the disc has been either partially ruptured, linearly in the time t during the period of the rupture t_{rup} as governed by $(t - t_0)/t_{rup}$ or completely ruptured, i.e., $S = 1$ at the end of the rupture time t_{rup} and onwards, while a zero rupture time corresponds to an instant bursting may be used as well which corresponds to an instant bursting;

- (1) if $t \leq t_0$, $S = 0$, $Q = 0$
- (2) if $t_0 < t < t_{rup}$ $S = (t - t_0)/t_{rup}$ (partially burst) \Rightarrow Pressure drop models [A]
- (3) if $t \geq t_0 + t_{rup}$ $S = 1$ (fully burst) \Rightarrow Pressure drop models [B]

Partially burst - Pressure drop models [A]

1. Pressure drop models

Open k factor valve equation in SI units ($S > 0$)

$$\frac{1}{2} k \rho Q |Q| = S^2 A^2 (P_1 - P_2)$$

Open flow coefficient (C_v) valve equation in SI units ($S > 0$)

$$P_1 - P_2 = \frac{\rho Q |Q|}{\rho_0 C_v^2}$$

2. Continuity equation

$$Q_1 = Q_2$$

Fully burst - Pressure drop models [B]

1. Pressure drop models

Open k factor valve equation in SI units ($S = 1$)

$$\frac{1}{2} k \rho Q |Q| = A^2 (P_1 - P_2) \text{ or}$$

Open flow coefficient (C_v) valve equation in SI units ($S = 1$)

$$P_1 - P_2 = \frac{\rho Q |Q|}{\rho_0 C_v^2}$$

2. Continuity equation

$$Q_1 = Q_2$$

where:

P_1 is the inlet pressure.

P_2 is the outlet pressure which is left for the user to specify. If this is the case (normally, it is set to zero Bar G, which is equivalent to the atmospheric pressure), then the outflow rate Q_2 will be calculated accordingly.

P_{rup} is the rupture pressure, a pressure difference.

Q_1 is the inlet flow rate.

Q_2 is the outlet flow rate. If this is specified by the user*, then the outlet pressure P_2 will be calculated accordingly.

S is the disc nominal setting.

t is the real time.

t_0 is the time when the rupture begins, i.e., when we come across $(P_1 - P_2) = P_{rup}$ or $(P_1 - P_2) > P_{rup}$ for the first time.

t_{rup} is the rupture time, ranging between 0.1 ms and 10 ms. A zero rupture time may be used as well which corresponds to an instant bursting.

A is the disc open area.

ρ is the density of the fluid.

ρ_0 is the density of the water at standard conditions.

Note that the flow rate specifications must be used cautiously as it may lead to infeasible problems or unrealistic solutions.

18.28 Pressure surge relief valve

This is modelled on a Daniel Model 762 gas loaded relief/back pressure control valve and is specifically designed to regulate and control maximum pipeline pressures or to maintain a minimum back pressure in a system. This quick opening and controllable closing speed valve is assembled by valve body, oil vessel and gas tank. The typical applications include:

- pipeline pressure and surge relief,
- back pressure control
- bypass pressure control
- pipeline pump station by-pass

See the figures below, cited from [DANIEL].

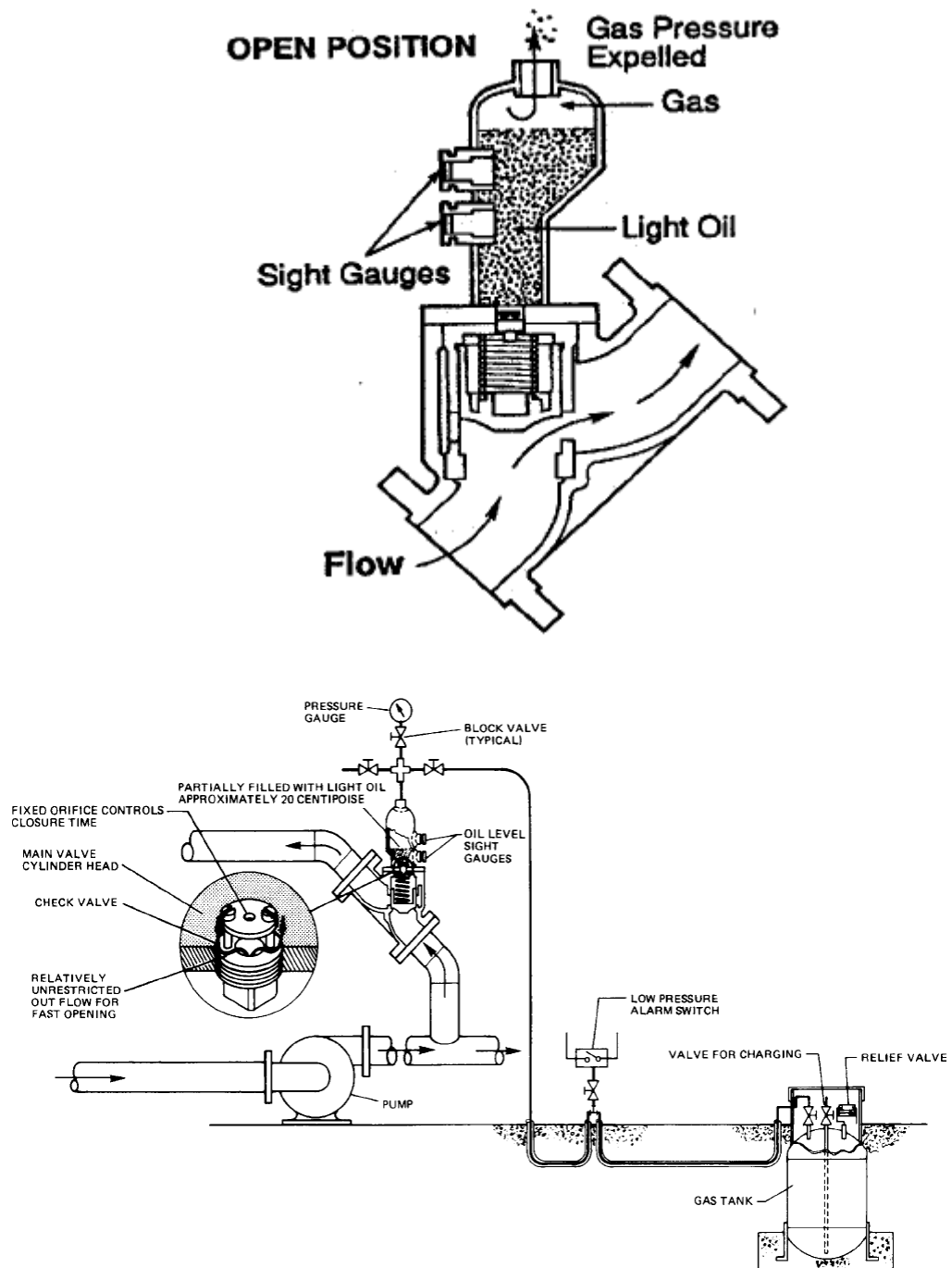


Figure 1 Excess Pressure and Surge Relief or Pump By-Pass Control

Modelling valve equations

1. Valve equation

Open C-factor valve ($s \geq 0$)

$$P_1 - P_2 = \frac{\rho Q |Q|}{\rho_0 C_v^2}$$

Fully closed valve ($s = 0$)

$$Q_1 = 0$$

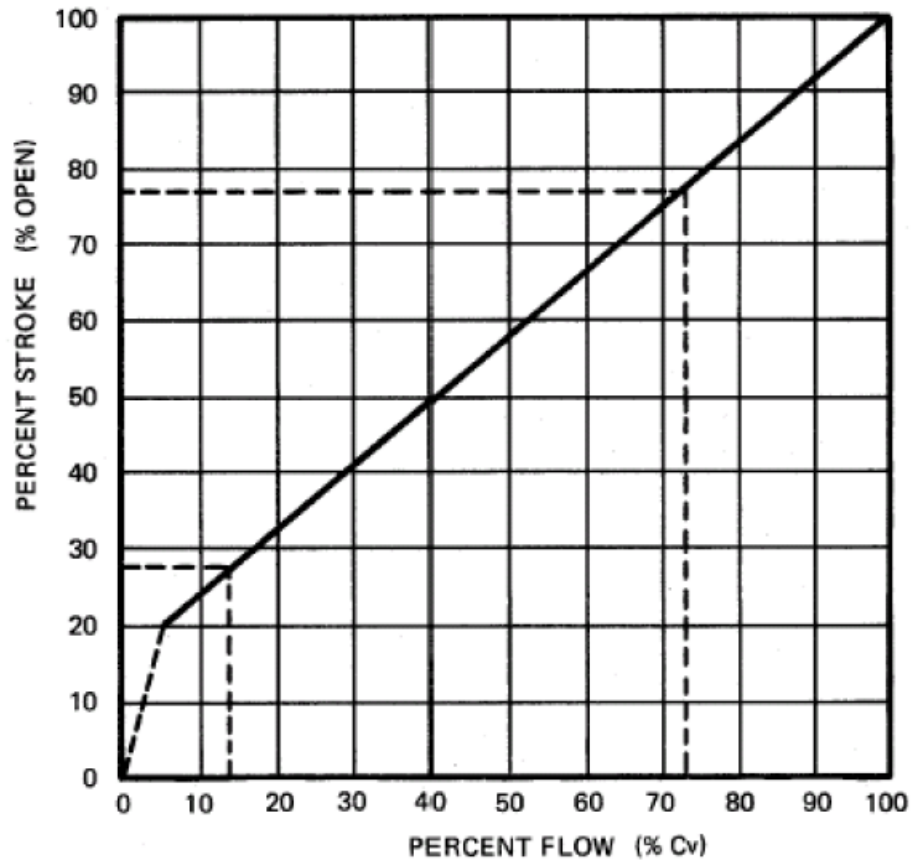
2. Continuity equation

$$Q_1 = Q_2$$

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
ρ	is the fluid density
ρ_0	is the density of water at standard conditions
s	is the valve setting

Valve Cv curve



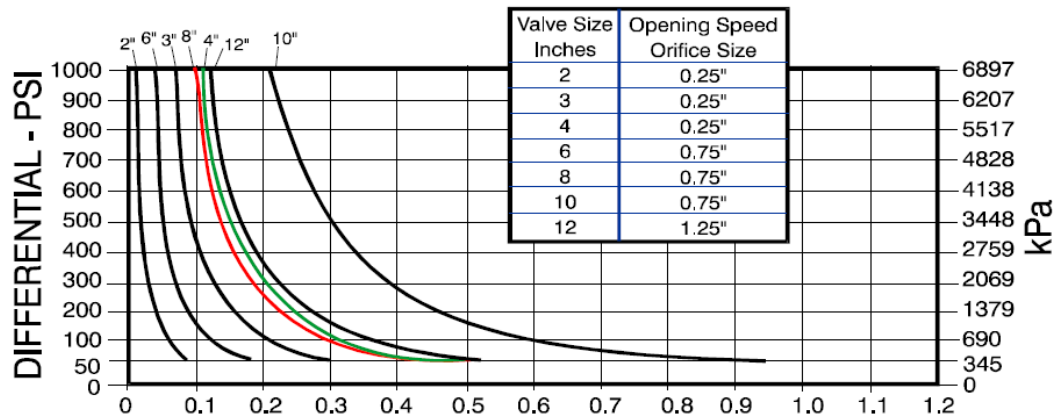
$$0 \leq s \leq 20\% \quad C_v = 0.25 s C_{v_{max}}$$

$$20\% < s \leq 100\% \quad C_v = (1.1875s - 0.1875) C_{v_{max}}$$

Valve position equation

A. Opening process

Opening speed of DANIEL Model 762 is relatively unrestricted which results in very fast opening response, reference the figure below.



B: Closing model:

Valve closing speed is controlled by a fix orifice in the check valve. Closing speed is standard at 3 seconds from a full-open position, based on gas pressure being 25 psi (1724 kPa) above line pressure. The fastest closure is limited to 0.5 seconds at 1000 psi (6897 kPa) differential pressure to protect the piston and seat ring. The closing pattern is similar to the opening pattern except for that it would be slower. Therefore, the above model equations at the opening state are also suitable for the closing process and all valves have same time constant c . If the required closure time is longer than the standard time, the orifice size can be changed to match the customer requirements. It should be noted that the calculated closing speeds are also a function of the viscosity of the oil in the reservoir.

Valve size (in.)	Diff. press (Pa)	Closing time (s)
2" ~ 16"	172425	3

18.29 Accumulator

Accumulators are placed in systems in order to prevent pressure surges. They consist of a sealed cylindrical vessel connected to the network at one point. The trapped gas in the accumulator provides a cushioning effect which reduces any pressure surges. The accumulator is usually a cost effective way of reducing a pressure surge as they are cheap, reliable and easy to maintain.

The sizing of the accumulator is crucial as incorrectly sized accumulators can actually increase pressure surges in other parts of the network.

User Input Parameters

1. Accumulator type - Vertical cylinder (default), Horizontal cylinder or Sphere
2. Accumulator diameter
3. Accumulator height - Vertical cylinder only

4. Accumulator length - Horizontal cylinder only
5. Reference fluid depth
6. Reference gas (see following section)
7. Heat capacity ratio (see following section)
8. Reference gas pressure
9. Reference gas temperature

Reference gas characteristics

The reference gas is selected as one of the following (Diatomic gas being the default):-

Gas type	Heat capacity ratio
Monatomic gas - He, Ne, Ar, Xe, Kr, etc.	1.66
Diatomic gas - Air, H ₂ , N ₂ , O ₂ , NO, etc.	1.40
Triatomic gas - Steam, CO ₂ , N ₂ O, SO ₂ , etc.	1.33
User-defined	specified by user as ≥ 1.0

Note

1. All heat capacity ratios listed are based on atmospheric conditions (zero bar g, 20° C)
2. The heat capacity ratio of triatomic gas is set as 1.33. The actual value is in the range 1.29 to 1.33 because of its asymmetrical molecular structure. For example, Steam = 1.33, CO₂ = 1.30, N₂O = 1.31 and SO₂ = 1.29.

Modelling Equations

Gas volume in accumulators:

$$P_0 V_0^{\gamma} = \text{constant}$$

Pressure head:

$$P - P_0 = \rho g z + \frac{z \rho}{A_h} \frac{dQ}{dt} + \frac{2f \rho z}{D_h A_h^2} Q|Q|$$

Rate of change of fluid height

$$A_s \frac{dz}{dt} = Q$$

Dimension equations

Vertical cylinder $V_l = \frac{\pi D^2}{4} z$

$$D_s = D \quad A_s = \frac{\pi D^2}{4}$$

$$D_h = D_s \quad A_h = A_s$$

Horizontal cylinder $V_l = \left[R^2 a \cos \frac{R-z}{R} - (R-z) \sqrt{2Rz - z^2} \right] L$

$$D_s = \frac{4L \sqrt{zD - z^2}}{2 \sqrt{zD - z^2} + L} \quad A_s = 2L \sqrt{zD - z^2}$$

$$D_h = \frac{D_s}{2} \quad A_h = \frac{A_s}{4}$$

Sphere $V_l = \frac{\pi}{3} z^2 (3R - z)$

$$D_s = 2 \sqrt{zD - z^2} \quad A_s = \pi (zD - z^2)$$

$$D_h = \frac{D_s}{2} \quad A_h = \frac{A_s}{4}$$

where:

A_h is the average area of the liquid column

A_s is the cross-sectional area at the fluid surface

D is the input diameter

D_h is the average hydraulic diameter of the liquid column

D_s is the hydraulic diameter at the fluid surface

f is the friction factor

g is the acceleration due to gravity

L is the tank length

P is the accumulator inlet pressure:

P_0 is the gas pressure:

Q is the inlet flow rate

R is the tank radius

V is the total volume of the accumulator

V_0 is the gas volume of the accumulator

V_l is the liquid volume of the accumulator

z is the height of the fluid surface from the bottom of the accumulator

γ is the gas heat capacity ratio

ρ is the fluid density

If during a simulation the accumulator drains completely then the simulation is automatically halted.

18.30 SurgeTank

The Surge Tank can be used to reduce severe pressure surges. It consists of an open-ended tank in which the fluid level can rise or fall as pressure surges occur near the inlet of the tank.

User Input Parameters

1. **Type** - Vertical cylinder, horizontal cylinder, conical frustum, uniform cross-section
3. **Diameter** - applicable to all tank types except conical frustum. for the uniform cross-section type it is the hydraulic diameter
3. **Top diameter** - top diameter of conical frustum
4. **Bottom diameter** - bottom diameter of conical frustum
5. **Cross-sectional area** - only for uniform cross-section type
4. **Height** - height of tank, not applicable to a horizontal tank
5. **Length** - only required for horizontal cylinder
5. **Initial fluid depth** - start value for the initial fluid depth. This is not applicable if the simulation has a static start with no run-in time - in all other cases the initial fluid depth is calculated automatically based on the inlet pressure of the tank.

Modelling Equations

Momentum equation:

$$P - P_0 = \rho g z + \frac{z \rho}{A_h} \frac{dQ}{dt} + \frac{2f \rho z}{D_h A_h^2} Q |Q|$$

Rate of change of fluid height:

$$A_s \frac{dz}{dt} = Q$$

Dimension equations

Vertical cylinder $V_l = \frac{\pi D^2}{4} z$

$$D_s = D$$

$$A_s = \frac{\pi D^2}{4}$$

$$D_h = D_s$$

$$A_h = A_s$$

Horizontal cylinder $V_l = \left[R^2 a \cos \frac{R-z}{R} - (R-z) \sqrt{2Rz - z^2} \right] L$

$$D_s = \frac{4L \sqrt{zD - z^2}}{2 \sqrt{zD - z^2} + L} \quad A_s = 2L \sqrt{zD - z^2}$$

Conical frustum $V_l = \frac{\pi}{3} z \left[\left[R - (R-r) \frac{z}{H} \right]^2 + \left[R - (R-r) \frac{z}{H} \right] R + R^2 \right]$

$$D_s = 2 \left[R - (R-r) \frac{z}{H} \right] \quad A_s = \pi \left[R - (R-r) \frac{z}{H} \right]^2$$

$$D_h = \frac{D_s + 2R}{2} \quad A_h = \frac{\pi}{4} D_h^2$$

$$\text{Uniform section} \quad V_l = Az$$

$$D_s = D \qquad A_s = A$$

$$D_h = D \qquad A_h = A$$

where:

A is the input cross-sectional area

A_h is the average area of the liquid column

A_s is the cross-sectional area at the fluid surface

D is the input diameter

D_h is the average hydraulic diameter of the liquid column

D_s is the hydraulic diameter at the fluid surface

f is the friction factor

g is the acceleration due to gravity

H is the tank height

L is the tank length

P is the inlet pressure:

P_a is the atmospheric pressure

Q is the inlet flow rate

R is the tank radius or bottom radius (conical frustum type)

r is the top radius (conical frustum type)

t is the time

V_l is the liquid volume in the surge tank

z is the height of the fluid surface from the bottom of the surge tank

ρ is the fluid density

The simulation will stop if the surge tank drains or overflows during the simulation.

18.31 Receiving Vessel

A Receiving Vessel can be used to reduce severe pressure surges or receive and store fluid from the network. It is an atmospheric tank which is similar to a Surge Tank. The only difference between a surge tank and a receiving vessel is that an overflow is allowed in a receiving vessel. The effect of a weir crest can be considered or ignored in the model.

User Input Parameters

1. **Tank type** - vertical cylinder; horizontal cylinder, conical frustum and uniform section (default).
2. **Dimension** – diameter and height for vertical cylinder; diameter and length for horizontal cylinder; top diameter, bottom diameter and height for conical frustum; hydraulic diameter, cross-sectional area and height for uniform section.
3. **Roughness or C-factor** - roughness is used in standard mode and C-factor is used

- in spray mode
4. **Initial fluid depth** - start value for the initial fluid depth. This is only applicable if the simulation has a static start (not required for calculating initial steady-state) - in all other cases the initial fluid depth is calculated automatically based on the inlet pressure of the tank.
 5. **Weir crest** – ignored (default) or considered
 6. **Weir type** – rectangular; V-notch and user defined
 7. **Weir dimension** – width for rectangular and user defined weir and angle degree for v-notch
 8. **Weir discharge coefficient** – only for user defined type weir

Modelling Equations

Pressure head:

$$P_1 - P_a = \rho g z + \frac{z \rho}{A_h} \frac{dQ_1}{dt} + \frac{2 f \rho (z - Y)}{D_h A_h^2} Q_1 |Q_1|$$

Fluid height:

$$\text{Case 1: } A_s \frac{dz}{dt} = Q_1 \quad (z < z_{\max})$$

$$\text{Case 2: } z = z_{\max} \quad (\text{weir crest is ignored and } z = z_{\max})$$

$$A_s \frac{dz}{dt} = Q_1 - Q_2 \quad (\text{weir crest is considered and } z = z_{\max})$$

Weir Crest height:

$$\text{Case 1: } Y = 0 \quad (z < z_{\max})$$

$$\text{Case 2: } Y = 0 \quad (\text{weir crest is ignored and } z = z_{\max})$$

$$Y = z - z_{\max} \quad (\text{weir crest is considered and } z = z_{\max})$$

Overflow rate:

$$\text{Case 1: } Q_2 = 0 \quad (z < z_{\max})$$

$$\text{Case 2: } (z = z_{\max})$$

weir crest is ignored

$$Q_2 = Q_1$$

weir crest is considered

$$Q_2 = \sqrt{\frac{4}{27}} \sqrt{2gbY}^{1.5} \quad (\text{broad-crested, side wall})$$

$$Q_2 = \sqrt{\frac{4}{27}} \left(1 - 0.2 \frac{Y}{b}\right) \sqrt{2gbY}^{1.5} \quad (\text{broad-crested, end contracted})$$

$$Q_2 = \frac{2}{3} \left(0.61 + 0.08 \frac{Y}{z_{\max}}\right) \sqrt{2gbY}^{1.5} \quad (\text{sharp-crested, side wall})$$

$$Q_2 = \frac{2}{3} \left(0.61 + 0.08 \frac{Y}{z_{\max}}\right) \left(1 - 0.2 \frac{Y}{b}\right) \sqrt{2gbY}^{1.5} \quad (\text{sharp-crested, end contracted})$$

$$Q_2 = 0.58 \frac{8}{15} \sqrt{2g} \tan \frac{\theta}{2} Y^{2.5} \quad (\text{V-notch})$$

$$Q_2 = C_d \sqrt{2gbY}^{1.5} \quad (\text{user defined})$$

Dimension equations

Vertical cylinder:

$$V_l = \frac{\pi D^2}{4} z$$

$$D_s = D \quad A_s = \frac{\pi D^2}{4}$$

Horizontal cylinder:

$$V_l = \left[R^2 \arccos \frac{R-z}{R} - (R-z) \sqrt{2Rz - z^2} \right] L$$

$$D_s = \frac{4L \sqrt{2Rz - z^2}}{2 \sqrt{2Rz - z^2} + L} \quad A_s = 2L \sqrt{2Rz - z^2}$$

Conical frustum

$$V_l = \frac{\pi}{3} z \left\{ \left[R - (R-r) \frac{z}{H} \right]^2 + \left[R - (R-r) \frac{z}{H} \right] R + R^2 \right\}$$

$$D_s = 2 \left[R - (R - r) \frac{z}{H} \right]$$

$$A_s = \pi \left[R - (R - r) \frac{z}{H} \right]^2$$

Uniform section

$$V_l = A z$$

$$D_s = D$$

$$A_s = A$$

Average hydraulic diameter/area:

$$A_h = \frac{V_l}{z}$$

$$D_h = \sqrt{\frac{4A_h}{\pi}}$$

where:

- A** is the input cross-sectional area.
- A_h** is the average area of the liquid column.
- A_s** is the cross area at the fluid surface.
- b** is the weir crest width.
- D** is the input diameter.
- D_h** is the average hydraulic diameter of the liquid column.
- D_s** is the hydraulic diameter at the fluid surface.
- f** is the friction factor.
- g** is the gravity acceleration.
- H** is the input tank height
- L** is the input tank length
- P₁** is the inlet pressure.
- P_a** is the atmospheric pressure.
- Q₁** is the inlet flow rate.
- Q₂** is the overflow rate.
- R** is the tank radius or bottom radius (conical frustum type).
- r** is the top radius (conical frustum type).
- t** is the time.
- V_l** is the liquid volume in the receiving vessel.
- Y** is the weir crest height.
- z** is the height of the fluid surface to the bottom of the receiving vessel.
- z_{max}** is the height of the receiving vessel.
- ρ** is the fluid density.
- θ** is the V-notch angle.

The simulation will stop if the receiving vessel drains during the simulation.

18.32 Simple Tank

The Simple Tank can be used to reduce severe pressure surges. It is similar to the [Surge Tank](#) model but can additionally have a flow into the top of the tank, by means of either a flow rate specification or a connection to the pipe network.

User Input Parameters

1. **Tank pressure** - default is atmospheric pressure
2. **Type** - Vertical cylinder, horizontal cylinder, sphere, inverted cone
3. **Diameter** - applicable to all tank types
4. **Height** - Only for vertical cylinder and inverted cone
5. **Length** - only required for horizontal cylinder
5. **Initial fluid depth** - start value for the initial fluid depth. This is not applicable if the simulation has a static start with no run-in time - in all other cases the initial fluid depth is calculated automatically based on the inlet pressure of the tank.

Modelling Equations

Momentum equation:

$$P_1 = P_0$$

$$P_2 - P_0 = \rho g z + \frac{z \rho}{A_s} \frac{dQ_2}{dt} + \frac{2f \rho z}{D_s A_s^2} Q_2 |Q_2|$$

Rate of change of fluid height:

$$A_s \frac{dz}{dt} = Q_1 + Q_2$$

Dimension equations

<i>Vertical cylinder</i>	$D_s = D$	$A_s = \frac{\pi D^2}{4}$
<i>Horizontal cylinder</i>	$D_s = \frac{4L\sqrt{zD - z^2}}{2\sqrt{zD - z^2} + L}$	$A_s = 2L\sqrt{zD - z^2}$
<i>Sphere</i>	$D_s = 2\sqrt{zD - z^2}$	$A_s = \pi(zD - z^2)$
<i>Inverted cone</i>	$D_s = \frac{z}{H} D$	$A_s = \frac{\pi z^2 D^2}{4H^2}$

where:

A_s is the cross-sectional area at the fluid surface.

D is the input tank diameter.

D_s is the hydraulic diameter at the fluid surface.

f is the friction factor.

g is acceleration due to gravity

H is the tank height

L is the tank length

P_0 is the tank air pressure.

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,

z is the height of the fluid surface.

ρ is the fluid density.

The simulation will stop if the simple tank drains or overflows during the simulation.

18.33 Vacuum Breaker

Vacuum breakers are particularly effective in alleviating pressure surges due to cavitation (sometimes called column separation). If the pressure in a system falls below the fluid vapour pressure (as can occur in higher areas of pipework due to static head differences) then the fluid becomes a gas and a cavity of air forms in the system. If such a cavity collapses very large pressure surges occur when fluid on either side of the cavity meet.

Vacuum breakers help eliminate cavitation by sucking air at atmospheric pressure into the network when pressures become too low. The air is expelled to the atmosphere when the system pressure starts to rise again.

There are two set pressure values for the vacuum breaker, set pressure P_{in} of inlet valve

and set pressure P_{out} of outlet valve. When the air pressure P_0 inside the vacuum breaker becomes lower than P_{in} , the vacuum breaker will start to suck the air from atmosphere through the inlet air valve, and the inlet valve will close once P_0 is higher than P_{in} . When P_0 rises higher than P_{out} , the vacuum breaker will start to expel the air to atmosphere through the outlet air valve, and the outlet valve will close once P_0 is lower than P_{out} .

User Input Parameters

1. **Diameter air inlet** - Diameter of vacuum breaker air inlet valve.
2. **Coefficient discharge of inlet** - Coefficient of discharge of air inlet valve.
3. **Set pressure of air inlet** - default is the ambient pressure. The pressure must be less than or equal to the ambient pressure and greater than the vapour pressure of the fluid in use.
4. **Diameter air outlet** - Diameter of vacuum breaker air outlet valve.
5. **Coefficient discharge of outlet** - Coefficient of discharge of air outlet valve.
6. **Set pressure of air outlet** - default is the ambient pressure. The pressure must be greater than or equal to the ambient pressure.
7. **Valve type:-**
 - Standard type: the air valves are controlled by pressure only
 - With timer control: the air valves are controlled by pressure in the active period but are closed in other time.
8. **Start active time** - start time that the air valves are active.
9. **End active time** - end time that the air valves are active.

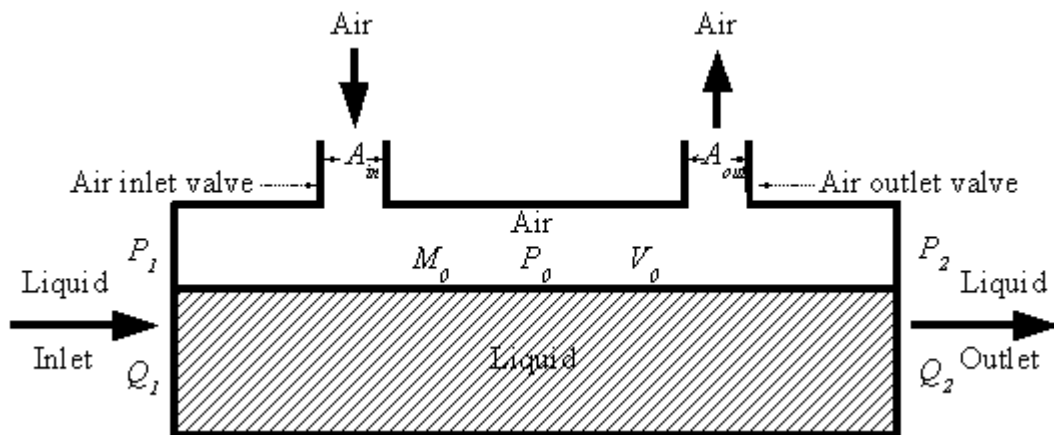
Note: The inputs should be checked by program after users change values.

1. The set pressure of air inlet should be lower than or equal to the ambient pressure and the set pressure of air outlet should be higher than or equal to the ambient pressure.
2. Set pressure of air inlet should be greater than vapour pressure.
3. The default values of both set pressures are set as the ambient pressure and displayed at beginning.
4. Start and end times must both be specified for the With-timer-control option and the end active time must be greater than the start active time. These parameters are not used for the Standard type.

Modelling Equations

The vacuum breaker can connect with the network in two methods:-





State 1: $P_0 > P_{in}$ without any air inside the vacuum breaker

No air is sucked into the network, or expelled to the atmosphere.

Pressure:

$$P_1 = P_2$$

Flow rate:

$$Q_1 = Q_2$$

Mass of air:

$$M_0 = 0$$

Volume of air:

$$V_0 = 0$$

Change rate of the air mass:

$$A_C = 0$$

State 2: $P_0 \leq P_{in}$, or $P_0 \geq P_{out}$ with air inside, or $P_{in} < P_0 < P_{out}$ with air inside

Air is being sucked into the network, or being expelled to the atmosphere, or both valves are closed.

Pressure:

$$P_1 = P_2 = P_0$$

Change of rate of air volume:

$$\frac{dV_0}{dt} = Q_2 - Q_1$$

Air inside the vacuum breaker:

$$P_0 = \frac{M_0}{V_0} RT$$

Change rate of the air mass:

$$A_C = Y(P_0, P_a) \quad \text{if } P_0 \leq P_{in}, \text{ or } P_0 \geq P_{out}$$

$$A_C = 0 \quad \text{if } P_{in} < P_0 < P_{out} \text{ or } t > t_2$$

Air mass:

$$A_C = \frac{dM_0}{dt}$$

Where:

A_C	is the changing rate of the air mass inside the vacuum breaker
M_0	is the mass of the air inside the vacuum breaker
P_1	is the inlet pressure.
P_2	is the outlet pressure.
P_0	is the pressure of the air inside the vacuum breaker.
P_{in}	is the set pressure of the air inlet valve.
P_{out}	is the set pressure of the air outlet valve.
P_a	is the ambient pressure.
Q_1	is the inlet liquid flow rate.
Q_2	is the outlet liquid flow rate.
R	is the gas constant.
T	is the air temperature inside the vacuum breaker.
t_1	is the start active time for a timer type vacuum breaker
t_2	is the end active time for a timer type vacuum breaker
V_0	is the volume of the air inside the vacuum breaker.

$Y(P_0, P_a)$ is a well-defined function for the air flow through the air inlet/outlet valve. The models can be found in [\[WANDS\]](#).

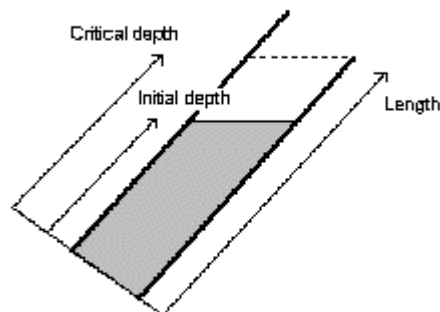
18.34 One-node caisson

A Caisson is a pipe, sealed at one end with an air inlet/outlet valve. The Caisson is used to model dry pipes which can fill or drain the contained fluid during a simulation.

User Input Parameters

1. Caisson length.
2. Caisson diameter.
3. Caisson elevation.
4. Roughness or C-factor - caisson roughness in [Standard option](#) or [Spray option](#), Caisson C-factor only in Spray option.
5. Initial depth of fluid in the Caisson, measured along the length of the Caisson (see below).
6. Critical depth of fluid in the Caisson, measured along the length of the Caisson (see below).
7. Diameter of the Caisson air inlet/outlet valve.
8. Coefficient of discharge of the Caisson air inlet/outlet valve.

At the beginning of a simulation the fluid in the Caisson has an initial fluid depth, measured along the Caisson from the inlet.



If during a simulation the fluid depth rises to the critical fluid depth then the simulation is automatically halted. The simulation can then be continued with the Caisson replaced by a pipe.

Modelling Equations

Fluid in Caisson

$$\frac{dQ}{dt} = \frac{A}{\rho z} (P_1 - P_2) - \frac{2f}{AD} Q|Q| - Ag \sin \alpha$$

Air in Caisson

$$P_2 = \frac{M_c}{V_c} RT$$

Rate of change of air volume

$$A_s \frac{dz}{dt} = Q$$

Volume of air

$$V_c = A(h - z)$$

here:

P_1 is the inlet pressure.

P_2 is the air pressure.

Q is the flow rate into the Caisson.

V_c is the air volume.

M_c is the mass of air.

T is the air temperature.

R is the gas constant.

z is the height of the fluid surface.

h is the height of the Caisson.

A is the cross-sectional area of the Caisson.

D is the diameter of the Caisson.

f is the friction factor.

α is the angle of inclination of the Caisson from horizontal.

g is the acceleration due to gravity.

ρ is the fluid density.

The setting of the air inlet/outlet valve must be specified via a specification on the information node. The air inlet/outlet valve uses the same model as the [Vacuum Breaker](#) for calculating air inflow or outflow.

18.35 Two-node caisson - type 1

The Two-Node Caisson differs from the One-node caisson model only in the fact that:

1. When full, the component will function as a short pipe.
2. There is a built-in non-return valve at the outlet of the caisson. This has the effect that as soon as the flow in the caisson reverses, the caisson will begin to drain.
3. There is a built-in air valve just before the built-in non-return valve, which

is the only path for air exchange with the ambient atmosphere.

18.36 Two-node caisson - type 2

A two-node caisson type 2 is a pipe with a built-in air inlet/outlet valve. Its difference to the earlier two-node caisson model lies in that (1) the valve is now open or closed automatically according to the pressure difference between inside air pressure P_0 inside the caisson pipe and the ambient pressure P_a outside the caisson as well as the level of the fluid inside the caisson, rather than specified by the user beforehand, (2) when caisson pipe is fully filled, rigorous pipe model, instead of short pipe model, is used. This provides the model with a great deal of flexibility and reality. It is used to model dry pipes that can fill or drain the contained fluid during a simulation.

User input parameters

Length	Caisson length/height.
Diameter	Caisson diameter.
Elevation	Caisson elevation.
Roughness or C-factor	Caisson roughness in the Standard or Spray option, Caisson C-factor in Spray option.
Initial Fluid Depth	Initial depth of fluid in the Caisson measured along the length of the caisson.
Valve diameter	Diameter of the Caisson air inlet/outlet valve.
Valve coefficient of discharge	Coefficient of discharge of the Caisson air inlet/outlet valve.
Beta factor	A factor for the air inlet/outlet valve, representing the sensitivity of the valve position to the pressure of air inside the Caisson. It is defaulted to 1.

Modelling equations

(A) When $z < h$ or when $z = h (P_0 < P_a)$

1. Fluid in caisson:

$$\frac{dQ_1}{dt} = \frac{A}{\rho z} (P_1 - P_2) - \frac{2f}{AD} Q_1 |Q_1| - Ag \sin \alpha$$

2. Air in caisson:

$$P_0 = \frac{M_0}{V_0} RT$$

3. Rate of change of air volume:

$$\frac{dV_0}{dt} = -Q_1$$

4. Volume of air:

$$V_0 = A(h - z)$$

5. Caisson air inlet/outlet opening equation:

$$S - 1 - \exp[-B(P_0 - P_a)^2 / P_a^2]$$

6. Rate of change of the air mass:

$$\frac{dM_0}{dt} = SY(P_0, P_a)$$

7. Flowrate:

$$Q_1 = 0$$

(B) When $z = h(P_0 \geq P_a)$

1. Momentum equation:

$$\frac{dQ}{dt} = \frac{A}{\rho} \frac{\delta P}{\delta x} - \frac{2f}{AD} Q|Q| - Ag \sin \alpha$$

2. Continuity equation:

$$\frac{1}{\rho A} u \frac{\delta \rho A}{\delta x} + \frac{1}{\rho A} \frac{\delta \rho A}{\delta t} + \frac{\delta u}{\delta x} = 0$$

3. Pressure of the air:

$$P_0 = P_2$$

4. Mass of the air:

$$M_0 = 0$$

5. Volume of the air:

$$V_0=0$$

6. The height of the fluid surface:

$$z = h$$

7. Position of the air inlet/outlet valve

$$S = 0$$

Where:

P	is the pressure at position x from the pipe inlet at the time t .
P_1	is the inlet pressure.
P_2	is the outlet pressure.
P_0	is the pressure of the air inside the Caisson pipe.
P_a	is the ambient pressure of the air inlet/outlet valve.
Q	is the flowrate at position x from the pipe inlet.
u	is the fluid velocity at position x from the pipe inlet.
Q_1	is the inlet flowrate.
Q_2	is the outlet flowrate.
$Y(P_0, P_a)$	is a well-defined function for the air flow through the valve (refer to Vacuum Breaker).
S	is the position of air inlet/outlet valve ($S=0$, fully closed; $S=1$, fully open).
B	is a factor in the governing function of the position S of the air inlet/outlet valve. Its default value is 1. A value of 0 renders the valve fully closed throughout the simulation. A very larger value renders it fully open. The larger the value of b , the more sensitive the valve is to the air pressure inside the Caisson.
M_0	is the mass of the air inside the Caisson pipe.
V_0	is the volume of the air inside the Caisson pipe.
R	is the gas constant.
T	is the air temperature.
z	is the height of the fluid surface.
h	is the height/length of the Caisson pipe.
A	is the cross-sectional area of the Caisson pipe.
D	is the diameter of the Caisson.
f	is the friction factor.
α	is the angle of the inclination of the Caisson pipe from horizontal.
g	is the acceleration due to the gravity.
ρ	is the fluid density.

Implementation notes

1. If during the simulation both the height z of the fluid and the inlet flowrate Q_1 drop to or below the their respective negative tolerance values, the simulation will stop.
2. If during the simulation the height z of the fluid reaches the height h of the Caisson and the inner air pressure P_0 is equal to or greater than the ambient pressure P_a , the Caisson will be modelled as a regular (rigorous) pipe.
3. The air inlet/outlet valve uses the same model as the Vacuum Beaker for calculating the air inflow or outflow (discharge).
4. When the user-defined time step renders the caisson pipe to be a short pipe, then the short pipe model is used.

18.37 General Pressure Loss

In many cases a component's resistance factor is not constant and may vary slightly with the flow rate. In addition, the pressure drop and the corresponding flow rate may not always have a simple quadratic relationship so that the pressure loss characteristic cannot be modelled by, for example, an operating valve. The General pressure Loss component is designed to be used in those situations demanding a more accurate modelling of the flow rate and pressure relationship.

Modelling equations

Resistance equation

$$P_1 - P_2 = KQ|Q|^{m-1}$$

where:

P_1	is the inlet pressure
P_2	is the outlet pressure
Q	is the flow rate
K	is the resistance factor
m	is the exponent

Resistance factor equation

Constant resistance factor

$$K = \frac{\Delta P_{ref}}{Q_{ref}^m}$$

where:

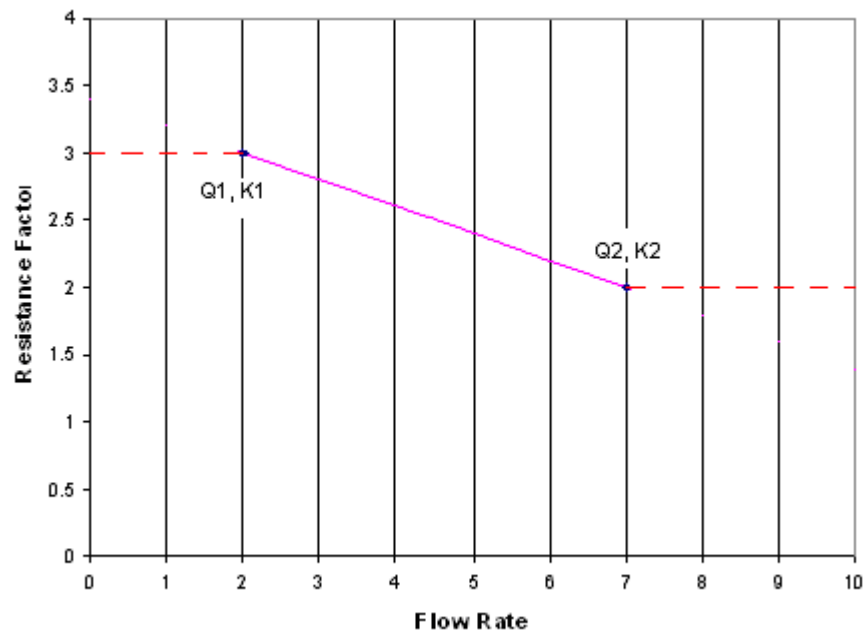
ΔP_{ref}	is the reference pressure drop
Q_{ref}	is the reference flow rate

User-defined type

The resistance factor at the known flow rate and pressure drop can be calculated based on the following equation. The resistance factor at any flow rate can be interpolated based on the above deduced factors.

$$K_i = \frac{\Delta P_i}{Q_i^m}$$

Case1 - only two points exist in the library definition



$$Q_{\min} < Q < Q_{\max}$$

We assume the unknown resistance factor is linear to the flow rate:

$$\frac{K - K_1}{K_2 - K_1} = \frac{Q - Q_1}{Q_2 - Q_1} \quad \text{or} \quad K = CQ + D$$

$$Q < Q_{\min}$$

The Q-K curve will extrapolate to the lower flow rate along the constant K line with the same K factor at the lower limit point:

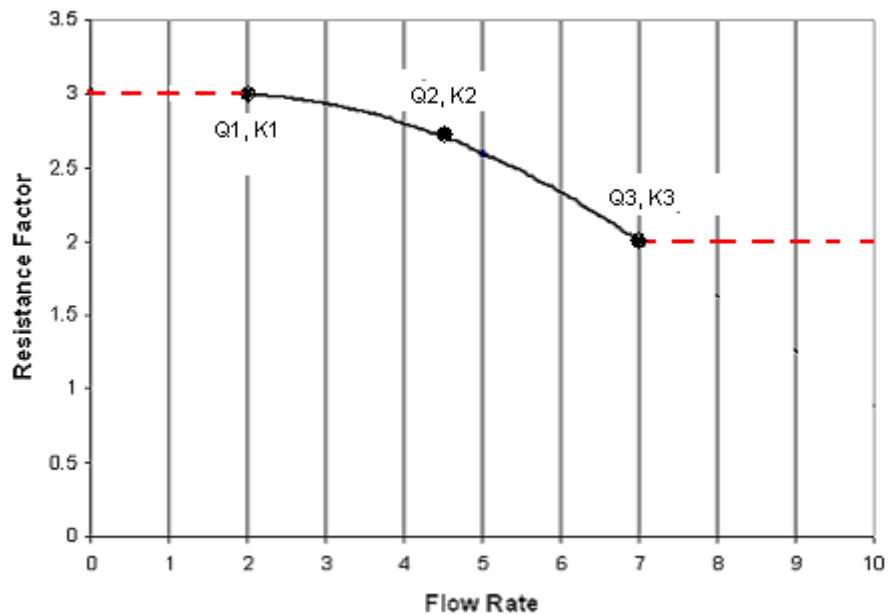
$$K = K_1$$

$$Q > Q_{\max}$$

The Q-K curve will extrapolate to the higher flow rate along the constant K line with the same K factor at the higher limit point:

$$K = K_2$$

Case2 - three points exist in the library definition



$$Q_{\min} < Q < Q_{\max}$$

The resistance factor and flow rate have a quadratic relationship:

$$K = BQ^2 + CQ + D$$

$$Q < Q_{\min}$$

The Q-K curve will extrapolate to the lower flow rate along the constant K line with the same K factor at the lower limit point:

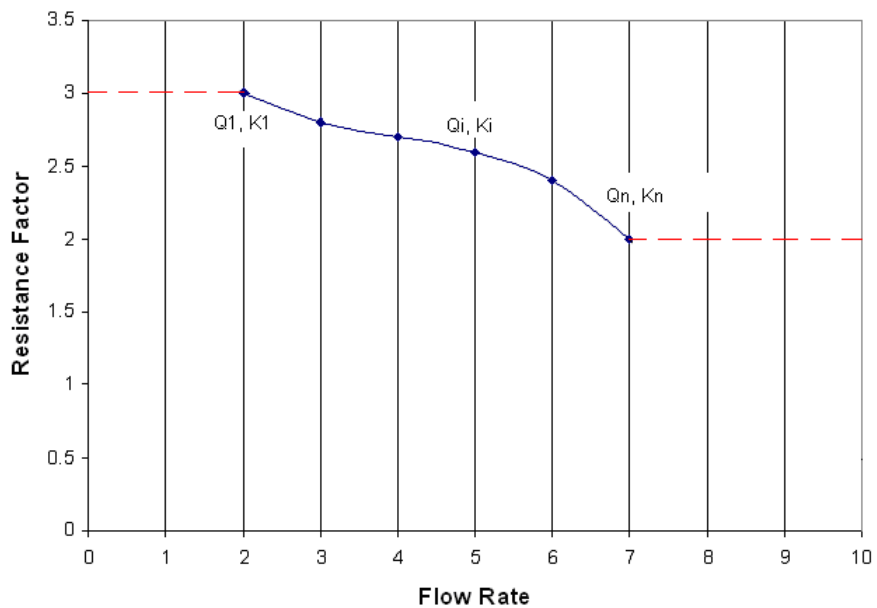
$$K = K_1$$

$$Q > Q_{\max}$$

The Q-K curve will extrapolate to the higher flow rate along the constant K line with the same K factor at the higher limit point:

$$K = K_3$$

Case3 - more than three points exist in the library definition:



$$Q_{\min} < Q < Q_{\max}$$

A cubic spline is used to fit the Q-K curve:

$$K = A_i Q^3 + B_i Q^2 + C_i Q + D \quad (\text{in the region of } [Q_i, Q_{i+1}])$$

$$Q < Q_{\min}$$

The Q-K curve will extrapolate to the lower flow rate along the constant K line with the same K factor at the lower limit point:

$$K = K_1$$

$$Q > Q_{\max}$$

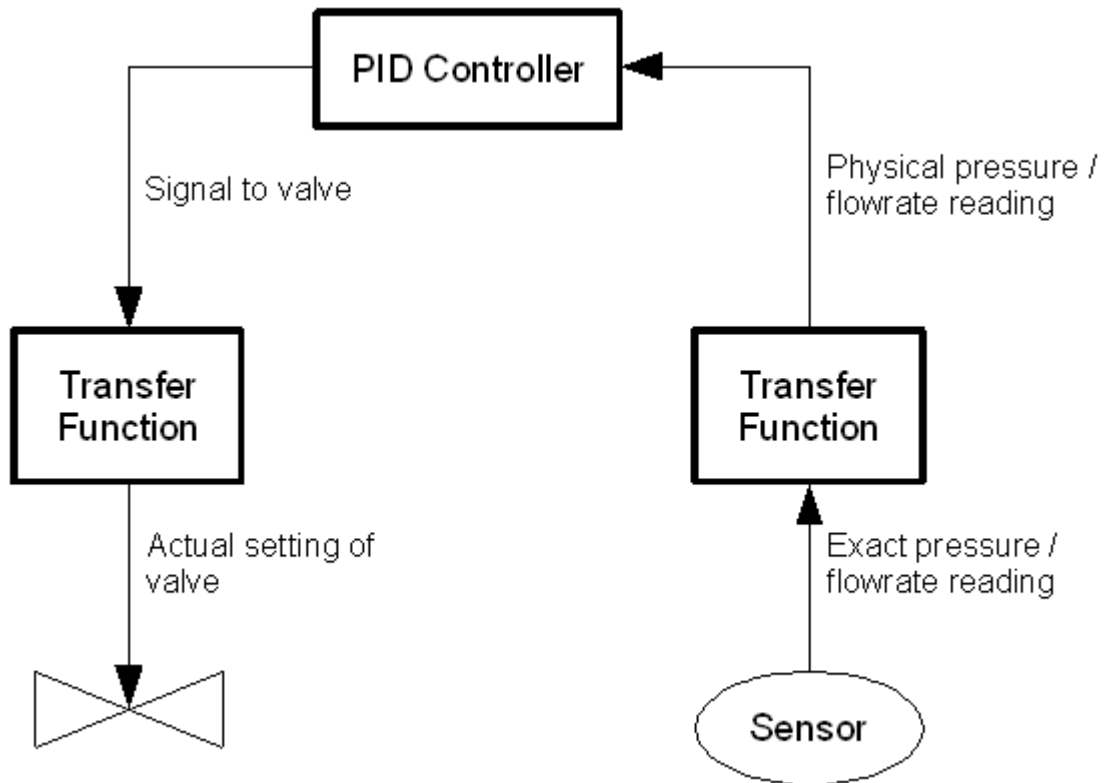
The Q-K curve will extrapolate to the higher flow rate along the constant K line with the same K factor at the higher limit point:

$$K = K_n$$

18.38 Control Systems

Control systems can be introduced into a network to allow components such as [valves](#) or [pumps](#) to react to changes in pressure at a node or flow rate between two nodes. For example, an [operating valve](#) can be converted to a pressure control valve which opens wider as its inlet pressure rises.

A typical control system is shown below:



The [sensor components](#) in Transient Module provides instantaneous readings for pressure or flow rate. However, in reality physical sensors have their own dynamics and their output signal can take time to react to changes in the input signal.

The [transfer function](#) component allows these dynamic effects to be modelled by Transient Module. The transfer function can also be used to model the dynamics of the controlled device (valve or pump).

The [PID Controller](#) takes an input signal from a sensor and converts it to a control signal for the device to be controlled.

A [Switch](#) is available, and can be used to examine the flow rate or pressure at the designated point in the network.

A [Signal selector](#) is also available, and can be used to select the minimum or the maximum of two input signals.

18.39 Sensors

Two generic types of sensor are provided; analogue and digital. The modelling for both types is very similar and they use the same user interface, the main difference being the presence of attribute indicating whether the sensor is analogue or digital.

A. Analogue sensors

Sensors are used to take a Pressure, Pressure Difference or Flow rate reading for the control system. Pressure is measured at a particular node. Pressure difference as the difference in pressure between two specified nodes, Flow rate is measured between two nodes. Sensors are connected to PID Controllers, either directly or via a Transfer Function.

User Input Parameters

1. If a pressure sensor or a pressure difference sensor - Input and Output Node.
2. If a flow rate sensor or a pressure difference sensor - Output info node.
3. Information node for connection to a transfer function or PID Controller.

Modelling Equations

Pressure sensor

$$P_1 = P_2$$

and

$$Q_1 = Q_2 = 0$$

Pressure difference sensor

$$\Delta P = P_1 - P_2$$

and

$$Q_1 = Q_2 = 0$$

Flow sensor

$$P_1 = P_2$$

and

$$Q_1 = Q_2$$

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.

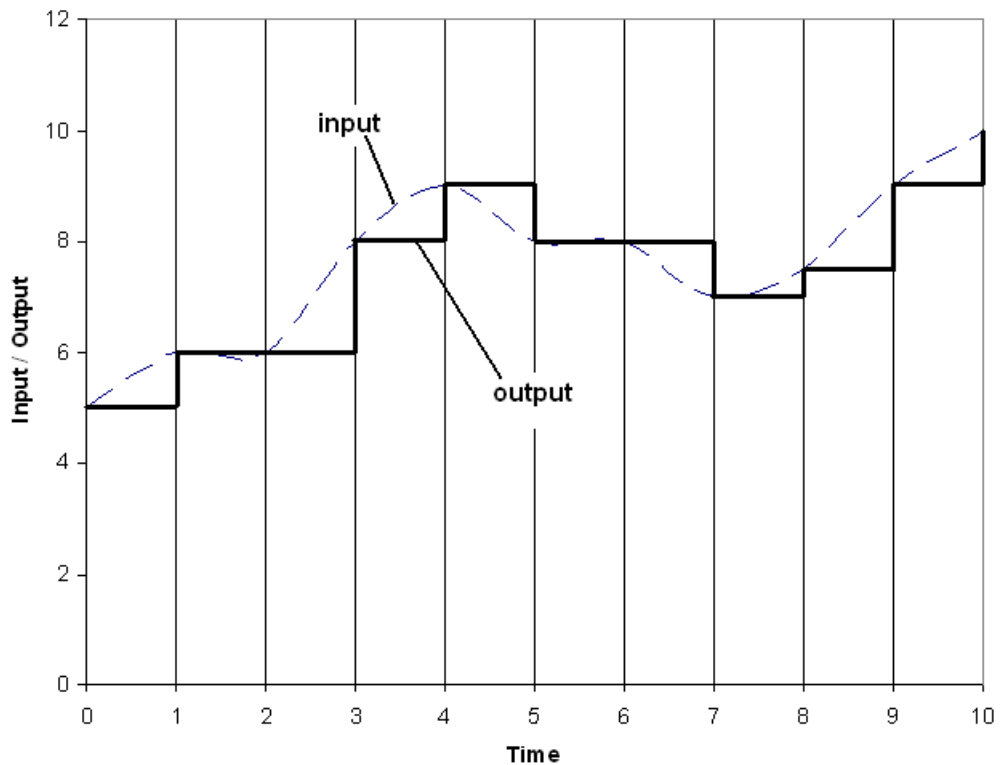
ΔP is the output signal, i.e. the pressure difference.

B. Digital sensors

Digital sensors are used to simulate the behaviour of discrete data system. The designed components include digital pressure sensor, digital flow sensor and digital differential pressure sensor.

The digital sensors share the same symbols and interfaces with the conventional (analogue) sensors. In addition, the digital sensors have a new parameter, is Scan frequency with the unit of 1/time, which must be equal to or larger than the time step.

Input and output



Modelling equations

Pressure sensor

$$P_2 = P_1|_{t=ti}$$

$$Q_1 = Q_2 = 0$$

Pressure difference sensor

$$\Delta P = (P_1 - P_2)|_{t=ti}$$

$$Q_1 = Q_2 = 0$$

Flow sensor

$$P_2 = P_1$$

$$Q_2 = Q_1$$

$$Q = Q_1|_{t=ti}$$

where:

- ΔP is the pressure difference.
 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.
 t is the time
 ti is the discrete time point, that is $i * dt$

18.40 Transfer Function

Transfer functions model the dynamics of a physical device. For example, if connected between a [sensor](#) and the [PID Controller](#) the Transfer Function models the dynamic response of the sensor. If connected between a [PID Controller](#) and an [operating valve](#) the transfer function models the dynamic response of the valve.

User Input Parameters

Input Variable Type - Pressure, Pressure Difference or Flow rate if from [sensor](#), information if from [PID Controller](#).

Output Variable Type - Pressure, Pressure Difference or Flow rate if to [PID Controller](#), information if to [Operating Valve](#) or [\(Turbo\) Pump](#).

Order - Indicates physical behaviour of transfer function - Zero order, First order, Second order or Limiting power ramp

Parameters for dynamic modelling equations (see below)

Gain - default value 1

Bias - default value 0

Time Constant - represents the time delay for the device to react to changes in its input

Damping Ratio - represents the damping effect of the transfer function on the input signal

Output range (min) - default value 0 for the information output type transfer function, unlimited for other types

Output range (max) - default value 1 for the information output type transfer function, unlimited for other types

Ramp-up limit - must be positive or unset. Only available for the Limiting

power ramp order option. An unset value indicates no limitation on ramp up rate.

Ramp-down limit - must be negative or unset. Only available for the Limiting power ramp order option. An unset value indicates no limitation on ramp down rate.

Modelling Equations

Zero Order

$$y = Gx + B$$

and

$$y_{min} \leq y \leq y_{max}$$

where:

$x(t)$ is the input signal.

$y(t)$ is the output signal.

G is the Gain. The gain represents the amplification of the signal.

B is the bias, which represents the offset of the signal

y_{min} is the minimum output, which represents the lower limitation of the signal

y_{max} is the maximum output, which represents the upper limitation of the signal

First Order

$$\tau \frac{dy}{dt} + y = Gx + B$$

and

$$y_{min} \leq y \leq y_{max}$$

where:

τ is the time constant. The time constant represents the time delay for the device to react to changes in its input.

Second Order

$$\tau^2 \frac{d^2 y}{dt^2} + 2r\tau \frac{dy}{dt} + y = Gx + B$$

and

$$y_{min} \leq y \leq y_{max}$$

where:

r is the damping ratio. The damping ratio represents the damping effect of the transfer function on the input signal.

Limiting power ramp

$$\frac{dy}{dt} = u + \quad (y > Gx + B)$$

$$\frac{dy}{dt} = u - \quad (y < Gx + B)$$

$$y = Gx + B \quad (u - < dy/dt < u +)$$

and

$$y_{min} \leq y \leq y_{max}$$

where

$u+$ is the ramp up rate

$u-$ is the ramp down rate

18.41 PID Controller

The PID Controller takes an input signal y from a [sensor](#) and converts it to a control signal u for the device to be controlled. The three types of controller

- P - proportional
- I - integral and
- D - differential

can be used in isolation or combination to give a wide range of controller behaviour.

The controller uses a set point (y_s, u_s) to determine its steady state behaviour. When supplied with a constant signal y_s the controller produces a constant output signal u_s . If something happens to change the input signal the controller changes its output signal based on the error $e = y - y_s$.

Note that the output signal u is limited to the range $[0,1]$ except in the case of controllers with integral (I) term but without the Anti-Windup option. Generally control systems will eventually settle down and the error e will approach a final value called the offset. Controllers are usually judged on how quickly they settle down, and on the size of the offset.

P-Controllers are usually the cheapest, but tend to give a non-zero offset.

I-Controllers tend to eliminate the offset, but can introduce oscillations before the signal settles down. **Anti-Windup I-Controllers** stop integrating when the controller output hits its limits. This reduces the oscillations, particularly in situations such as start-up or when large changes occur.

D-Controllers damp the oscillations, causing the signal to settle down more quickly. Too large a D-term should be avoided as this makes the controller very sensitive to measurement noise which may result in excessive high frequency controller action.

User Input Parameters

The input node collects signals from a [sensor](#) or a [transfer function](#). The Output node sends a signal to [transfer function](#) or controlled device.

Type - Pressure, Pressure Difference or Flow rate Controller

Set Point - (y_s, u_s) as defined above.

Controller Type - any combination of P (proportional), I (Integral with or without Anti-Windup option) and D (differential)

Parameters for PID modelling equations (see below)

Gain

Reset Time

Anti-Windup

Rate Time

Tracking Time Constant

Modelling Equations

$$P \quad u(t) = u_s + Ke(t)$$

$$I \quad u(t) = u_s + K \frac{1}{T_I} \int_0^t e(\tau) d\tau$$

$$D \quad u(t) = u_s + KT_D \frac{de}{dt}$$

$$PI \quad u(t) = u_s + K \left(e(t) + \frac{1}{T_I} \int_0^t e(\tau) d\tau \right)$$

$$PD \quad u(t) = u_s + K \left(e(t) + T_D \frac{de}{dt} \right)$$

$$ID \quad u(t) = u_s + K \left(\frac{1}{T_I} \int_0^t e(\tau) d\tau + T_D \frac{de}{dt} \right)$$

$$PID \quad u(t) = u_s + K \left(e(t) + \frac{1}{T_I} \int_0^t e(\tau) d\tau + T_D \frac{de}{dt} \right)$$

Notation:

K is the Gain

T_I is the Reset Time

T_D is the Rate Time

T_T is the Tracking Time Constant

u, u_s is the Output, output set point

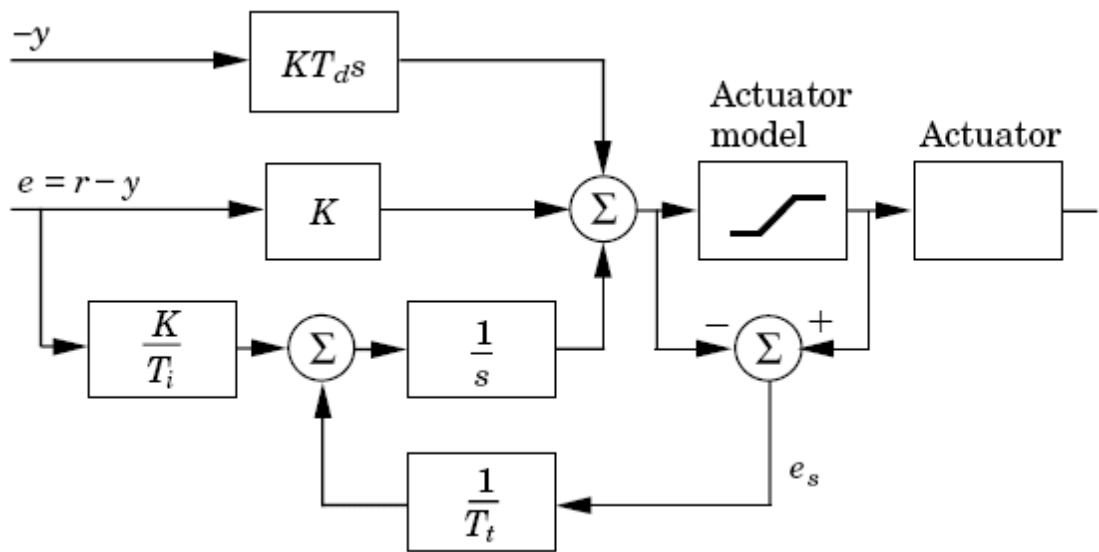
y, y_s is the Input, input set point (also called reference signal)

The error is defined as:

$$e(t) = y(t) - y_s$$

Anti-Windup

Anti-Windup is modelled by introducing a negative feedback signal when saturation occurs, the logic diagram and equation being as follows:



$$u_I = \frac{K}{T_I} \int_0^t e(\tau) d\tau + \frac{1}{T_T} \int_0^t [v(\tau) - u(\tau)] d\tau$$

where:

$$\begin{aligned} v &= u_{\min} && \text{if } u \leq u_{\min} \\ v &= u && \text{if } u_{\min} \leq u \leq u_{\max} \\ v &= u_{\max} && \text{if } u \geq u_{\max} \end{aligned}$$

Here u is the calculated output and v is the desired (real) output. T_T is the tracking time constant.

Ideally the tracking time constant T_T should be larger than the rate time T_D but smaller than the reset time T_I . A rule of thumb is to choose T_T such that:

$$T_T = \sqrt{T_D T_I}$$

User interface considerations

If Anti-Windup is specified then a value for the Tracking time constant must be provided, if no value is provided then a default value will be provided as follows:

for I and PI type controllers $T_T = 0.5T_I$

for ID and PID controllers $T_T = (T_I \times T_D)^{0.5}$

Note that if the tracking time is not provided, the user interface will calculate the appropriate default automatically as new values of T_I and T_D are provided. However, if at any time a value is explicitly provided for T_T any further changes to either T_I or T_D will not affect the value of T_T .

Notes

1. Usually a negative gain must be chosen to obtain negative (= stable) feedback.
2. Setting $T_D > T_I / 4$ results in oscillatory behaviour of the controller by itself.
3. The output signal u is limited to the range [0,1] except in the case of controllers with integral (I) term but without the Anti-Windup option.
4. The differential term is bandwidth limited $\omega = 100/T_D$.

18.42 Cascade PID Controller

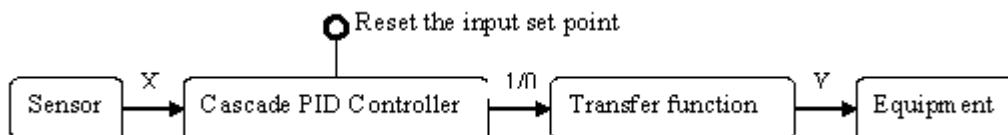
Introduction

Differing from the traditional PID controller, the set point of a cascade PID controller can be changed with time or based on the signal from pressure, differential pressure or flow sensors. If a time type cascade PID controller is given a constant set point, it can work as a traditional PID controller. One of the significant advantages of a Cascade PID controller is its quick reaction in transient conditions. However, the system and the setting are more complex.

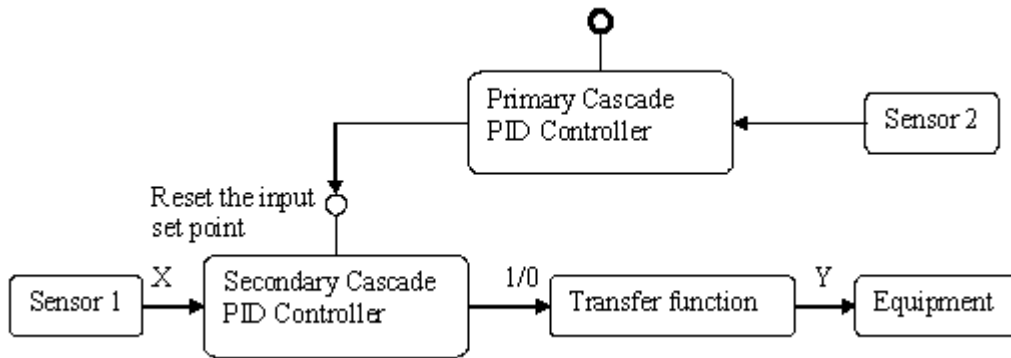
Typical connection

Based on the control type of the set point, a Cascade PID Controller can be classified as a time, pressure, differential pressure or flow rate type controller.

Time type Cascade PID Controller



Pressure, differential pressure and flow rate type Cascade PID Controllers

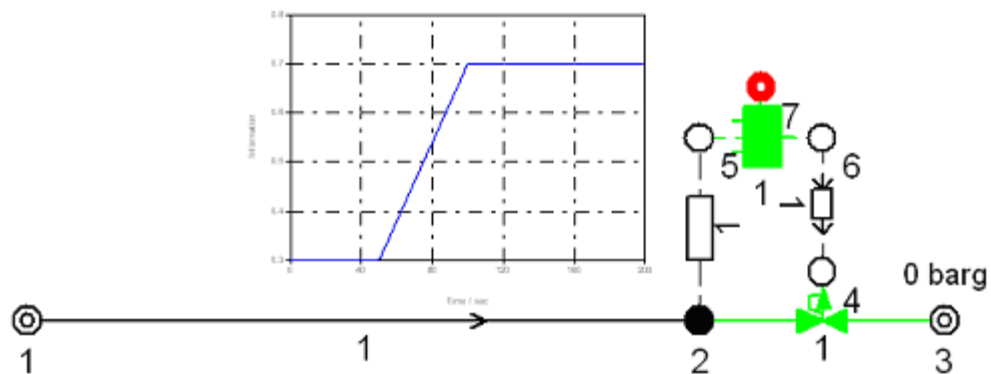


Components

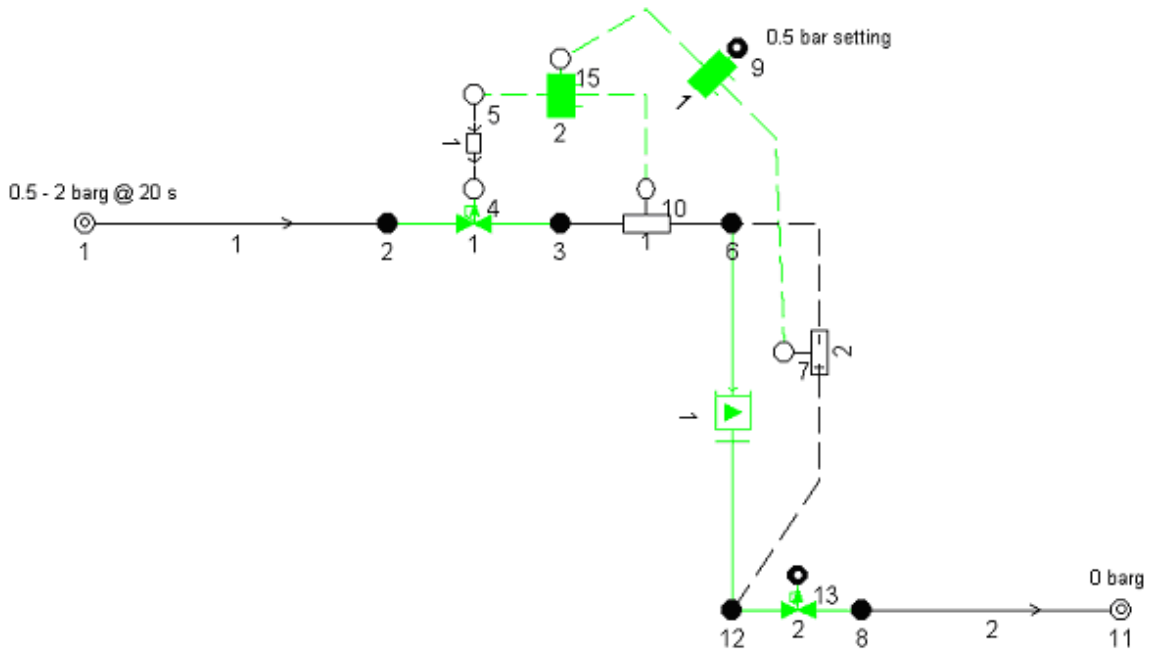
Sensor:	pressure, differential pressure or flow sensor.
Cascade PID Controller:	
Transfer function:	
Equipment:	valve, pump etc.

Typical applications

Time type controller

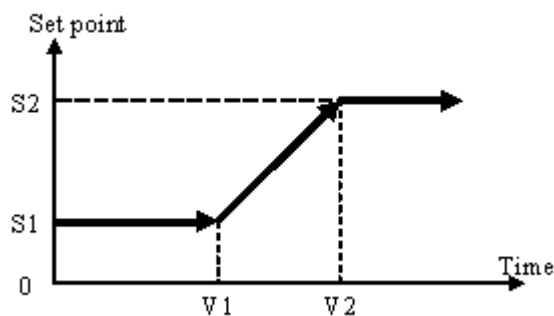


Pressure, differential pressure and flow type controller



Condition and logical diagrams

Time type controller



Pressure, differential pressure and flow type controller

The set point of the secondary cascade PID controller depends on the output of the primary cascade PID controller. The primary cascade PID controller should be a time type controller. In addition, the secondary loop process must have influence over the primary loop and its dynamics must be at least four times as fast as primary loop process.

Output list

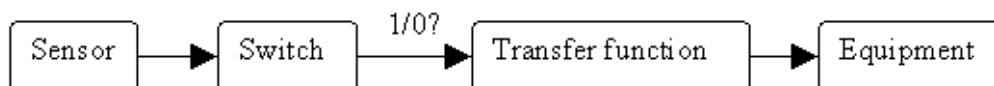
Table:	Input set point, Input variable, output variable
Graph:	Input set point, Input variable, output variable

18.43 Switch

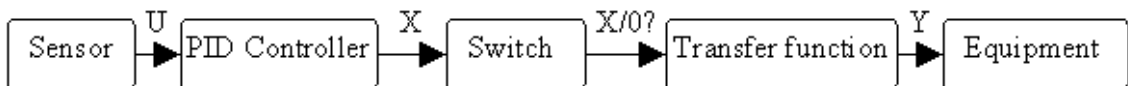
A switch is used to examine the flow rate, differential pressure or flow rate at a designated point in the network, or to act as a timer switch. It compares the monitored value at the designated point and compares it with the preset value to decide the operation (start-up or trip) of the identified equipment (valve, pump, etc)

Typical connection

Pressure, differential pressure and flow type switches

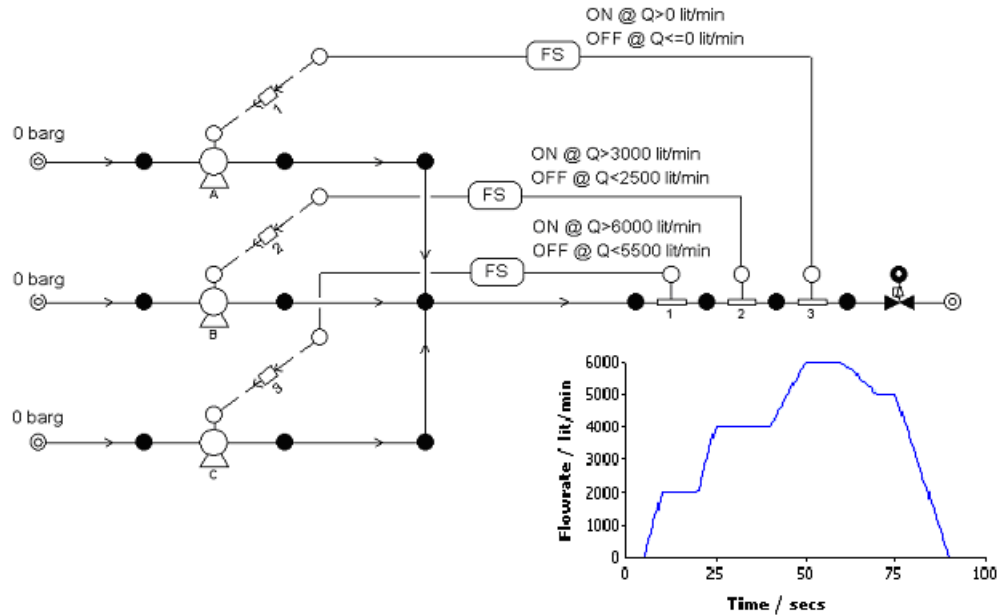


Timer switch



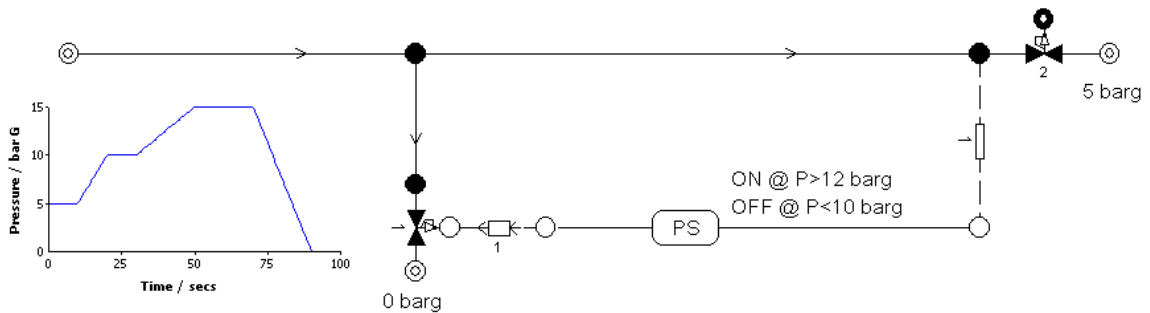
Flow switch scenario

This scenario simulates the start-up of the pumps group. Pump A starts up first; pump B starts up when the required flow rate increases to 3000 lit/min and then stops when the required flow rate decreases to 2500 lit/min, i.e. pump B has 500 lit/min hysteresis; pump C starts up or stops when the measured flow rate is beyond or below 6000 or 5500 lit/min separately.



Pressure switch scenario

This scenario simulates the action of the unloading valve 1. Valve 1 is opened when the system pressure is higher than the setting value (12 bar g) to protect the network from over-pressure. The hysteresis of valve is 2 bar G



User Input Parameters

Switch Type - Pressure, Pressure Difference, Flow rate or Timer.

Initial setting - Default, On or Off - not used for timer switches.

Switch on value - X1, together with an indication of whether the switch on is above or below the specified value - below option is not used with timer switches.

Switch on value X2, together with an indication of whether the switch on is above or below the specified value - below option is not used with timer switches.

Positive or negative going see Logic diagrams below - not used with timer switches.

Dead Time - default value is zero - not used with timer switches.

Flow direction to be considered or ignored - if flow direction is to be considered then the switch-on and switch-off values can be signed; if the flow

direction is to be ignored then the values cannot be less than zero.

Input and output

Input

P - pressure, D - differential pressure, F - flow rate or T - information signal (information node).

Output:

Timer type switch: X — switch on or 0 — switch off (information node).

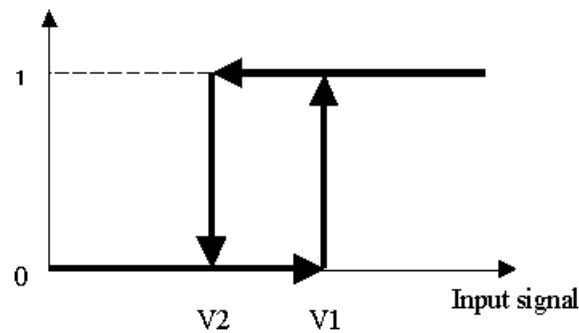
Other type switch: 1 — switch on or 0 — switch off (information node).

Logic diagrams

Pressure, differential pressure and flow type switches

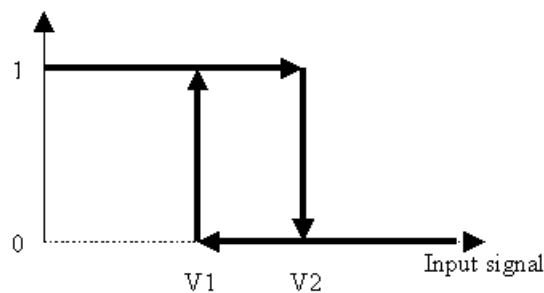
Case 1: switch on above V1 and off below V2 ($V1 \geq V2$).

Output signal



Case 2: switch on below V1 and off above V2 ($V1 \leq V2$).

Output signal

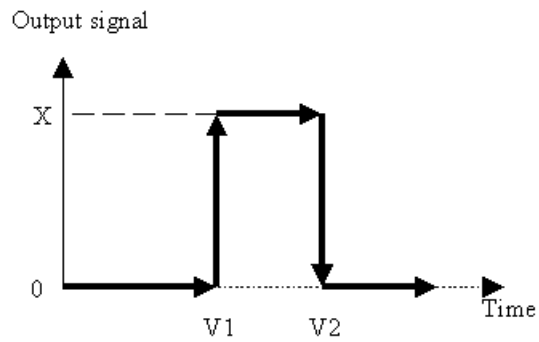


Notes

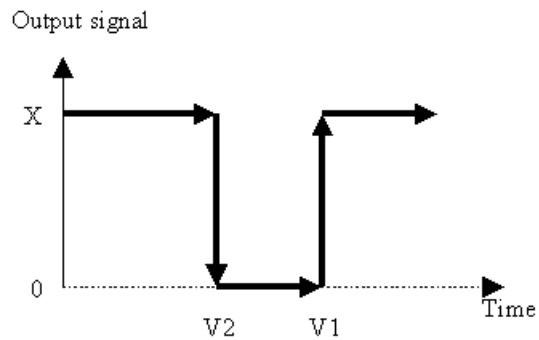
1. When considering flow direction, the setting value can be positive or negative. For instance, the switch settings are: switch on above -8000 lit/min and switch off below -10000 lit/min, so the switch should be on when the flow rate is -7000 lit/min and be off when the flow rate is -12000 lit/min respectively.
2. When ignoring flow direction, the setting values must be positive.

Timer type switches

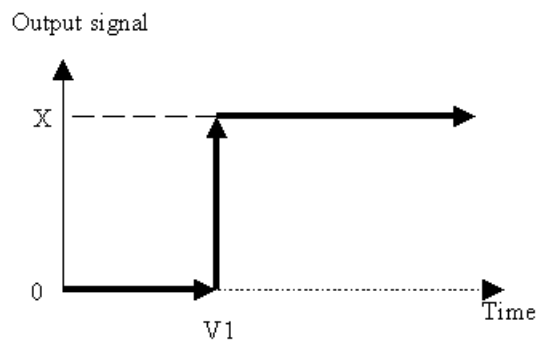
Case 1: switch on above $V1$ and off below $V2$ ($V1 < V2$).



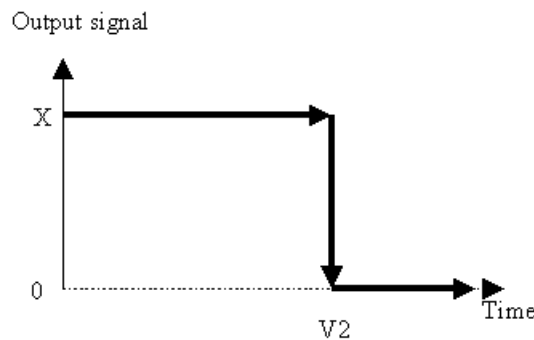
Case 2: switch on above $V1$ and off below $V2$ ($V1 > V2$).



Case 3: switch on above $V1$ and unset $V2$ (switch off time).



Case 4: unset V1 (switch on time) and switch off above V2.



Dead time (time hysteresis)

Dead time is called as time hysteresis sometimes. I.e. the switch postpones a dead time to change its state. Dead time can implement another important function - anti-noise. I.e. the switch will keep at its original state without change if the input signal cross the setting value more than once within a dead time.

Initial setting state

We can appoint the initial switch state if the input signal at the beginning of simulation is in the range between V1 and V2. This does not affect the switch state when the initial input signal is at out of range between V1 and V2. I.e. the switch state can be decided uniquely by calculation. However, if the input signal relates to the controlled equipment, it may cause a conflict state and in this case the default (or calculated) setting has priority. For instance, a switch controls the operation of a pump; the signal comes from a pressure sensor located behind the pump.

- Case 1: If the pump runs, the pressure (signal) will be beyond the setting value which will force the switch off and the pump stop.
- Case 2: If the pump stops, the pressure (signal) will be between the setting values, in which case the switch could be on or off.

In the above simulation, the initial state of the switch always is set to off whatever you set it because it is the only possibility.

18.44 Signal Selector

Signal selectors are used to compare the input signals and output either (a) the minimum or maximum signal to implement the override control function, or (b) the first signal up until a specified time (in the user-supplied units), and the second signal thereafter. The compared input signals must be of the same type. If they are not, the signals must be converted using other control components (such as a transfer function) before transmission.

User Input Parameters

Component label

Input node 1 - the first input

Input node 2 - the second input

Output node - the output, being the minimum or maximum value of the two inputs, dependent on the model type

Signal type - Pressure, pressure difference, flow rate or information

Model type - Minimum, Maximum or Time switch

Switch time - the time at which the output changes from the first input to the second one (used only if the model type is “Time switch”)

Modelling Equations

Minimum type

$$y = \min(x1, x2)$$

Maximum type

$$y = \max(x1, x2)$$

Time switch

$$y = \begin{cases} x1, & t < t_s \\ x2, & t \geq t_s \end{cases}$$

where

$x1, x2$ are the two input signals

y is the output signal

t is the time

t_s is the time at which the output is switched from the first input to the second one.

Exporting the Schematic

Part

19

19 Exporting the Schematic

19.1 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](#)].

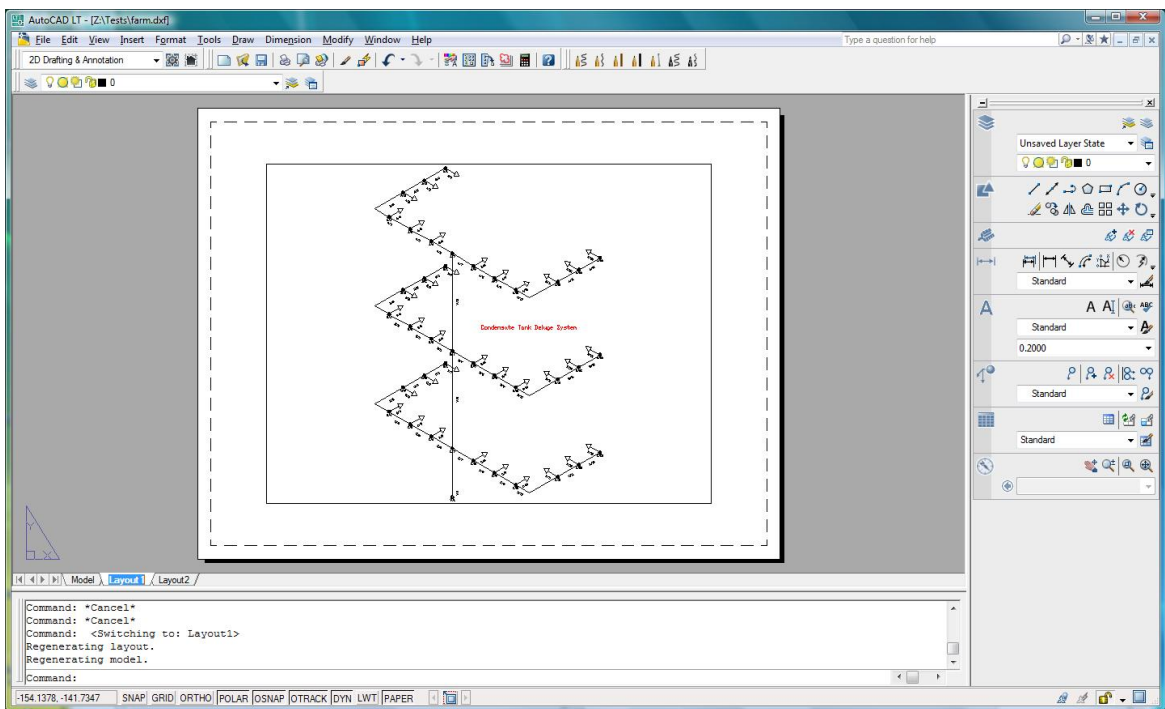
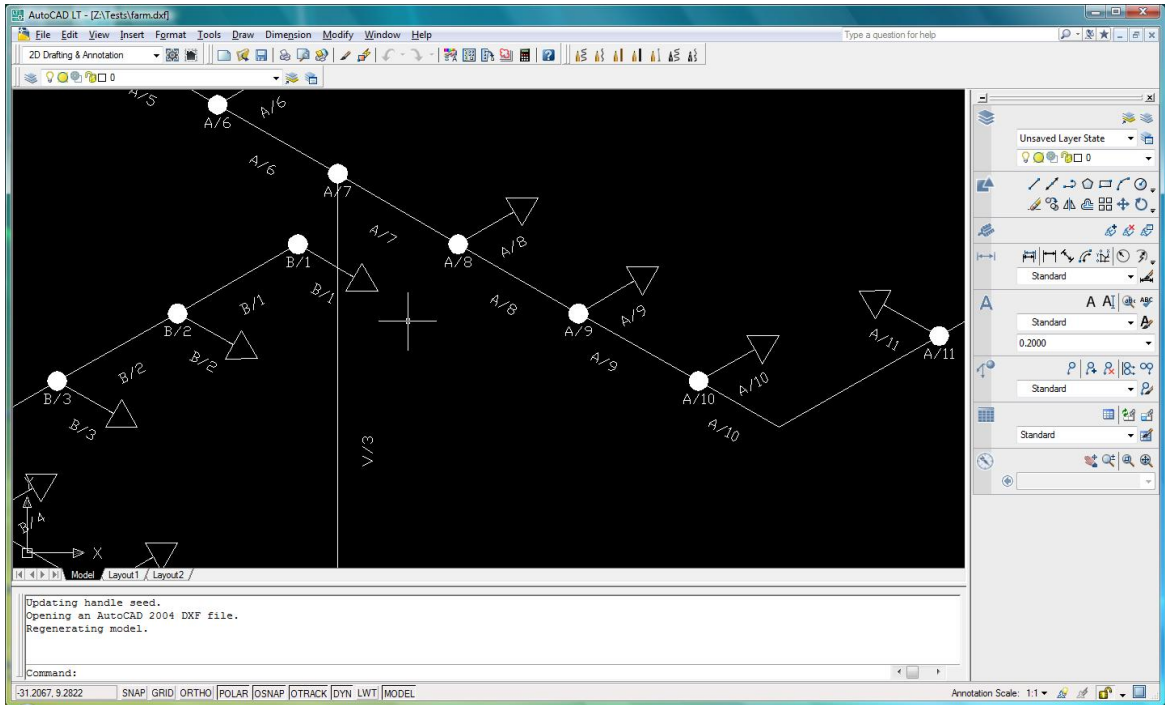
19.2 DXF Output

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:



Errors

Part

20

20 Errors

20.1 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.

20.2 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 refer to 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.

20.3 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.

20.4 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 Transient module the following errors are possible, classified as errors, warnings or information:

A library <component> must be defined – error

User defined pumps and turbo pumps are not allowed. One of these components exists in the network but there is no library item defined.

At least one pipe type must be defined – error

If the Spray option has been requested then all pipes must belong to a pipe type, therefore at least one pipe type must have been defined.

Calculator does not permit a mixture of scheduled and non-scheduled pipes – error

In Standard mode all pipes must belong to a pipe type or no pipes must belong to a pipe type, the calculator does not permit a mixture of scheduled and non-scheduled pipes.

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.

Info node has more than two attached components – error

Information nodes must have two and only two attached components, where a component may be a PID controller, switch, etc., or a specification.

Linear Network: Pressure envelope available - information

The network is linear and therefore a pressure envelope can be produced.

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.

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.

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](#).

Node has less than 2 attached components – error

Every node must have at least two attached components, where a component may be a pipe, a pump, a valve, etc., or a specification.

This node may need to be 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..

Non-Linear Network: Pressure envelope not available - information

The network is non-linear, that is there are one or more pipes in branches, and therefore a pressure envelope cannot be produced.

No pressure specifications defined – error

No pressure specification has been provided.

No rigorous pipe - error

A user defined time step has been specified and all normal pipes in the network are classed as short given this time step (that is for all normal pipes, length < time step * wave speed). To correct this the user should either decrease the time step or increase the length of a pipe.

Part of the network is disconnected – error

The network is disconnected, that is the network is separated into two or more unconnected sections.

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.

Pump is not protected against flow reversal - warning

The pump is not protected against flow reversal, a non-return valve should be placed after the pump.

Specification is on a non-boundary flow node - error

A specification has been provided for a node that is not a boundary (input or output) node.

There must be at least two specifications - error

There must be at least two specifications provided for the analysis phase – see [specifications requirements](#).

20.5 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)
- Air temperature must be specified (accumulators)
- Inclination must be between -90 and 90 degrees (fluid damped check valves)
- Set pressure of air outlet should be greater than the vapour pressure (vacuum breakers)
- Flow coefficient must be given (bursting valves)
- Equipment description must be given (equipment items)

Note that many of these errors are of the general form *<attribute name> must be given*, as in the last two 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.

20.6 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. 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:

- If the initial fluid depth of a caisson is zero and the downstream pressure is less than the pressure inside the caisson when the calculation starts then the air inside the caisson will be expelled into the network.
- The simulation will stop if a simple tank drains.
- The simulation will stop if a simple tank overflows.
- The simulation will stop if the flow rate into a tank at the top goes negative.

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

21

21 Reference Data

21.1 Bibliography

- [CRANE] Technical paper No 410M "Flow of Fluids Through Valves, Fittings and pipe"; Crane Co, 1988
- [ARDT] "Fluid Transients in Pipeline Systems"; A.R.D. Thorley, D & L George, Herts., England, 1991
- [WANDS] "Fluid Transients in Systems"; E.B. Wylie, V. L. Streeter, Prentice Hall. 1993
- [DANIEL] "Daniel Model 762 Design Specifications"
- [API] "Technical Data Book - Petroleum Refining"
- [HPGL2] "The HP-GL/2 and HPRTL Reference Guide", Third edition, Addison Wesley, 1997
- [ACAD06] "DXF Reference", March 2006, Autodesk
- [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

21.2 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	$5.91 \cdot 10^{-4}$ to $2.36 \cdot 10^{-3}$
Brass	0.001 to 0.003	$3.94 \cdot 10^{-5}$ to $1.18 \cdot 10^{-4}$
Cement – smoothed	0.3	$1.18 \cdot 10^{-2}$
Concrete - ordinary	0.6 - 1.5	$2.36 \cdot 10^{-2}$ to $5.91 \cdot 10^{-2}$
Concrete – precast	0.06 – 0.6	$2.36 \cdot 10^{-3}$ to $2.36 \cdot 10^{-2}$

Material	Roughness	
	(mm)	(inch)
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}
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

21.3 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
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

Material	C-factor	
	Possible values	Suggested value (NFPA13)
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

21.4 Physical Constants

All values are SI.

Atmospheric Pressure	101325 Pascal
Acceleration due to gravity	9.80665 ms ⁻²
0° C	273.15 Kelvin

21.5 Built-in schedules and fittings

PIPENET Transient Module offers a choice of built-in pipe schedules and fittings. These data cannot be modified by the User - if any changes are required please contact [Sunrise Systems Limited](#).

Built-in Pipe Schedules in Standard

[ANSI B36.10 Schedule 10](#)
[ANSI B36.10 Schedule 20](#)
[ANSI B36.10 Schedule 30](#)
[ANSI B36.10 Schedule S](#)
[ANSI B36.10 Schedule 40](#)
[ANSI B36.10 Schedule 60](#)
[ANSI B36.10 Schedule XS](#)
[ANSI B36.10 Schedule 80](#)
[ANSI B36.10 Schedule 100](#)
[ANSI B36.10 Schedule 120](#)
[ANSI B36.10 Schedule 140](#)
[ANSI B36.10 Schedule 160](#)
[ANSI B36.10 Schedule XXS](#)
[ANSI B36.19 Schedule 5S](#)
[ANSI B36.19 Schedule 10S](#)
[ANSI B36.19 Schedule S](#)
[ANSI B36.19 Schedule XS](#)
[ANSI B36.19 Schedule XXS](#)
[JIS 1990 Steel Schedule 10](#)
[JIS 1990 Steel Schedule 20](#)
[JIS 1990 Steel Schedule 30](#)
[JIS 1990 Steel Schedule 40](#)
[JIS 1990 Steel Schedule 60](#)
[JIS 1990 Steel Schedule 80](#)
[JIS 1990 Steel Schedule 100](#)
[JIS 1990 Steel Schedule 120](#)
[JIS 1990 Steel Schedule 140](#)
[JIS 1990 Steel Schedule 160](#)
[JIS 1990 Steel Schedule 5S](#)
[JIS 1990 Steel Schedule 10S](#)
[JIS 1990 Steel Schedule 40S](#)
[DIN Standard](#)

Built-in Pipe Schedules in Spray/Sprinkler

[BS 1387 Heavy](#)
[BS 1387 Medium](#)
[ANSI B36.10 Schedule 40](#)
[ANSI B36.10 Schedule 80](#)
[BS3505 Class D \(uPVC\)](#)
[BS.1221 Class C \(Cast Iron\)](#)
[Copper/Nickel \(90/10\) 14 bar g](#)
[Copper/Nickel \(90/10\) 20 bar g](#)

Built-in Linings Data in Spray/sprinkler

[Concrete](#)
[Asbestos](#)
[Epoxy](#)

Fittings

[Fittings K-Factor Values](#)
[Values for Entry and Exit to Large Cell or Pipe](#)

[Equivalent Lengths of Fittings for NFPA Rules](#)
[Equivalent Lengths of Fittings for FOC Rules](#)

21.6 ANSI B36.10 Schedule 10

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63				
20	0.75				
25	1.0				
32	1.25				
40	1.5				
50	2.0				
65	2.5				
80	3.0				
90	3.5				
100	4.0				
125	5.0				
150	6.0				
200	8.0				
250	10.0				
300	12.0				
350	14.0	342.9	13.5	358.14	14.1
400	16.0	393.7	15.5	406.4	16
450	18.0	444.5	17.5	457.2	18
500	20.0	495.3	19.5	508	20
600	24.0	596.9	23.5	609.6	24
700	28.0	695.452	27.38	711.2	28
750	30.0	746.252	29.38	762	30
800	32.0	797.052	31.38	812.8	32
850	34.0	847.852	33.38	863.6	34

21.7 ANSI B36.10 Schedule 20

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63				
20	0.75				
25	1.0				
32	1.25				
40	1.5				
50	2.0				
65	2.5				
80	3.0				
90	3.5				
100	4.0				
125	5.0				
150	6.0				
200	8.0	206.502	8.13	219.202	8.63
250	10.0	260.35	10.25	273.05	10.75
300	12.0	311.15	12.25	323.85	12.75
350	14.0	339.852	13.38	355.6	14
400	16.0	390.652	15.38	406.4	16
450	18.0	441.452	17.38	457.2	18
500	20.0	488.95	19.25	508	20
600	24.0	590.55	23.25	609.6	24
700	28.0	685.8	27	711.2	28
750	30.0	736.6	29	762	30
800	32.0	787.4	31	812.8	32
850	34.0	838.2	33	863.6	34

21.8 ANSI B36.10 Schedule 30

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63				
20	0.75				
25	1.0				
32	1.25				
40	1.5				
50	2.0				
65	2.5				
80	3.0				
90	3.5				
100	4.0				
125	5.0				
150	6.0				
200	8.0	204.978	8.07	219.202	8.63
250	10.0	257.556	10.14	273.05	10.75
300	12.0	307.086	12.09	323.85	12.75
350	14.0	336.55	13.25	355.6	14
400	16.0	387.35	15.25	406.4	16
450	18.0	435.102	17.13	457.2	18
500	20.0	482.6	19	508	20
600	24.0	581.152	22.88	609.6	24
700	28.0	679.45	26.75	711.2	28
750	30.0	730.25	28.75	762	30
800	32.0	781.05	30.75	812.8	32
850	34.0	831.85	32.75	863.6	34

21.9 ANSI B36.10 Schedule S

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25	9.144	0.36	13.716	0.54
8	0.31				
10	0.38	12.446	0.49	17.145	0.675
12	0.5	15.748	0.62	21.336	0.84
15	0.63				
20	0.75	20.828	0.82	26.67	1.05
25	1.0	26.67	1.05	33.401	1.315
32	1.25	35.052	1.38	42.164	1.66
40	1.5	40.894	1.61	48.26	1.9
50	2.0	52.578	2.07	60.325	2.375
65	2.5	62.378	2.47	73.025	2.875
80	3.0	77.978	3.07	88.9	3.5
90	3.5	90.17	3.55	101.6	4
100	4.0	102.362	4.03	114.3	4.5
125	5.0	128.27	5.05	141.3	5.563
150	6.0	154.178	6.07	168.275	6.625
200	8.0	202.692	7.98	219.075	8.625
250	10.0	254.508	10.02	273.05	10.75
300	12.0	304.8	12	323.85	12.75
350	14.0	336.55	13.25	355.6	14
400	16.0	387.35	15.25	406.4	16
450	18.0	438.15	17.25	457.2	18
500	20.0	488.95	19.25	508	20
600	24.0	590.55	23.25	609.6	24
700	28.0				
750	30.0	742.95	29.25	762	30
800	32.0	793.75	31.25	812.8	32
850	34.0	844.55	33.25	863.6	34

21.10 ANSI B36.10 Schedule 40

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25	9.2456	0.364	13.716	0.54
8	0.31				
10	0.38	12.5222	0.493	17.145	0.675
12	0.5	15.748	0.62	21.336	0.84
15	0.63				
20	0.75	20.828	0.82	26.67	1.05
25	1.0	26.67	1.05	33.528	1.32
32	1.25	35.052	1.38	42.164	1.66
40	1.5	40.894	1.61	48.26	1.9
50	2.0	52.578	2.07	60.452	2.38
65	2.5	62.738	2.47	73.152	2.88
80	3.0	77.978	3.07	88.9	3.5
90	3.5	90.17	3.55	101.6	4
100	4.0	102.362	4.03	114.3	4.5
125	5.0	128.27	5.05	141.224	5.56
150	6.0	154.178	6.07	168.402	6.63
200	8.0	202.692	7.98	219.202	8.63
250	10.0	254.508	10.02	273.05	10.75
300	12.0	303.276	11.94	323.85	12.75
350	14.0	333.502	13.13	355.6	14
400	16.0	381	15	406.4	16
450	18.0	428.752	16.88	457.2	18
500	20.0	477.774	18.81	508	20
600	24.0	574.802	22.63	609.6	24
700	28.0	676.402	26.63	711.2	28
750	30.0	723.9	28.5	762	30
800	32.0	777.748	30.62	812.8	32
850	34.0	828.548	32.62	863.6	34

21.11 ANSI B36.10 Schedule 60

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63				
20	0.75				
25	1.0				
32	1.25				
40	1.5				
50	2.0				
65	2.5				
80	3.0				
90	3.5				
100	4.0				
125	5.0				
150	6.0				
200	8.0	198.374	7.81	219.202	8.63
250	10.0	247.65	9.75	273.05	10.75
300	12.0	295.402	11.63	323.85	12.75
350	14.0	325.374	12.81	355.6	14
400	16.0	373.126	14.69	406.4	16
450	18.0	419.1	16.5	457.2	18
500	20.0	466.852	18.38	508	20
600	24.0	560.324	22.06	609.6	24
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.12 ANSI B36.10 Schedule XS

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25	7.62	0.3	13.716	0.54
8	0.31				
10	0.38	10.668	0.42	17.145	0.675
12	0.5	13.87	0.55	21.336	0.84
15	0.63				
20	0.75	18.796	0.74	26.67	1.05
25	1.0	24.384	0.96	33.401	1.315
32	1.25	32.512	1.28	42.164	1.66
40	1.5	38.1	1.5	48.26	1.9
50	2.0	49.276	1.94	60.325	2.375
65	2.5	58.928	2.32	73.025	2.875
80	3.0	73.66	2.9	88.9	3.5
90	3.5	85.344	3.36	101.6	4
100	4.0	97.282	3.83	114.3	4.5
125	5.0	122.174	4.81	141.3	5.563
150	6.0	146.304	5.76	168.275	6.625
200	8.0	193.675	7.625	219.075	8.625
250	10.0	247.65	9.75	273.05	10.75
300	12.0	298.45	11.75	323.85	12.75
350	14.0	330.2	13	355.6	14
400	16.0	381	15	406.4	16
450	18.0	431.8	17	457.2	18
500	20.0	482.6	19	508	20
600	24.0	584.2	23	609.6	24
700	28.0				
750	30.0	736.6	29	762	30
800	32.0	787.4	31	812.8	32
850	34.0	838.2	33	863.6	34

21.13 ANSI B36.10 Schedule 80

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25	7.6708	0.302	13.716	0.54
8	0.31				
10	0.38	10.7442	0.423	17.145	0.675
12	0.5	13.97	0.55	21.336	0.84
15	0.63				
20	0.75	18.796	0.74	26.67	1.05
25	1.0	24.384	0.96	33.528	1.32
32	1.25	35.052	1.38	42.164	1.66
40	1.5	38.1	1.5	48.26	1.9
50	2.0	49.276	1.94	60.452	2.38
65	2.5	58.928	2.32	73.152	2.88
80	3.0	73.66	2.9	88.9	3.5
90	3.5	85.344	3.36	101.6	4
100	4.0	97.282	3.83	114.3	4.5
125	5.0	122.174	4.81	141.224	5.56
150	6.0	146.304	5.76	168.402	6.63
200	8.0	193.802	7.63	219.202	8.63
250	10.0	242.824	9.56	273.05	10.75
300	12.0	289.052	11.38	323.85	12.75
350	14.0	317.5	12.5	355.6	14
400	16.0	363.474	14.31	406.4	16
450	18.0	409.702	16.13	457.2	18
500	20.0	455.676	17.94	508	20
600	24.0	547.624	21.56	609.6	24
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.14 ANSI B36.10 Schedule 100

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63				
20	0.75				
25	1.0				
32	1.25				
40	1.5				
50	2.0				
65	2.5				
80	3.0				
90	3.5				
100	4.0				
125	5.0				
150	6.0				
200	8.0	188.976	7.44	219.202	8.63
250	10.0	236.474	9.31	273.05	10.75
300	12.0	280.924	11.06	323.85	12.75
350	14.0	308.102	12.13	355.6	14
400	16.0	354.076	13.94	406.4	16
450	18.0	398.526	15.69	457.2	18
500	20.0	442.976	17.44	508	20
600	24.0	531.876	20.94	609.6	24
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.15 ANSI B36.10 Schedule 120

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63				
20	0.75				
25	1.0				
32	1.25				
40	1.5				
50	2.0				
65	2.5				
80	3.0				
90	3.5				
100	4.0	92.202	3.63	114.3	4.5
125	5.0	115.824	4.56	141.224	5.56
150	6.0	139.7	5.5	168.402	6.63
200	8.0	182.626	7.19	219.202	8.63
250	10.0	230.124	9.06	273.05	10.75
300	12.0	273.05	10.75	323.85	12.75
350	14.0	299.974	11.81	355.6	14
400	16.0	344.424	13.56	406.4	16
450	18.0	387.35	15.25	457.2	18
500	20.0	431.8	17	508	20
600	24.0	517.652	20.38	609.6	24
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.16 ANSI B36.10 Schedule 140

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63				
20	0.75				
25	1.0				
32	1.25				
40	1.5				
50	2.0				
65	2.5				
80	3.0				
90	3.5				
100	4.0				
125	5.0				
150	6.0				
200	8.0	177.8	7	219.202	8.63
250	10.0	222.25	8.75	273.05	10.75
300	12.0	266.7	10.5	323.85	12.75
350	14.0	292.1	11.5	355.6	14
400	16.0	333.502	13.13	406.4	16
450	18.0	377.952	14.88	457.2	18
500	20.0	419.1	16.5	508	20
600	24.0	504.952	19.88	609.6	24
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.17 ANSI B36.10 Schedule 160

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5	11.938	0.47	21.336	0.84
15	0.63				
20	0.75	15.494	0.61	26.67	1.05
25	1.0	20.828	0.82	33.528	1.32
32	1.25				
40	1.5	34.036	1.34	48.26	1.9
50	2.0	42.926	1.69	60.452	2.38
65	2.5	54.102	2.13	73.152	2.88
80	3.0	66.802	2.63	88.9	3.5
90	3.5				
100	4.0	87.376	3.44	114.3	4.5
125	5.0	109.474	4.31	141.224	5.56
150	6.0	131.826	5.19	168.402	6.63
200	8.0	172.974	6.81	219.202	8.63
250	10.0	215.9	8.5	273.05	10.75
300	12.0	257.302	10.13	323.85	12.75
350	14.0	284.226	11.19	355.6	14
400	16.0	325.374	12.81	406.4	16
450	18.0	315.976	12.44	457.2	18
500	20.0	407.924	16.06	508	20
600	24.0	490.474	19.31	609.6	24
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.18 ANSI B36.19 Schedule 5S

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5	18.034	0.71	21.336	0.84
15	0.63				
20	0.75	23.368	0.92	26.67	1.05
25	1.0	30.226	1.19	33.528	1.32
32	1.25	38.862	1.53	42.164	1.66
40	1.5	44.958	1.77	48.26	1.9
50	2.0	56.896	2.24	60.452	2.38
65	2.5	68.834	2.71	73.152	2.88
80	3.0	84.582	3.33	88.9	3.5
90	3.5	97.282	3.83	101.6	4
100	4.0	109.982	4.33	114.3	4.5
125	5.0	135.89	5.35	141.224	5.56
150	6.0	162.814	6.41	168.402	6.63
200	8.0	213.614	8.41	219.202	8.63
250	10.0	266.192	10.48	273.05	10.75
300	12.0	315.976	12.44	323.85	12.75
350	14.0	347.726	13.69	355.6	14
400	16.0	398.018	15.67	406.4	16
450	18.0	448.818	17.67	457.2	18
500	20.0	498.348	19.62	508	20
600	24.0	598.424	23.56	609.6	24
700	28.0	700.024	27.56	711.2	28
750	30.0	749.3	29.5	762	30
800	32.0	800.1	31.5	812.8	32
850	34.0	850.9	33.5	863.6	34

21.19 ANSI B36.10 Schedule XXS

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5	6.35	0.25	21.336	0.84
15	0.63				
20	0.75	10.922	0.43	26.67	1.05
25	1.0	15.24	0.6	33.401	1.32
32	1.25	22.86	0.9	42.164	1.66
40	1.5	27.94	1.1	48.26	1.9
50	2.0	38.1	1.5	60.325	2.38
65	2.5	44.958	1.77	73.025	2.88
80	3.0	58.42	2.3	88.9	3.5
90	3.5				
100	4.0	80.01	3.15	114.3	4.5
125	5.0	103.124	4.06	141.3	5.56
150	6.0	124.46	4.9	168.275	6.63
200	8.0	174.752	6.88	219.075	8.63
250	10.0	222.25	8.75	273.05	10.75
300	12.0	273.05	10.75	323.85	12.75
350	14.0				
400	16.0				
450	18.0				
500	20.0				
600	24.0				
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.20 ANSI B36.19 Schedule 10S

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25	10.414	0.41	13.716	0.54
8	0.31				
10	0.38	13.843	0.545	17.145	0.675
12	0.5	17.018	0.67	21.336	0.84
15	0.63				
20	0.75	22.352	0.88	26.67	1.05
25	1.0	27.94	1.1	33.528	1.32
32	1.25	32.6268	1.442	42.164	1.66
40	1.5	42.672	1.68	48.26	1.9
50	2.0	54.864	2.16	60.452	2.38
65	2.5	66.802	2.63	73.152	2.88
80	3.0	82.804	3.26	88.9	3.5
90	3.5	95.504	3.76	101.6	4
100	4.0	108.204	4.26	114.3	4.5
125	5.0	134.62	5.3	141.224	5.56
150	6.0	161.544	6.36	168.402	6.63
200	8.0	211.582	8.33	219.202	8.63
250	10.0	264.668	10.42	273.05	10.75
300	12.0	314.706	12.39	323.85	12.75
350	14.0	345.948	13.62	355.6	14
400	16.0	396.748	15.62	406.4	16
450	18.0	447.548	17.62	457.2	18
500	20.0	496.824	19.56	508	20
600	24.0	596.9	23.5	609.6	24
700	28.0				
750	30.0	746.252	29.38	762	30
800	32.0				
850	34.0				

21.21 ANSI B36.19 Schedule S

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25	9.246	0.364	13.716	0.54
8	0.31				
10	0.38	12.5222	0.493	17.145	0.675
12	0.5	15.748	0.62	21.336	0.84
15	0.63				
20	0.75	20.828	0.82	26.67	1.05
25	1.0	26.67	1.05	33.528	1.32
32	1.25	35.052	1.38	42.164	1.66
40	1.5	40.894	1.61	48.26	1.9
50	2.0	52.578	2.07	60.452	2.38
65	2.5	62.738	2.47	73.152	2.88
80	3.0	77.978	3.07	88.9	3.5
90	3.5	90.17	3.55	101.6	4
100	4.0	102.362	4.03	114.3	4.5
125	5.0	128.27	5.05	141.224	5.56
150	6.0	154.178	6.07	168.402	6.63
200	8.0	202.692	7.98	219.202	8.63
250	10.0	254.508	10.02	273.05	10.75
300	12.0	304.8	12	323.85	12.75
350	14.0	336.55	13.25	355.6	14
400	16.0	387.35	15.25	406.4	16
450	18.0	438.15	17.25	457.2	18
500	20.0	488.95	19.25	508	20
600	24.0	590.55	23.25	609.6	24
700	28.0	692.15	27.25	711.2	28
750	30.0	742.95	29.25	762	30
800	32.0	793.75	31.25	812.8	32
850	34.0	844.55	33.25	863.6	34

21.22 ANSI B36.19 Schedule XS

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25	7.6708	0.302	13.716	0.54
8	0.31				
10	0.38	10.7442	0.423	17.145	0.675
12	0.5	13.97	0.55	21.336	0.84
15	0.63				
20	0.75	18.796	0.74	26.67	1.05
25	1.0	24.384	0.96	33.528	1.32
32	1.25	32.4612	1.278	42.164	1.66
40	1.5	38.1	1.5	48.26	1.9
50	2.0	49.276	1.94	60.452	2.38
65	2.5	58.928	2.32	73.152	2.88
80	3.0	73.66	2.9	88.9	3.5
90	3.5	85.344	3.36	101.6	4
100	4.0	97.282	3.83	114.3	4.5
125	5.0	122.174	4.81	141.224	5.56
150	6.0	146.304	5.76	168.402	6.63
200	8.0	193.802	7.63	219.202	8.63
250	10.0	247.65	9.75	273.05	10.75
300	12.0	298.45	11.75	323.85	12.75
350	14.0	330.2	13	355.6	14
400	16.0	381	15	406.4	16
450	18.0	431.8	17	457.2	18
500	20.0	482.6	19	508	20
600	24.0	584.2	23	609.6	24
700	28.0	685.8	27	711.2	28
750	30.0	736.6	29	762	30
800	32.0	787.4	31	812.8	32
850	34.0	838.2	33	863.6	34

21.23 ANSI B36.19 Schedule XXS

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5	6.35	0.25	21.336	0.84
15	0.63				
20	0.75	10.922	0.43	26.67	1.05
25	1.0	15.24	0.6	33.528	1.32
32	1.25				
40	1.5	27.94	1.1	48.26	1.9
50	2.0	38.1	1.5	60.452	2.38
65	2.5	44.958	1.77	73.152	2.88
80	3.0	58.42	2.3	88.9	3.5
90	3.5	69.342	2.73	101.6	4
100	4.0	80.01	3.15	114.3	4.5
125	5.0	103.124	4.06	141.224	5.56
150	6.0	124.46	4.9	168.402	6.63
200	8.0	174.752	6.88	219.202	8.63
250	10.0	222.25	8.75	273.05	10.75
300	12.0	273.05	10.75	323.85	12.75
350	14.0	323.85	12.75	355.6	14
400	16.0				
450	18.0				
500	20.0				
600	24.0				
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.24 JIS 1990 Steel Schedule 10

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63				
20	0.75				
25	1.0				
32	1.25				
40	1.5				
50	2.0				
65	2.5				
80	3.0				
90	3.5				
100	4.0				
125	5.0				
150	6.0				
200	8.0				
250	10.0				
300	12.0				
350	14.0	342.801	13.4961	355.6	14
400	16.0	393.601	15.4961	406.4	16
450	18.0	444.401	17.4961	457.2	18
500	20.0	495.201	19.4961	508	20
600	24.0	596.801	23.4961	609.6	24
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.25 JIS 1990 Steel Schedule 20

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63				
20	0.75				
25	1.0				
32	1.25				
40	1.5				
50	2.0	54.1	2.12992	60.5	2.38189
65	2.5	67.3001	2.64961	76.3001	3.00394
80	3.0	80.0999	3.15354	89.0999	3.50787
90	3.5	92.6	3.64567	101.6	4
100	4.0	104.5	4.11417	114.3	4.5
125	5.0	129.6	5.10236	139.8	5.50394
150	6.0	154.2	6.07087	165.2	6.50394
200	8.0	203.5	8.01181	216.3	8.51575
250	10.0	254.599	10.0236	267.401	10.5276
300	12.0	305.699	12.0354	318.501	12.5394
350	14.0	339.801	13.378	355.6	14
400	16.0	390.601	15.378	406.4	16
450	18.0	441.401	17.378	457.2	18
500	20.0	489.001	19.252	508	20
600	24.0	590.601	23.252	609.6	24
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.26 JIS 1990 Steel Schedule 30

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63				
20	0.75				
25	1.0				
32	1.25				
40	1.5				
50	2.0				
65	2.5				
80	3.0				
90	3.5				
100	4.0				
125	5.0				
150	6.0				
200	8.0	202.3	7.96457	216.3	8.51575
250	10.0	251.8	9.91339	267.401	10.5276
300	12.0	301.701	11.878	318.501	12.5394
350	14.0	336.601	13.252	355.6	14
400	16.0	387.401	15.252	406.4	16
450	18.0	435	17.126	457.2	18
500	20.0	482.6	19	508	20
600	24.0	581	22.874	609.6	24
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.27 JIS 1990 Steel Schedule 40

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63	16.1	0.63385	21.7	0.85433
20	0.75	21.4	0.84252	27.2001	1.07087
25	1.0	27.2001	1.07087	33.9999	1.33858
32	1.25	35.5001	1.39764	42.6999	1.6811
40	1.5	41.2001	1.62205	48.6001	1.91339
50	2.0	52.6999	2.0748	60.5	2.38189
65	2.5	65.9	2.59449	76.3001	3.00394
80	3.0	78.0999	3.0748	89.0999	3.50787
90	3.5	90.2	3.55118	101.6	4
100	4.0	102.3	4.02756	114.3	4.5
125	5.0	126.6	4.98425	139.8	5.50394
150	6.0	151	5.94488	165.2	6.50394
200	8.0	199.9	7.87008	216.3	8.51575
250	10.0	248.8	9.79528	267.401	10.5276
300	12.0	287.899	11.7283	318.501	12.5394
350	14.0	333.4	13.126	355.6	14
400	16.0	381	15	406.4	16
450	18.0	428.6	16.874	457.2	18
500	20.0	477.799	18.811	508	20
600	24.0	574.599	22.622	609.6	24
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.28 JIS 1990 Steel Schedule 60

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63	15.3	0.60236	21.7	0.85433
20	0.75	20.4	0.80315	27.2001	1.07087
25	1.0	26.2001	1.0315	33.9999	1.33858
32	1.25	33.7	1.32677	42.6999	1.6811
40	1.5	39.6001	1.55906	48.6001	1.91339
50	2.0	50.999	1.99606	60.5	2.38189
65	2.5	64.3001	2.5315	76.3001	3.00394
80	3.0	75.9	2.98819	89.0999	3.50787
90	3.5	87.6	3.44882	101.6	4
100	4.0	100.1	3.94094	114.3	4.5
125	5.0	123.6	4.86614	139.8	5.50394
150	6.0	146.6	5.77165	165.2	6.50394
200	8.0	195.7	7.70472	216.3	8.51575
250	10.0	242	9.52756	267.401	10.5276
300	12.0	289.9	11.4134	318.501	12.5394
350	14.0	325.399	12.811	355.6	14
400	16.0	372.999	14.685	406.4	16
450	18.0	419.199	16.5039	457.2	18
500	20.0	466.801	18.378	508	20
600	24.0	560.4	22.063	609.6	24
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.29 JIS 1990 Steel Schedule 80

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63	14.3	0.56299	21.7	0.85433
20	0.75	19.4	0.76378	27.2001	1.07087
25	1.0	25	0.98425	33.9999	1.33858
32	1.25	32.9001	1.29528	42.6999	1.6811
40	1.5	38.4	1.51181	48.6001	1.91339
50	2.0	49.5	1.94882	60.5	2.38189
65	2.5	62.3001	2.45276	76.3001	3.00394
80	3.0	73.9	2.90945	89.0999	3.50787
90	3.5	85.3999	3.3622	101.6	4
100	4.0	97.0999	3.82283	114.3	4.5
125	5.0	120.8	4.75591	139.8	5.50394
150	6.0	143.2	5.6378	165.2	6.50394
200	8.0	190.9	7.51575	216.3	8.51575
250	10.0	237.2	9.33858	267.401	10.5276
300	12.0	283.7	11.1693	318.501	12.5394
350	14.0	317.599	12.5039	355.6	14
400	16.0	363.601	14.315	406.4	16
450	18.0	409.6	16.126	457.2	18
500	20.0	455.6	17.937	508	20
600	24.0	547.601	21.5591	609.6	24
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.30 JIS 1990 Steel Schedule 100

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63				
20	0.75				
25	1.0				
32	1.25				
40	1.5				
50	2.0				
65	2.5				
80	3.0				
90	3.5				
100	4.0				
125	5.0				
150	6.0				
200	8.0	186.1	7.32677	216.3	8.51575
250	10.0	231	9.09449	267.401	10.5276
300	12.0	275.699	10.8543	318.501	12.5394
350	14.0	308	12.126	355.6	14
400	16.0	354	13.937	406.4	16
450	18.0	398.399	15.685	457.2	18
500	20.0	442.999	17.4409	508	20
600	24.0	531.8	20.937	609.6	24
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.31 JIS 1990 Steel Schedule 120

Surface roughness 0.04572mm (0.0017992 ins)
Young's Modulus 227.91 GPa
Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63				
20	0.75				
25	1.0				
32	1.25				
40	1.5				
50	2.0				
65	2.5				
80	3.0				
90	3.5				
100	4.0	92.0999	3.62598	114.3	4.5
125	5.0	114.4	4.50394	139.8	5.50394
150	6.0	136.6	5.37795	165.2	6.50394
200	8.0	179.9	7.08268	216.3	8.51575
250	10.0	224.6	8.84252	267.401	10.5276
300	12.0	267.701	10.5394	318.501	12.5394
350	14.0	299.999	11.811	355.6	14
400	16.0	344.599	13.5669	406.4	16
450	18.0	387.401	15.252	457.2	18
500	20.0	431.8	17	508	20
600	24.0	517.601	20.378	609.6	24
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.32 JIS 1990 Steel Schedule 140

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63				
20	0.75				
25	1.0				
32	1.25				
40	1.5				
50	2.0				
65	2.5				
80	3.0				
90	3.5				
100	4.0				
125	5.0				
150	6.0				
200	8.0	175.1	6.8937	216.3	8.51575
250	10.0	216.6	8.52756	267.401	10.5276
300	12.0	261.3	10.2874	318.501	12.5394
350	14.0	292.001	11.4961	355.6	14
400	16.0	333.4	13.126	406.4	16
450	18.0	377.8	14.874	457.2	18
500	20.0	419.199	16.5039	508	20
600	24.0	504.8	19.874	609.6	24
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.33 JIS 1990 Steel Schedule 160

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63	12.3	0.4843	21.7	0.8543
20	0.75	16.2	0.6378	27.2	1.0709
25	1.0	21.2	0.8346	34.0	1.3386
32	1.25	29.9	1.1772	42.7	1.6811
40	1.5	34.4	1.3543	48.6	1.9134
50	2.0	43.1	1.6969	60.5	2.3819
65	2.5	57.3	2.2559	76.3	3.0039
80	3.0	66.9	2.6339	89.1	3.5079
90	3.5	76.2	3.0000	101.6	4.0000
100	4.0	87.3	3.4370	114.3	4.5000
125	5.0	108	4.2520	139.8	5.5039
150	6.0	128.8	5.0709	165.2	6.5039
200	8.0	170.3	6.7047	216.3	8.5158
250	10.0	210.2	8.2756	267.4	10.5276
300	12.0	251.9	9.9173	318.5	12.5394
350	14.0	284.2	11.189	355.6	14
400	16.0	325.4	12.811	406.4	16
450	18.0	366.8	14.441	457.2	18
500	20.0	408.0	16.063	508.0	20
600	24.0	490.6	19.315	609.6	24
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.34 JIS 1990 Steel Schedule 5S

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63				
20	0.75				
25	1.0	30.7	1.20866	33.9999	1.33858
32	1.25				
40	1.5				
50	2.0	57.2	2.25197	60.5	2.38189
65	2.5				
80	3.0	84.9	3.34252	89.0999	3.50787
90	3.5				
100	4.0	110.1	4.33465	114.3	4.5
125	5.0	134.2	5.28346	139.8	5.50394
150	6.0	159.6	6.28346	165.2	6.50394
200	8.0	210.7	8.29528	216.3	8.51575
250	10.0	260.599	10.2598	267.401	10.5276
300	12.0	310.5	12.2244	318.501	12.5394
350	14.0				
400	16.0				
450	18.0				
500	20.0				
600	24.0				
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.35 JIS 1990 Steel Schedule 10S

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63				
20	0.75				
25	1.0	28.4	1.11811	33.999	1.33858
32	1.25				
40	1.5				
50	2.0	54.9001	2.16142	60.5	2.38189
65	2.5				
80	3.0	83.0999	3.27165	89.0999	3.50787
90	3.5				
100	4.0	108.3	4.26378	114.3	4.5
125	5.0	133	5.23622	139.8	5.50394
150	6.0	158.4	6.23622	165.2	6.50394
200	8.0	208.3	8.20079	216.3	8.51575
250	10.0	259.4	10.2126	267.401	10.5276
300	12.0	309.499	12.185	318.501	12.5394
350	14.0				
400	16.0				
450	18.0				
500	20.0				
600	24.0				
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.36 JIS 1990 Steel Schedule 40S

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63				
20	0.75				
25	1.0	27.9999	1.10236	33.9999	1.33858
32	1.25				
40	1.5				
50	2.0	53.5	2.1063	60.5	2.38189
65	2.5				
80	3.0	81.0999	3.19291	89.0999	3.50787
90	3.5				
100	4.0	106.3	4.18504	114.3	4.5
125	5.0	129.8	5.11024	139.8	5.50394
150	6.0	155.2	6.11024	165.2	6.50394
200	8.0	203.3	8.00394	216.3	8.51575
250	10.0	254.399	10.0157	267.401	10.5276
300	12.0	305.501	12.0276	318.501	12.5394
350	14.0				
400	16.0				
450	18.0				
500	20.0				
600	24.0				
700	28.0				
750	30.0				
800	32.0				
850	34.0				

21.37 DIN Standard

Surface roughness 0.04572mm (0.0017992 ins)

Young's Modulus 227.91 GPa

Poisson's Ratio 0.4

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
6	0.25				
8	0.31				
10	0.38				
12	0.5				
15	0.63	17.3	0.68110	21.3	0.83858
20	0.75	22.3	0.87795	26.9001	1.05906
25	1.0	28.5001	1.12205	33.7	1.32677
32	1.25	37.2001	1.46457	42.4	1.66929
40	1.5	43.1	1.69685	48.2999	1.90157
50	2.0	54.5	2.14567	60.3001	2.37402
65	2.5	70.3001	2.76772	76.0999	2.99606
80	3.0	82.5	3.24803	88.9	3.5
90	3.5				
100	4.0	107.1	4.21654	114.3	4.5
125	5.0	131.7	5.18504	139.7	5.5
150	6.0	159.3	6.27165	168.3	6.62598
200	8.0	206.5	8.12992	219.1	8.62598
250	10.0	260.401	10.252	272.999	10.748
300	12.0	309.7	12.1929	323.901	12.752
350	14.0	339.601	13.3701	355.6	14
400	16.0	389.001	15.315	406.601	16.0079
450	18.0				
500	20.0	486.001	19.1339	508	20
600	24.0	584.599	23.0157	609.6	24
700	28.0	696.999	27.4409	711.2	28
750	30.0				
800	32.0	796.801	31.3701	812.8	32
850	34.0				

21.38 Pipe Schedule 1 (SPRAY) BS 1387 Heavy

Surface roughness 0.04572mm (0.0018 ins)

Young's Modulus 203.4 GPa

Poisson's Ratio 0.3

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
350	14.0				
400	16.0				
450	18.0				
500	20.0				
600	24.0				
700	28.0				
750	30.0				
800	32.0				
850	34.0				
900	36.0				

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. This data is not part of BS.1387.

21.39 Pipe Schedule 2 (SPRAY) BS 1387 Medium

Surface roughness 0.04572mm (0.0018 ins)
Young's Modulus 203.4 GPa
Poisson's Ratio 0.3

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
350	14.0				
400	16.0				
450	18.0				
500	20.0				
600	24.0				
700	28.0				
750	30.0				
800	32.0				
850	34.0				
900	36.0				

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. This data is not part of BS.1387.

21.40 Pipe Schedule 3 (SPRAY) ANSI B36.10 Schedule 40

Surface roughness 0.04572mm (0.0018 ins)
Young's Modulus 203.4 GPa
Poisson's Ratio 0.3

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
700	28.0				
750	30.0				
800	32.0				
850	34.0				
900	36.0				

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. This data is not part of BS.1387.

21.41 Pipe Schedule 4 (SPRAY) ANSI B36.10 Schedule 80

Surface roughness 0.04572mm (0.0018 ins)
Young's Modulus 203.4 GPa
Poisson's Ratio 0.3

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
700	28.0				
750	30.0				
800	32.0				
850	34.0				
900	36.0				

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. This data is not part of BS.1387.

21.42 Pipe Schedule 5 (SPRAY) BS3505 Class D

Surface roughness 0.001524 mm (0.00006 ins)

Young's Modulus 3.0 GPa

Poisson's Ratio 0.3

Nominal Diameter		Internal Diameter		Outside Diameter	
mm	in	mm	in	mm	in
5	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
500	20.0				
600	24.0				
700	28.0				
750	30.0				
800	32.0				
850	34.0				
900	36.0				

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; however, these have been retained for users' convenience.

21.43 Pipe Schedule 6 (SPRAY) BS.1221 Class C

Surface roughness 0.254mm (0.01 ins)
Young's Modulus 180 GPa
Poisson's Ratio 0.3

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	560.324	22.06
600	24.0	629.412	24.78	667.004	26.26
700	28.0				
750	30.0				
800	32.0				
850	34.0				
900	36.0				

Notes

1. Imperial sizes taken from BS.1211-(1958)
2. This standard does not include metric nominal sizes; however, these have been retained for users' convenience.

21.44 Pipe Schedule 7 (SPRAY) BS.2871

Surface roughness 0.001534 mm (0.00006 ins)

Young's Modulus 140.0 GPa

Poisson's Ratio 0.3

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
700	28.0				
750	30.0				
800	32.0				
850	34.0				
900	36.0				

Notes

1. Outside diameters to BS.2871, Part 2, Table 3; BSMA 18, Table 2; BSMA 60. Wall thickness' conform to Lloyds offshore requirements.
2. Imperial sizes are direct conversions of metric data.
3. Nominal sizes are included for users' convenience.

21.45 Pipe Schedule 8 (SPRAY) BS.2871

Surface roughness	0.001534 mm (0.00006 ins)
Young's Modulus	140.0 GPa
Poisson's Ratio	0.3

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
700	28.0				
750	30.0				
800	32.0				
850	34.0				
900	36.0				

Notes

1. Outside diameters to BS.2871, Part 2, Table 3; BSMA 18, Table 2; BSMA 60. Wall thickness' conform to Lloyds offshore requirements.
2. Imperial sizes are direct conversions of metric data.
3. Nominal sizes are included for users' convenience.

21.46 Lining Type

Lining Type	Lining Material	Surface Roughness (mm)	Surface Roughness (inches)
1	Concrete	0.030	1.181x10 ⁻³
2	Asbestos	0.015	5.906x10 ⁻³
3	Epoxy	0.003	1.181x10 ⁻³

21.47 K-factor - Type of Fitting or Valve

Type of Fitting or Valve	K-Factor
90° Bend	0.75
180° Bend; Close return	1.50
T (STAND Along run) Branch blanked off	0.40
T used as Pipe entering run or entering branch	1.00
Gate Valve - Open	0.17
Diaphragm - Open	2.30
Globe Valve - Open	6.00

21.48 Entry and Exit K-factors

Values for Entry and Exit to Large Cell or Pipe in Standard option

Entrance to large space:

$$K = \left(1 - \frac{A_1}{A_0}\right)^2$$

Exit from large space:

A ₁ /A ₂	K
0.0	0.50
0.2	0.45
0.4	0.36
0.6	0.21
0.8	0.07
1.0	0.00

A₁ = Cross-sectional area of smaller duct.

A_2 = Cross-sectional area of larger duct.

21.49 Equivalent Lengths (FOC)

Code	Fitting Type	Nominal pipe size					
		mm in	15.0 0.5	20.0 0.75	25.0 1.0	32.0 1.25	40.0 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

Code	Fitting Type	Nominal pipe size					
		mm in	50.0 2.0	65.0 2.5	80.0 3.0	90.0 3.5	100.0 4.0
1	45° Screwed Elbow		0.76	1.02	1.27	1.44	1.61
2	90° Screwed Elbow (Standard)		1.46	1.89	2.37	2.71	3.04
3	90° Welded Elbow (Long Radius)		0.69	0.88	1.1	1.26	1.43
4	Screwed Tee or Cross (Flow turned 90°)		2.91	3.81	4.75	5.42	6.1
5	Gate Valve (Straightway)		0.38	0.51	0.63	0.72	0.81
6	Alarm or Non-Return Valve (Swinging)		2.42	3.18	3.94	4.5	5.07
7	Alarm or Non-Return Valve (Mushroom)		12.08	18.91	19.71	22.54	25.36
8	Globe Valve (Straightway)		16.43	21.64	26.8	30.64	34.48
9	Butterfly Valve		2.19	2.86	3.55	4.06	4.56

Code	Fitting Type	Nominal pipe size					
		mm in	125 5.0	150 6.0	200 8.0	250 10.0	300-900 12-36
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. 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.
5. Equivalent lengths of 90mm fittings are the arithmetic means of the corresponding 80mm and 100mm fittings.
6. All fittings above 250mm (10in) are assumed to have the same equivalent length as the corresponding 250mm fitting.

21.50 Equivalent Lengths (NFPA)

Code	Fitting Type	Nominal Pipe Size					
		mm in	15.0 0.5	20.0 0.75	25.0 1.0	32.0 1.25	40.0 1.5
1	Standard 45° Elbow		0.3048	0.3048	0.3048	0.3048	0.6096
2	Standard 90° Elbow		0.6096	0.6096	0.9144	0.9144	1.2192
3	Long radius Elbow		0.3048	0.6096	0.6096	0.6096	0.6096
4	Tee or Cross (Flow turned 90°)		1.2192	1.524	1.8288	1.8288	2.4384
5	Gate Valve		0.3048	0.3048	0.3048	0.3048	0.3048
6	Swing Check Valve		1.2192	1.524	2.1336	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

Code	Fitting Type	Nominal Pipe Size					
		mm in	125.0 5.0	150.0 6.0	200.0 8.0	250.0 10.0	300 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°)		7.62	9.144	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		10.0	12.200	15.300	19.800	23.000
8	Ball Valve		5.100	6.7100	10.970	12.800	15.240
9	Butterfly Valve		2.7432	3.0480	3.6576	5.7912	6.4008

Code	Fitting Type	Nominal pipe size					
		mm in	360.0 14.0	400.0 16.0	450.0 18.0	500.0 20.0	600.0 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

Code	Fitting Type	Nominal pipe size					
		mm in	700.0 28.0	750.0 30.0	800.0 32.0	850.0 34.0	900.0 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	104.553
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. 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 -7-2 (g).
3. For NFPA 1996 rules the above values only apply to internal diameters identical to the Schedule 40 steel pipe. For other internal diameters the equivalent lengths are scaled in accordance with NFPA 1996 rules Table 5-5.2.1 (Note 2).

4. Equivalent lengths of 15mm fittings are assumed the same as the corresponding 20mm fittings.
5. Equivalent lengths of gate and butterfly valves below 50mm are assumed the same as the 50mm values.
6. Equivalent lengths for pipe sizes above 600mm have been obtained by extrapolation from equivalent lengths for the smaller pipe sizes.

Index

- . -

.DXF 106
 .EMF 106
 .frc 179
 .out 179
 .res 179
 .WMF 106

- < -

<Any> 32

- A -

Absolute tolerances 177
 Accumulator 261
 Add multiple pipes 140
 Adding waypoints 107
 American Petroleum Institute 165, 204
 Analogue sensors 284
 ANSI B36.10 Schedule 10 318
 ANSI B36.10 Schedule 100 326
 ANSI B36.10 Schedule 120 327
 ANSI B36.10 Schedule 140 328
 ANSI B36.10 Schedule 160 329
 ANSI B36.10 Schedule 20 319
 ANSI B36.10 Schedule 30 320
 ANSI B36.10 Schedule 40 322
 ANSI B36.10 Schedule 60 323
 ANSI B36.10 Schedule 80 325
 ANSI B36.10 Schedule S 321
 ANSI B36.10 Schedule XS 324
 ANSI B36.10 Schedule XXS 331
 ANSI B36.19 Schedule 10S 332
 ANSI B36.19 Schedule 5S 330
 ANSI B36.19 Schedule S 333
 ANSI B36.19 Schedule XS 334
 ANSI B36.19 Schedule XXS 335
 Anti-Windup 289
 API 204

API fluid 165
 Area tool 108
 Asymmetric Pulse 208
 AutoCAD 302
 Autosave 20

- B -

Background colour 168
 Background colours 134
 Basic Errors 306
 Bibliography 312
 Bursting disc 255

- C -

calculation Go 26
 Calculation menu 26
 Calculation options... 26
 Calculation time step 216
 Calculation timesteps 89
 Calculation toolbar 31
 Cascade PID Controller 292
 Cavitation 95
 Cell shading 120
 C-factor 204, 314
 Changing units 159
 Channel cavitation 96
 Check valve 246
 Choosing the graphical timestep 92
 Closing a file 20
 Colebrook-White 214
 Colour schemes 130
 Colouration menu 26
 Complex colouring rules 130
 Complex force 86
 Complex links 26
 Complex nodes 26
 Component properties 16
 Component tools 112
 Compressible flow pipe 222
 Constant gradient 100
 Constant specification 208
 Contacting Sunrise Systems 4
 Continuity equation 206
 Control switch 295

Control Systems 283
 Control valve library 149
 Control volumes 86
 Controls 177
 Copy 22, 107
 Copy/Paste 107
 Copying cells 122
 Copying selected items 108
 Coulson and Richardson 203, 204, 214
 Crane 312
 Creating a path 136
 Crosses 223
 Cubic Profile 208
 Cubic pump curve 225
 Cubic spline 225
 Cursor keys 115
 Cut 22

- D -

D controllers 289
 Damped sine wave 208
 Darcy equation 214
 Data page 120
 Defaults options 25
 Defining Forces 175
 DEfining tables 173
 Degeneration factor 225
 Digital sensors 284
 DIN Standard 349
 Display options 25, 168
 Displaying Results 176
 Drawing components 112
 DXF 302
 Dynamic forces 86

- E -

Edit menu 22
 Editing a cell 120
 Editing system libraries 156
 Elastic connections 86
 Elastic pipes 214
 Elevation Profile Window 136
 Elevation tolerance 177
 Enter milestone data 27

Entry and Exit K-factors 358
 Equal Percentage Opening Valve 241
 Equipment items 223
 Equivalent Lengths (FOC) 360
 Equivalent lengths 223
 Equivalent Lengths (NFPA) 362
 Errors 306
 Examples directory 3
 Exit the program 20
 Expired licence 8
 Expiry date checking 8
 Expiry date licencing 8
 Explode node 107
 Exponential 208
 Export 20
 Export clipboard 27
 Export DXF file 27
 Export HP-GL/2 27
 Exporting libraries 144
 Exporting the schematic 115, 302
 Exporting to an HPGL file 115

- F -

Fanning friction factor 203
 Fast components 92
 File menu 20
 Find button 32
 Finding components 32
 Fittings 24, 223
 Fittings library 147
 Fittings window 19
 Flow coefficient valve 241
 Flow sensor 284
 Flow switch 295
 Fluid 163
 Fluid calculation 163
 Fluid damped check valve 248
 Fluid equations 206
 Fluid option 25
 Fluid parameters 204
 Fluids 204
 Font sizes 168
 Force on a 90 degree bend 70
 Forces 86
 Forces extrema 80

Forces page 120

- G -

Gain 289
 General Pressure Loss 155
 Global errors 307
 Graph Viewer 50, 184
 Graph Viewer annotation 196
 Graph Viewer curve properties 190
 Graph Viewer data points 197
 Graph Viewer limit lines properties 194
 Graph Viewer properties 189
 Graph Viewer types 187
 Graph Viewer x axis properties 193
 Graph Viewer y axis properties 192
 Graph Viewer zoom 198
 Graphical timestep 92
 Graphs page 120
 Graphs... 26
 Grid display options 168

- H -

Hazeni-Williamms 202
 Hazen-Williams 204, 214
 Hazen-Williams coefficients 314
 Height Checking 126
 Help topics 28
 Henri Darcy 203
 HPGL 115
 HP-GL/2 302
 H-Q curve 233

- I -

I controllers 289
 Import graphical underlay 23
 Import library 20
 Importing libraries 144
 Inertial check valve 254
 Inertial pump 233
 Initial Guess File 179, 213
 Initial state 179, 213
 Input errors 307
 Installation 8

Interpolation tolerance 177
 Interpreting force results 86
 Invalid components 38, 112, 126
 Invert 22

- J -

JIS 1990 Steel Schedule 10 336
 JIS 1990 Steel Schedule 100 342
 JIS 1990 Steel Schedule 10S 347
 JIS 1990 Steel Schedule 120 343
 JIS 1990 Steel Schedule 140 344
 JIS 1990 Steel Schedule 160 345
 JIS 1990 Steel Schedule 20 337
 JIS 1990 Steel Schedule 30 338
 JIS 1990 Steel Schedule 40 339
 JIS 1990 Steel Schedule 40S 348
 JIS 1990 Steel Schedule 5S 346
 JIS 1990 Steel Schedule 60 340
 JIS 1990 Steel Schedule 80 341

- K -

Key status 28
 K-factor - Fittings in Standard option 358
 K-factor valve 241

- L -

Label display options 168
 Labelling the diagram 34
 Labels 34
 Laminar flow 203, 214
 Libraries 144
 Libraries menu 24
 License 8
 Licensing 9
 Limiting power ramp 287
 Line thickness 168
 Linear Opening Valve 241
 Linear Profile 208
 Lining type 358
 Linings 154, 204
 Liquid surge relief valve 250
 Local user library 144

- M -

Make a path 27
 Mass flow 159
 Maximum and minimum pressure 181
 Maximum number of iterations 177
 Maximum/minimum forces 86
 Mile Post Data 100
 Mirror 22
 Model options 158
 Moment of inertia 229
 Momentum equation 206
 Moody diagram 203
 Moving and editing text 111
 Moving around the network 115

- N -

Next button 32
 Non-return valve 245
 Numerical errors 310

- O -

Obtaining support 4
 One pipe system 34
 One-node caisson 275
 Open library 20
 Opening a file 20
 Opening libraries 144
 Operating valve 244
 Options menu 25
 Options toolbar 30
 Output 179
 Output menu 26
 Output report 50
 Output timesteps 89
 Overall appearance 14
 Overview window 19

- P -

P controllers 289
 Pan 108
 Panning with the mouse 116

Paste 22, 107
 Paste (Incl. layout) 107
 Paste in column 22
 Physical constants 315
 PID controller 289
 Pipe Bundle 217
 Pipe length 89
 Pipe Schedule 1 (SPRAY) BS 1387 Heavy 350
 Pipe Schedule 2 (SPRAY) BS 1387 Medium 351
 Pipe Schedule 3 (SPRAY) ANSI B36.10 Schedule 40 351
 Pipe Schedule 4 (SPRAY) ANSI B36.10 Schedule 80 353
 Pipe Schedule 5 (SPRAY) BS3505 Class D 354
 Pipe Schedule 6 (SPRAY) BS.1221 Class C (cast iron) 355
 Pipe Schedule 7 (SPRAY) BS.2871 Copper/Nickel (90/10) 14 bar g 356
 Pipe Schedule 8 (SPRAY) BS.2871 Copper/Nickel (90/10) 20 bar g 357
 Pipe type extrema 167
 Pipe types 167
 Pipe types option 25
 Pivot threshold parameter 177
 Power ramp 208
 P-Q curve 233
 Prerequisites 8
 Pressure difference sensor 284
 Pressure Envelope 136
 Pressure envelopes 102
 Pressure sensor 284
 Pressure switch 295
 Print preview 20, 113
 Print setup 113
 Printing the grid 120
 Printing the schematic 20
 Product modules 4
 Properties window 16
 Property window scroll buttons 16
 Pumps - Coefficients Known 24, 152
 Pumps - Coefficients Unknown 24, 150

- Q -

Quadratic pump curve 225
 Quick Opening Valve 241

- R -

Rankine-Hugeniot 222
Receiving tank 266
Receiving vessel 266
Redo 22
Regulator valve 252
Relative tolerances 178
Remote update 8
Remove tags 27
Report 26
Result graphs page 120
Results in the data window 50
Results page 120
Reynold's number 203
Rigid connections 86

- S -

Saturated liquid 163
Saturated vapour 163
Save as 20
Saving a file 20
Schedules 24, 146
Schematic printing 113
Schematic underlay 106
Schematic window 15, 106
Scroll buttons 16
Security key 8
Security key problems 9
Selected items 108
Selecting a row 120
Selecting graphs for components 172
Selecting several components 107
Selection tool 107
Sensor 284
Set point 289
Short pipe 216
Short pipes 89
Sign conventions 86
Signal selector 299
Simple colouring rules 130
Simple force 86
Simple pump 225
Simple rules 26

Simple tank 270
Sine wave 208
Single-node caisson 275
Small sprinkler system 54
Smooth pump curve 225
Snapshots 179
Sorting in the grid 120
Specific component errors 309
Specifications 208
Spray model option 202
Standard and Spray models 202
Standard model option 202
Status checking 126
Status dialog 126
Step Profile 208
Streeter 312
Support 4
Surface roughness 312
Surfaces 86
Surge tank 264
Suter curves 229
Switch 295
System library 144

- T -

Tables page 120
Tabular results page 120
Tabular view 16, 120
Tag selected items 27
tag toolbar 31
Tagging 133
Tee junction 223
Tees 223
Temporary path 177
Text tool 111
The forces file 179
The graph file 179
The report file 179
Thorley 312
Time control 89
time steps 89
Timestep 179
Title 158
Title option 25
Tool tips 168

Tools menu 27
 Tracking time constant 289
 Transfer function 287
 Transient 172
 Transient options 25, 158
 Transient Output 179
 Transient toolbar 29
 Transitional flow 203, 214
 Turbo pump 229
 Turbo pumps 24, 153
 Turbulent flow 203, 214
 Two-node caisson - type 1 276
 Two-node caisson - type 2 277

- U -

Undefined components 38, 112
 Undo 22
 Undo/Redo buttons 22
 Units 159
 Units option 25
 Unsaturated 163
 Update key 28
 Updating a license 8
 Use PIPENET window style 27
 Use Windows menu style 27
 User-defined fluid 163
 User-defined timesteps 89
 User-defined wavespeed 207

- V -

Vacuum Breaker 271
 Validation 122
 Valve characteristics 241
 Valves 24, 241
 Vapour cavitation 95
 Variable time step 92
 Version numbers 28
 View graphical underlly 23
 View menu 23
 View palette 23
 View schematic overview 23
 View schematic window 23
 view status bar 23
 View toolbar 23

Vista 3
 Volumetric flow 159
 VTS 92

- W -

Water-steam 163
 Wave Speed 207
 Waypoints 112
 Weir Crest 266
 Welcome 3
 Wiley 312
 Windows menu 27

- Z -

Zoom 108
 Zooming with the mouse 116