PIPENET VISION™

Standard Module

User and Reference Manual

Software Revision 1.5

SUNRISE

azarnasim

AIR CONDITIONING COMPANY

azarnasim

AIR CONDITIONING COMPANY

Foreword

Standard User and Reference Manual

Sunrise Systems Limited

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

azarnasim

AIR CONDITIONING COMPANY

Table of Contents

Part 1

azarnasim

AIR CONDITIONING COMPANY

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
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.

Part 2

2 Installation

2.1 Installation

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

2.2 Prerequisites

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

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

2.3 Updating a key license

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

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

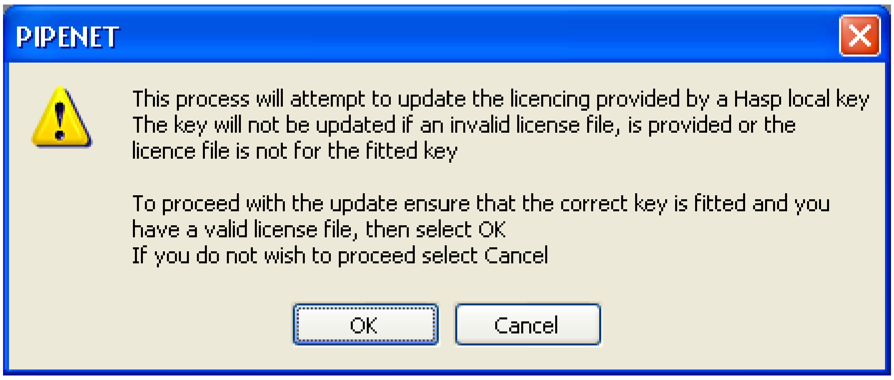
Expiry dates

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

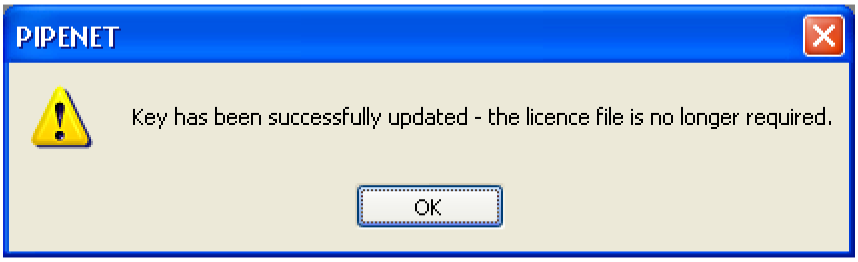
Updating a licence file

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

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



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



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

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

2.4 Security Key Problems

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

Security access rights

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

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

that PIPENET will never replace an existing file in this directory if it is more recent than the one shipped with PIPENET).
- Access to the Windows' System registry

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

Fitting the security key

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

Licensing provided by a key

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

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

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

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

Checking installation of security key drivers

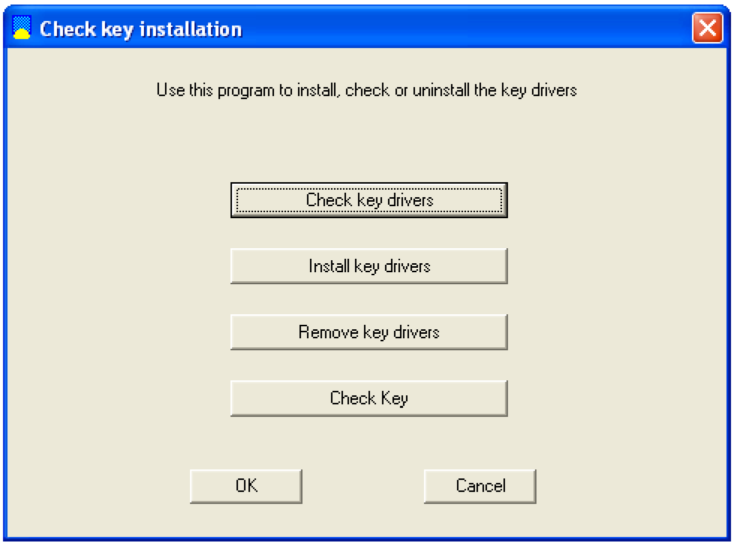
If you have followed all of the instructions above and you have a key licence for the module you are trying to run then you should have no further problems. However, if you are still having problems, please perform the following checks and email the results to Sunrise Systems. Be sure to include the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

Trouble shooting

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

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

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



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

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

The key check utility is stored on your hard drive during installation in the sub-directory keydriver, the key check utility program is named KeySetup.exe.

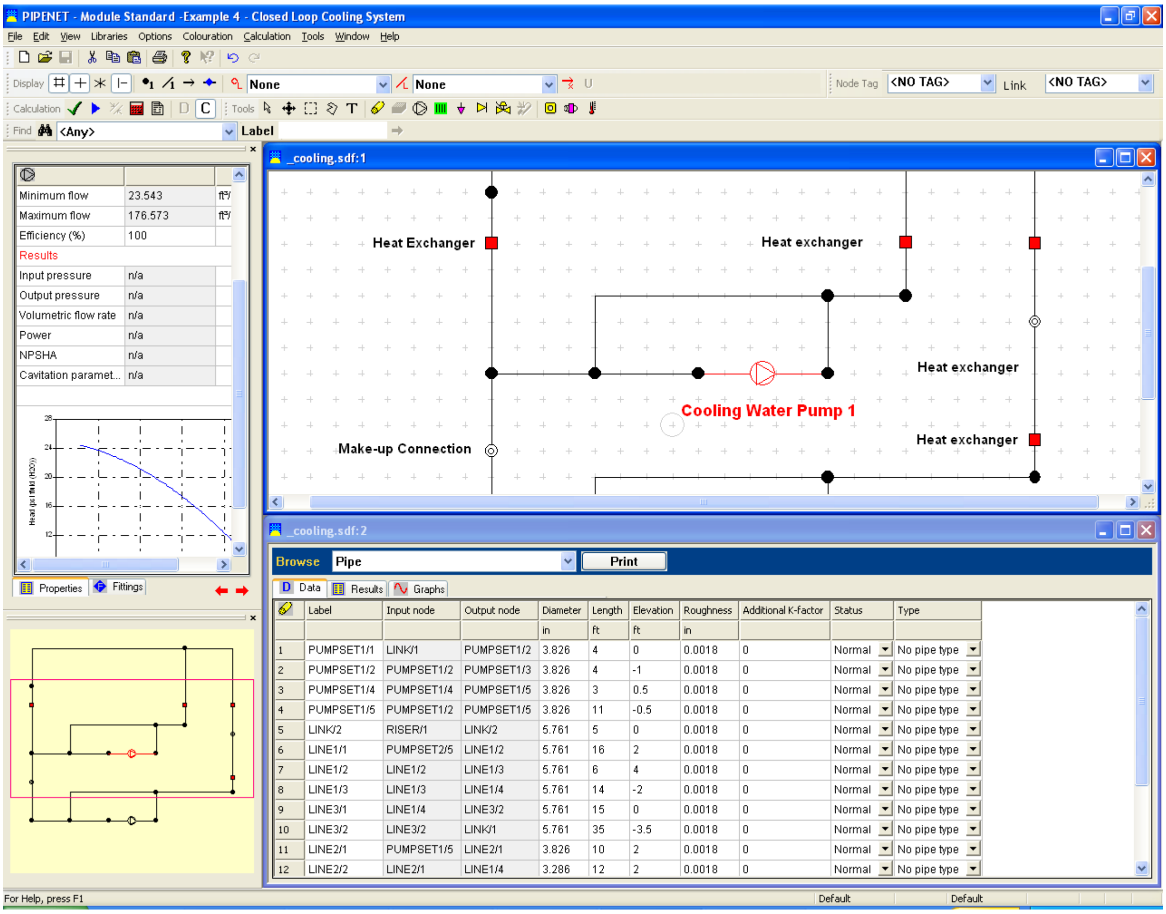
A Tour of the User Interface

Part 3

3 A Tour of the User Interface

3.1 Overall view

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



The four main windows depicted are as follows:

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

undo/redo facility.
- Lower-right: a Tabular View of the database is provided by a browse window, via which the user can display and edit component properties, and display results.

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

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

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

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

3.2 The Schematic Window

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

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

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

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

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

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

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

3.3 Tabular view window

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

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

Tabular View windows are opened via the View menu.

3.4 Properties Window

This is a tabbed set of windows showing:

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

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

Pump operating point

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

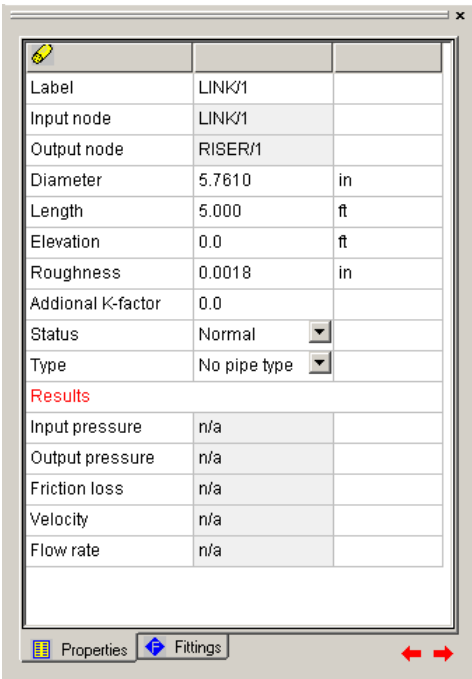
3.5 Component Properties

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

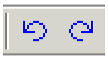
When a component has been drawn on the schematic it will be assigned default parameters. These defaults may be edited in the **properties** tab. Data entry fields in the properties window are either straightforward numeric or text entry, or of the selection style when there is a limited number of options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

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

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



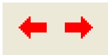
The above example shows the attributes for a pipe. All attributes can be edited: The length and elevation of the pipe are shown in feet and the diameter in inches. The status of the pipe is selectable from a drop-down list, and will be one of Normal, Blocked or Broken. To edit a field click in the appropriate row in the second column and enter the new value, or select from the available options. To accept the value enter Tab to move down to the next field or Enter on the keyboard. Values with a light-grey background cannot be edited. Changes made in the window can be undone and redone using the undo and/redo keys:



The left key is Undo and the right key Redo.

Scroll buttons

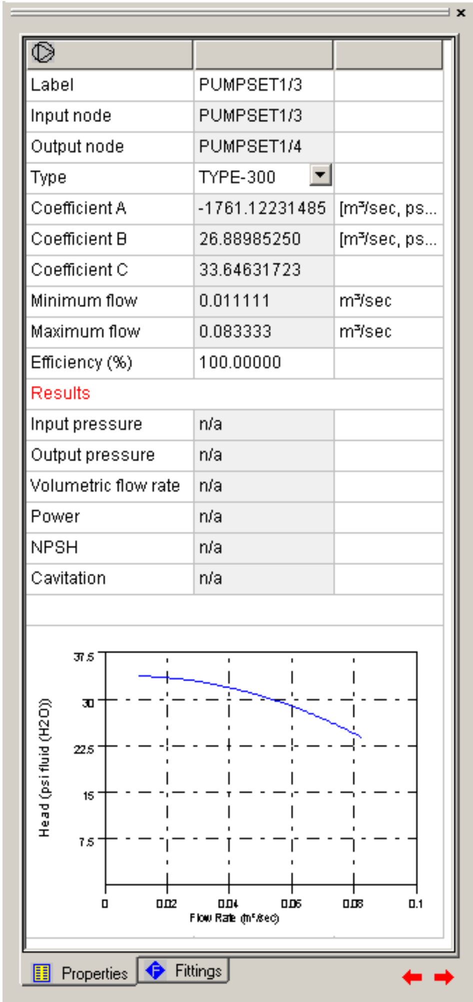
In the bottom right-hand corner of the window are two scroll buttons, these are used to move from the currently displayed component to the next or previous component of the same type.



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

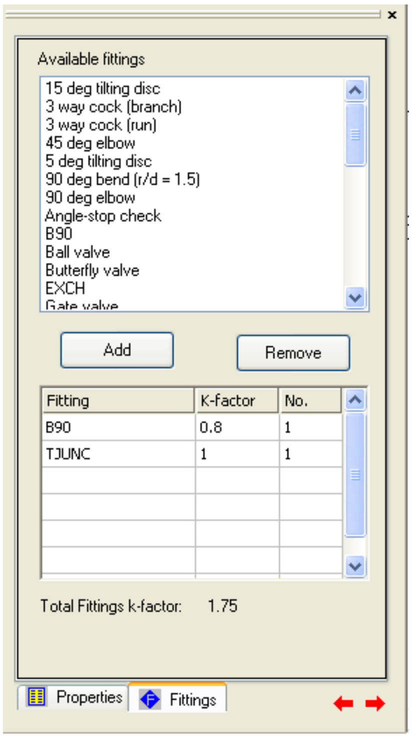
Display of accompanying graph

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



3.6 Fittings

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



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.

Problems displaying the k-factor for a fitting

For all of the built-in fittings, and in most cases for user-defined fittings, the calculated K-factors can be displayed in the user interface. However, if the fluid density is unknown prior to the calculation, as will happen if the fluid is one of the following:-

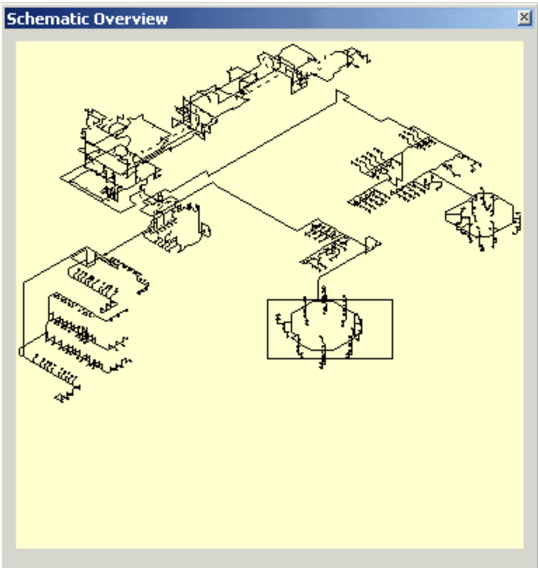
- Liquid - variable properties
- Steam
- Low pressure gas
- Medium pressure gas

then the K-factor for a user-defined **Device** type fitting cannot be displayed since it depends on the fluid density. In these cases the K-factor for Device fittings and the total

K-factor will be displayed as **n/a** (not available). The correct K-factor for a Device type fitting will of course be calculated correctly by the calculator and displayed correctly in the browser output.

3.7 The Schematic Overview Window

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 achieved via the **View | Schematic Overview** menu option.

3.8 Menus

3.8.1 File Menu

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

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

Open
Opens an existing data file - files may be old format data files with a .DAT file

extension or new style files with a .SDF file extension.

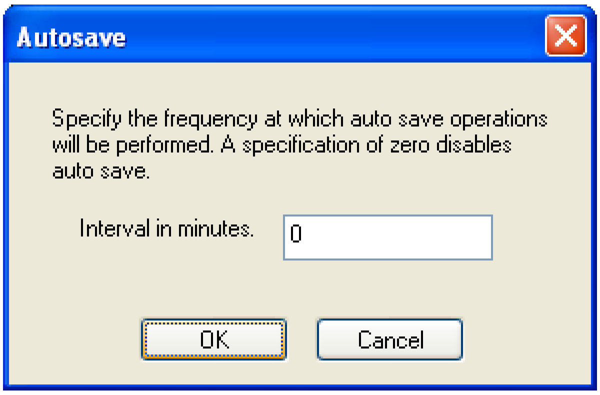
Close
Closes the network - if changes have been made to the network you will be prompted to save the file first before closing.

Save
Saves the current network and continue working

Save As
Saves the current file under a different file name.

Export...
This will export the current network as an old style .DAT file together with any associated old-style library files, that is .PDF files, .UFL files, etc. When selecting this option you should be aware that some graphical information will not be saved, for example colour schemes. When prompted for a file name enter the name of the .DAT file, any associated library files will be saved with the same name, but with a different file extension.

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

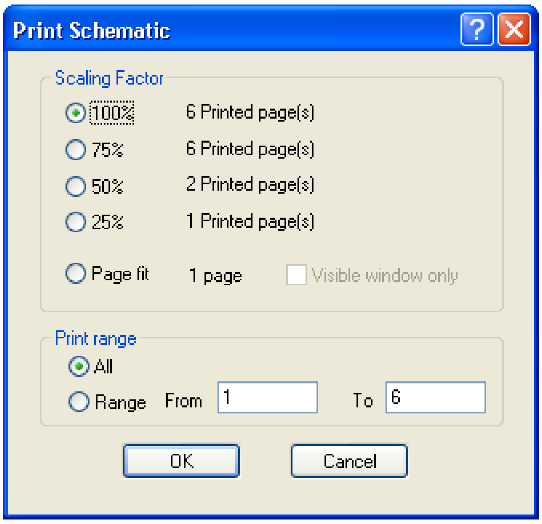


Open Library
Opens an external system library.

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

Print
Print the schematic or the grid - which is printed will depend on which of the

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



Print Preview
Previews appearance of printed output.

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

Exit
Exits PIPENET.

Recently used file list
Provides a list of recently opened files.

3.8.2 Edit Menu

This menu provides a number of edit related functions:

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

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

Cut
Combination of a copy operation followed by a delete.

Copy
If the select tool is in operation this will copy the attributes of the selected component. If the Area tool is in use it will copy all components within the selection rectangle.

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

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

Mirror
Used with the Area Tool to mirror the contents of the selected area left-right.

Invert
Used with the Area Tool to invert the contents of the selected area.

Undo/redo on the toolbar

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

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

3.8.3 View Menu

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

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

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

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

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

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

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

Schematic Overview
Displays or hides the Schematic Overview Window.

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

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

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

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

3.8.4 Libraries Menu

Before attempting to use libraries, the user should first read the chapter on libraries.

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

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

Fittings
Define or edit pipe fittings in the local user library.

Control valves

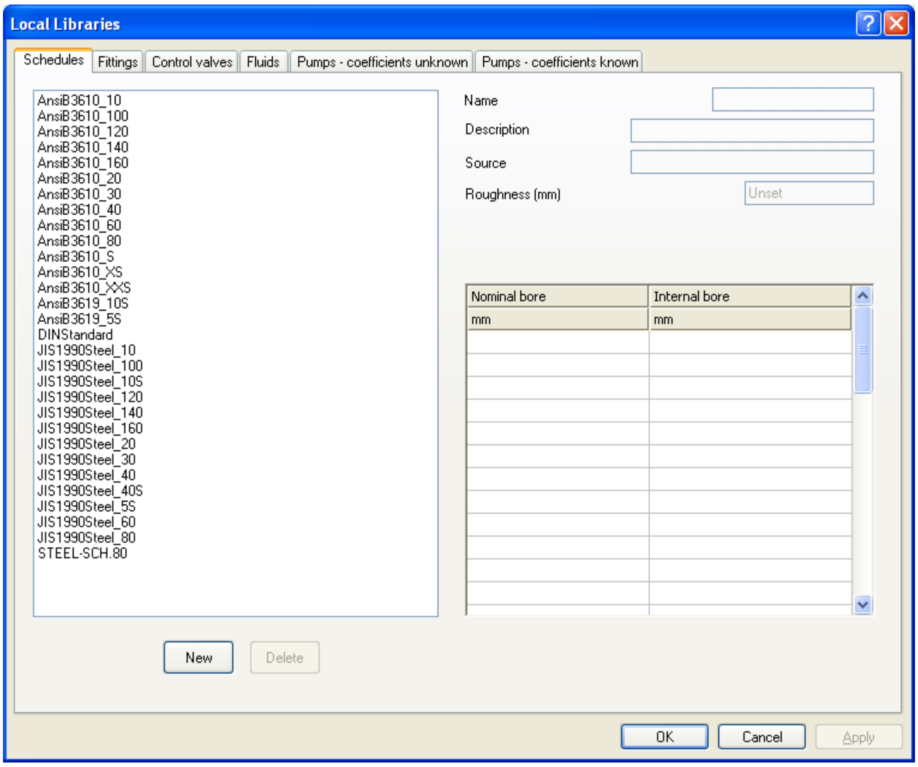
Define or edit control valves in the local user library.

Fluids
Define or edit library fluids 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.

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-emphasis the distinction between Local User Libraries, where changes made only affect the current user, and System Libraries, where changes may affect other users.

3.8.5 Options Menu

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

- Title - Title for the network. Up to four lines of text may be specified.
- Module Options - Various modelling options.

- Units - Units to be used.
- Fluid - Fluid properties.
- Pipe Types - Pipe types.
- Display options - Schematic display options.
- Calculation- Calculation options, including tolerances.
- Output tables - Selection of which output results are to appear in the browser output.
- Defaults - Default information for pipes and nozzles (module dependent).

3.8.6 Colouration Menu

This menu contains three items which can be used to setup the colouring of nodes and/or pipes 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

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

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

Spec. for Calculation
Displays the Options dialog with the Calculation options tab displayed.

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

Pipe Sizing
Runs the pipe sizing phase of the calculation only. his phase will calculate the bore of any pipes with an undefined bore. Validation does not consume a

calculation for those users with metered licenses. Pipe sizing is only possible if there is at least one pipe type and there are no ducts in the system.

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

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

3.8.8 Tools Menu

The tools menu provides a number of useful tools:

Tag selected items
Used in colour coding to tag, or mark, selected components as being significant in some way so that they can be colour coded.

Remove tags
Marks all tagged components as untagged.

Make Path
Used in the construction of an elevation profile to construct a path though 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 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 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.

3.8.9 Window Menu

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

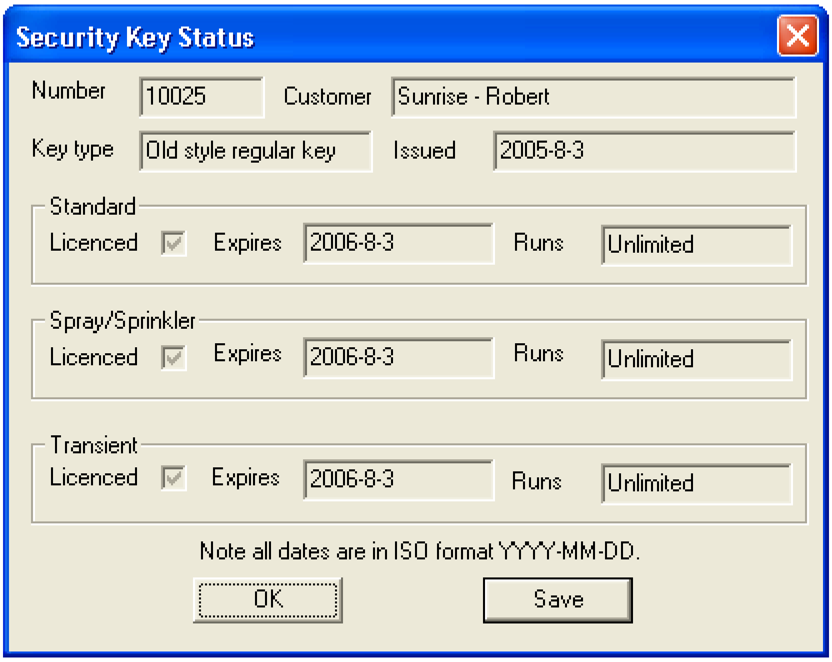
3.8.10 Help Menu

Help topics
Displays 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
Standard Calculator - version 4.nn

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



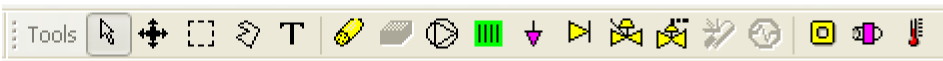
Selecting **OK** closes the window, whilst selecting **Save** will save the displayed details to a text file (the save feature is not implemented).

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

Pipe
Duct (only enabled if fluid is a gas)
Pump
Filter
Nozzle (only enabled if fluid is a liquid)
Non-return valve
Control valve
Elastomeric valve
Leak (only enabled if fluid is a gas)
Heat exchanger (only enabled if heat transfer mode has been selected)

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

Orifice plate - only one allowed per pipe, but not allowed if pipe already has

a fixed pressure drop fitted
Fixed pressure drop - only one allowed per pipe, but not allowed if pipe already has an orifice plate fitted
Property - only one per pipe or duct. Button is disabled if Heat Transfer mode has been selected

3.9.2 Options Toolbar

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

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

Group 1 - grid related options:

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

Group 2 - label options:

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

Group 3 - results related options:

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

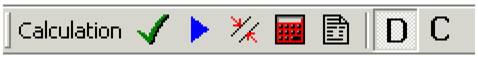
Group 4 - Underlay control:

A single button is provided. Selecting the button will display a dialog where the intensity of the underlay can be specified as a percentage.

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

3.9.3 Calculation Toolbar

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



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

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

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

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

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

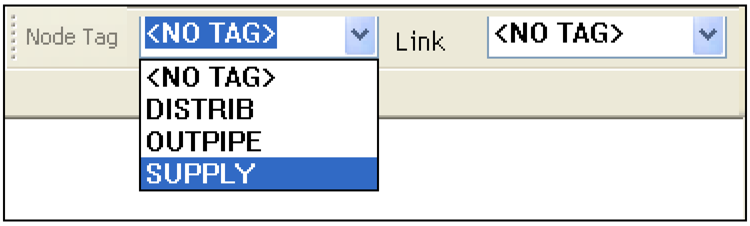
Design phase
Places the front-end in design phase, in this phase pipe sizing operations can be performed repeatedly.

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

3.9.4 Tag Toolbar

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

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



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

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

3.9.5 Find toolbar

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

The toolbar consists of:

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

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

Example 1 - Three Pipe System

Part 4

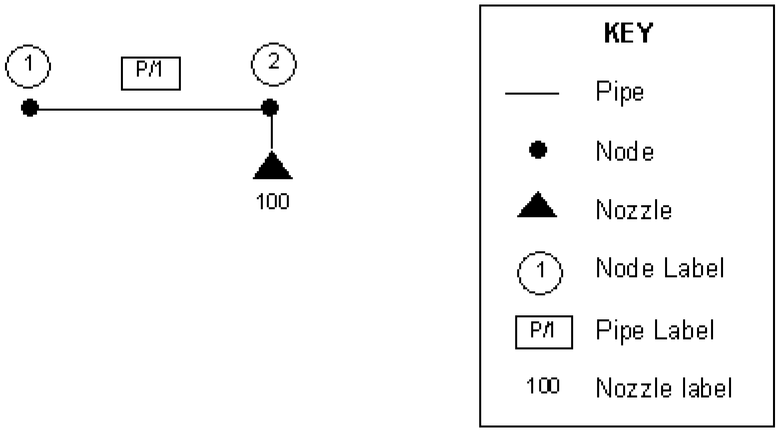
AIR CONDITIONING COMPANY

4 Example 1 - Three Pipe System

4.1 Network Representation

Schematic Diagrams

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



Other Network Data

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

4.2 Labelling the diagram

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

4.3 Labels

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

Untagged:
1
1273
9999

Tagged:
JETTY6/1
JETTY6/876
P/12

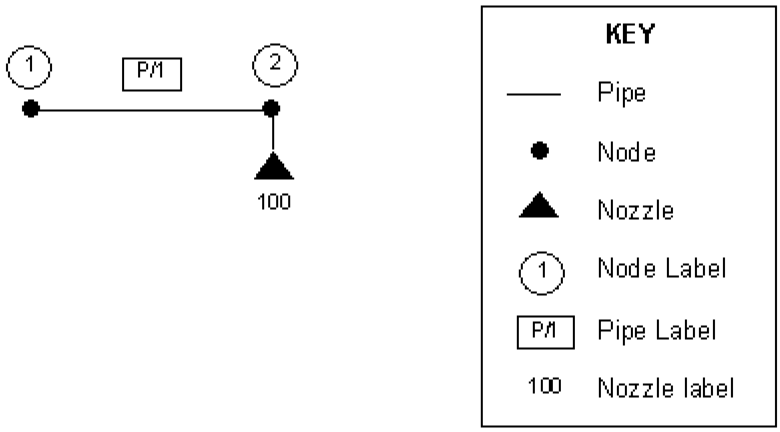
Notes:
1. Untagged labels greater than 999 and tagged labels may not be used in the same network.
2. Tags not followed by a slash and a number are not valid labels. For example, XYZ is a valid tag but not a valid label when used alone.
3. Tags are case insensitive, that is, RING is taken to be the same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

4.4 Inlets and Outlets

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

4.5 Network Topology

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



as having:

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

4.6 Calculation and Design Phases

The Design Phase

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

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

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

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

The following should be noted:

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

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

The Calculation Phase

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

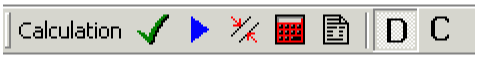
Typically the calculation phase is used to do one or more of the following:

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

User Interface and the Design and Calculation phases

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

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



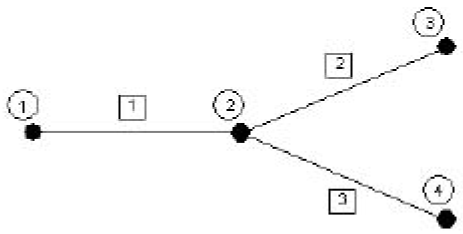
and the sequence of operations will typically proceed as follows:

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

Note that pipe sizing is only possible if there is at least one pipe type defined and there are no ducts in the system.

4.7 Three pipe system

The Network under consideration is shown in the figure below:



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

The details of the three pipes are as follows:

Pipe label	Input node	Output node	Diameter (mm)	Length (metres)	Elevation (metres)	Roughness (mm)	Velocity head loss
1	1	2	32	1	0	0.01	0
2	2	3	20	1	0	0.01	0
3	2	4	20	1	0	0.01	0

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

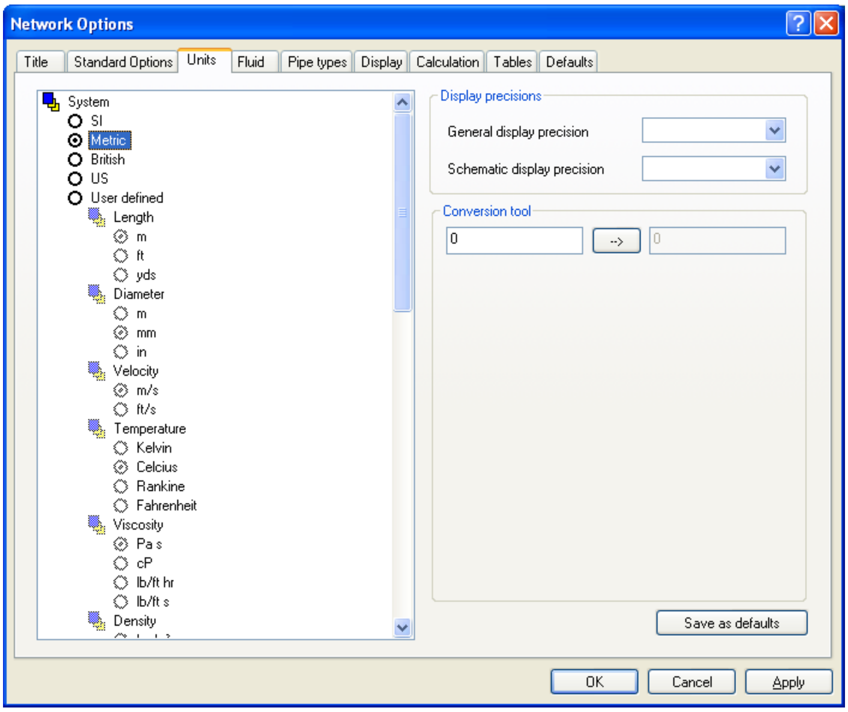
Note also that pipes have a notional direction from input to output, which does not necessarily correspond to the direction of flow. The elevations shown here are changes in elevation of the pipes as measured from the input to the output. Thus a positive value means that the elevation increases as we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

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

4.8 Specifying Units

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

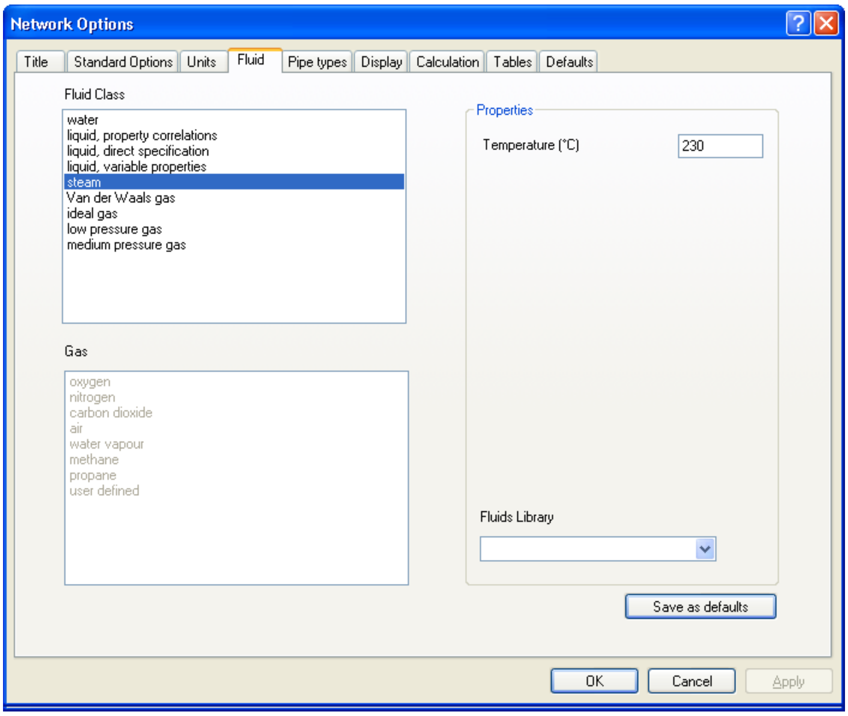
Select the menu option **Options | Units** and then from the dialog that appears select the unit system as M**etric** from the system options in the upper part of the left-hand window. Information on the other features available in this dialog are discussed in Specifying Options - Unit Options :



Select **OK** to close the dialog.

4.9 Initialisation

The initialisation of the network consists simply of specifying the fluid to be used via the Options menu. Select the menu item **Options | Fluid** and the following dialog appears:



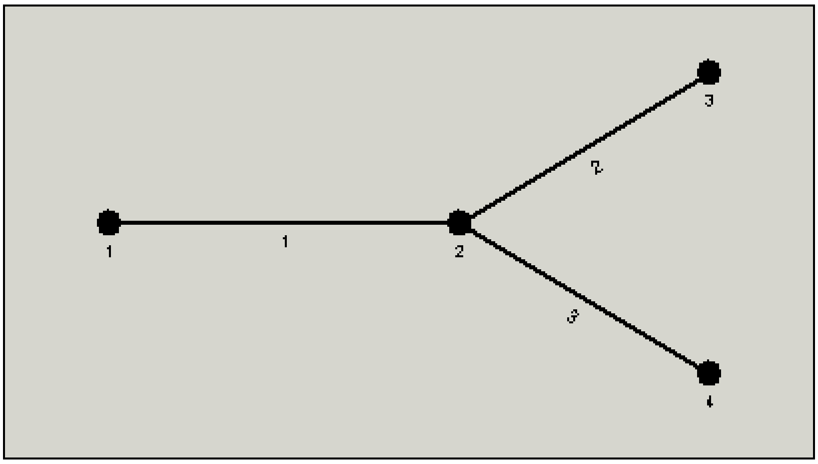
4.10 Entering the network data

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

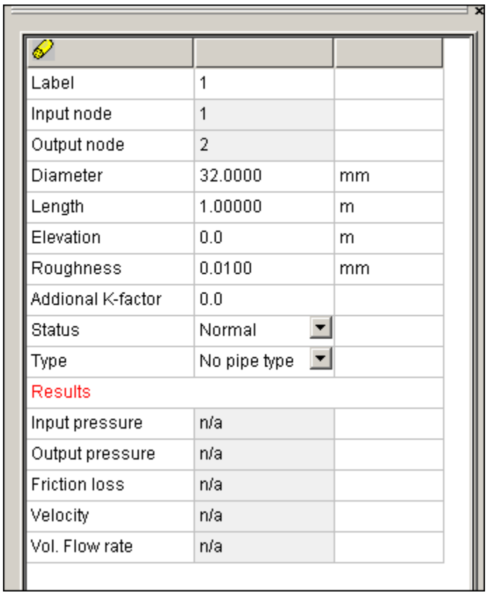
To draw the first point proceed as follows:

1. From the toolbar select the pipe drawing tool 
2. Place the cursor at the point where you want the input node to appear and left click.
3. If the selected point coincides with an existing node 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, between defining the input node and the output node, you want to abort creation of the link then select the Escape key.
8. Now draw the second pipe in a similar manner ensuring that on the first click the cursor is over the output node of the first pipe. Finally draw the third pipe in the same manner as the second pipe. Note that if a pipe is coloured red this means the pipe is selected and its properties will appear in the Properties Window.
9. Select the node and pipe label buttons  from the display toolbar if they

are not already selected and the network should appear as:

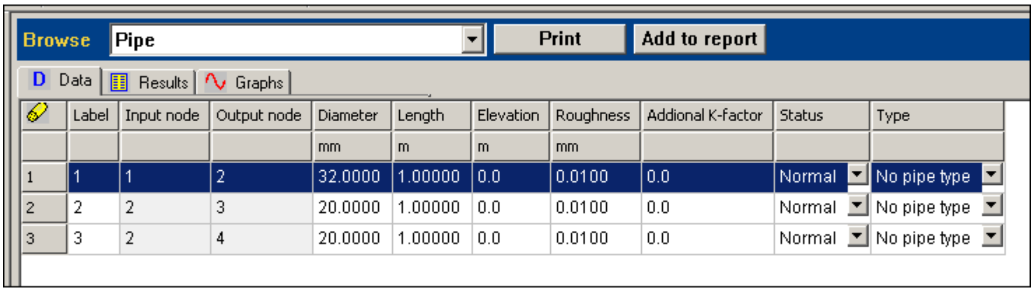


Now we can add the properties to the pipe, for example the bore and length of each pipe. Click on the first pipe and its current properties appear in the Properties Window.



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

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


Browse: Pipe | Print | Add to report
Data | Results | Graphs

	Label	Input node	Output node	Diameter	Length	Elevation	Roughness	Addional K-factor	Status	Type
				mm	m	m	mm			
1	1	1	2	32.0000	1.00000	0.0	0.0100	0.0	Normal	No pipe type
2	2	2	3	20.0000	1.00000	0.0	0.0100	0.0	Normal	No pipe type
3	3	2	4	20.0000	1.00000	0.0	0.0100	0.0	Normal	No pipe type

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

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

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

4.11 Undefined or invalid components

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

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

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

4.12 Specifications

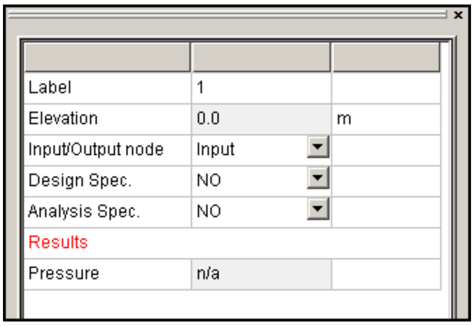
Before sending the network to the calculator, we need to specify conditions of pressure and flow rate at various nodes. Without these, the problem is not mathematically tractable. Also, we must designate certain nodes as input and output nodes. We achieve this simply by adding some attributes to some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

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

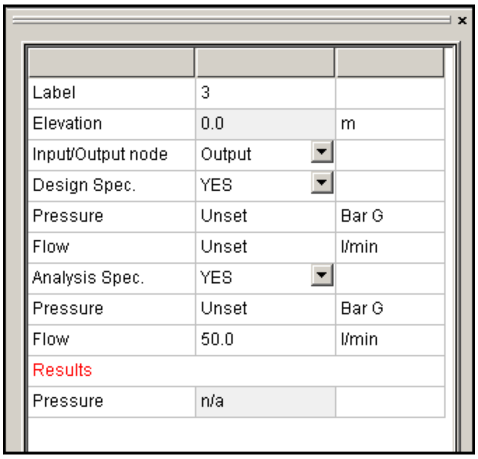
Since all the pipes have had their diameter set, the design phase of the calculation will not have much to do. However, it must still run and we must still provide appropriate specifications for it. The rules for design phase specifications say we must supply the flow rates at all but one of the I/O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

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

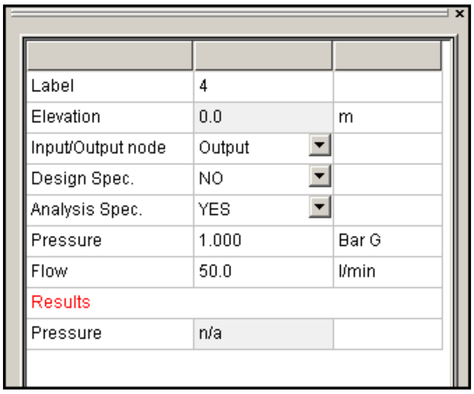
Select the first node and in the properties window, click on the Input/Output node drop-down in the Input/Output node field, change the selection from **No** to **Input** and then hit the Return key. The properties window should appear as:



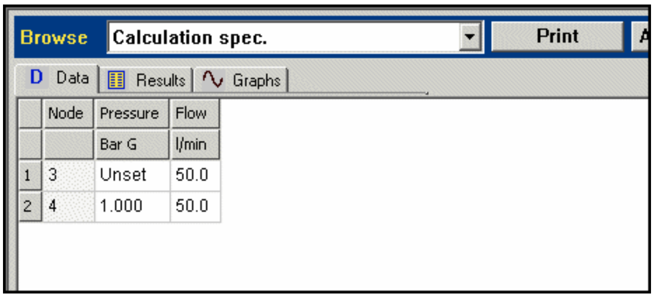
Now select node 3, then in the properties window change the Input/Output node status to **Output** and the Design and Analysis fields both from **NO** to **YES**, the dialog should appear as:



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



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



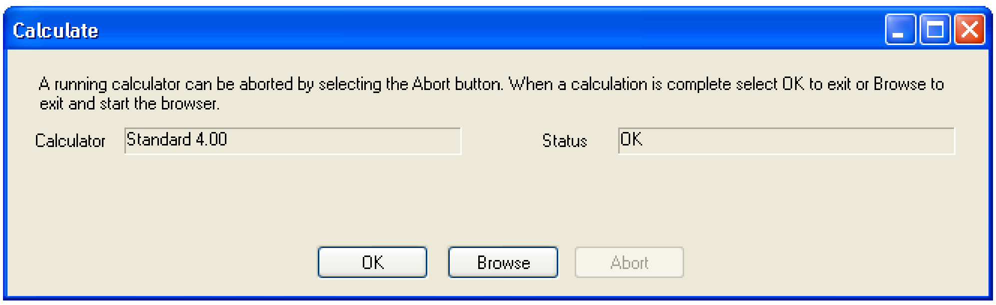
4.13 Calculating

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

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

If you want to check that the data is correct without performing a calculation then (useful for those users with limited run licenses) you can use the **Calc | Check** menu option instead, or click on the Check toolbar button .This option will check that the input data is valid without performing a calculation.

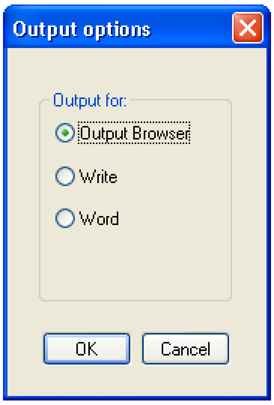
For all but very large networks (200 pipes or more) the calculation will complete almost immediately. Whilst the calculator is active the following dialog appears:



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

4.14 Inspecting the results

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



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

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

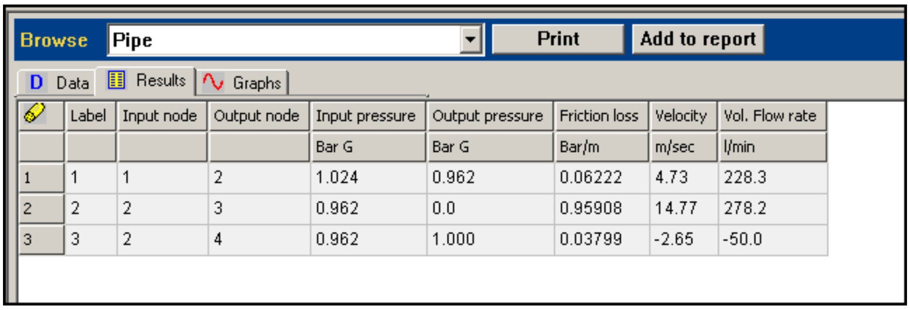
The report file is suitable for printing with most printers with the page orientation set to landscape. The scroll bars can be used to move around the report. At the top and bottom of the vertical scroll bars, page icons can be found. These page icons will allow users to move from one page to the next.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

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

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

Using the Tabular View to view results

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


Browse | Pipe | Print | Add to report

D Data | Results | Graphs

	Label	Input node	Output node	Input pressure	Output pressure	Friction loss	Velocity	Vol. Flow rate
				Bar G	Bar G	Bar/m	m/sec	l/min
1	1	1	2	1.024	0.962	0.06222	4.73	228.3
2	2	2	3	0.962	0.0	0.95908	14.77	278.2
3	3	2	4	0.962	1.000	0.03799	-2.65	-50.0

4.15 Saving and loading the network

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

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

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

Open PIPENET Standard

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

Example 2 - Nitrogen Distribution System

Part 5

5 Example 2 - Nitrogen Distribution System

5.1 Nitrogen distribution system

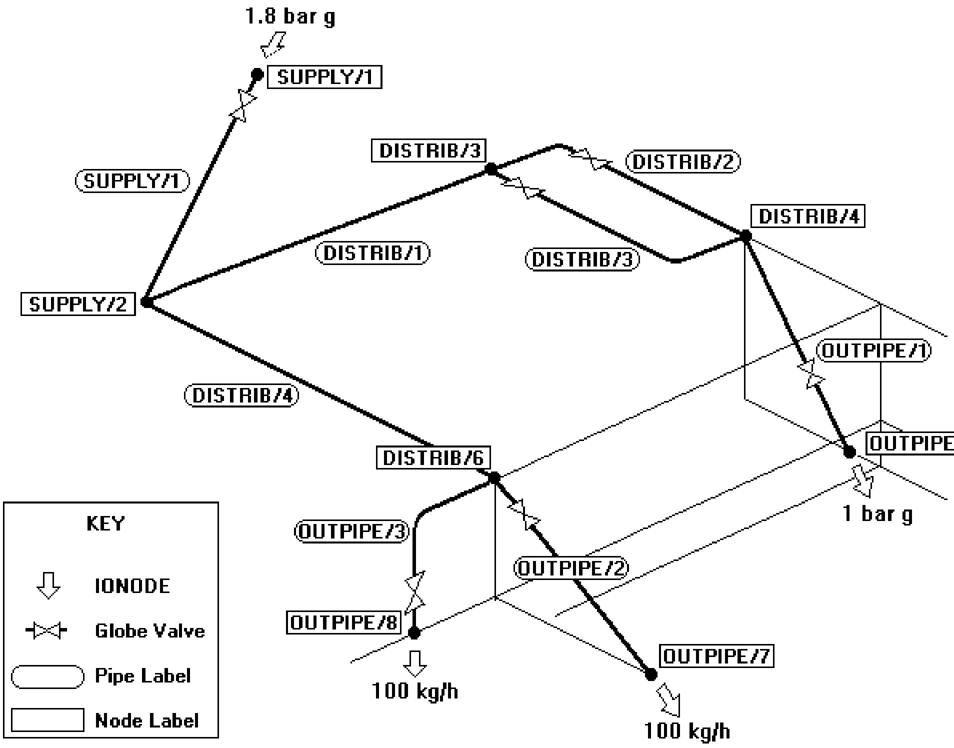
This example will look at a nitrogen distribution system. The network is fairly simple and consists only of pipes. The example illustrates the following points:

- The use of a default pipe roughness.
- The input of pipe data.
- The use of a built-in gas (Nitrogen) as the network fluid.
- The provision of specifications.

The files relating to this example are supplied with PIPENET, and are as follows: _nitroge.sdf and the associated library file _nitroge.slf.

The network

The network consists only of pipes, which are carrying nitrogen. There is a single input at which nitrogen flows into the network, and nitrogen is supplied at three output points. The diagram below shows the full network.



We can split the simulation into 4 stages:

- Initialisation.
- Network data entry.
- Specification data entry.

- Calculation and results.

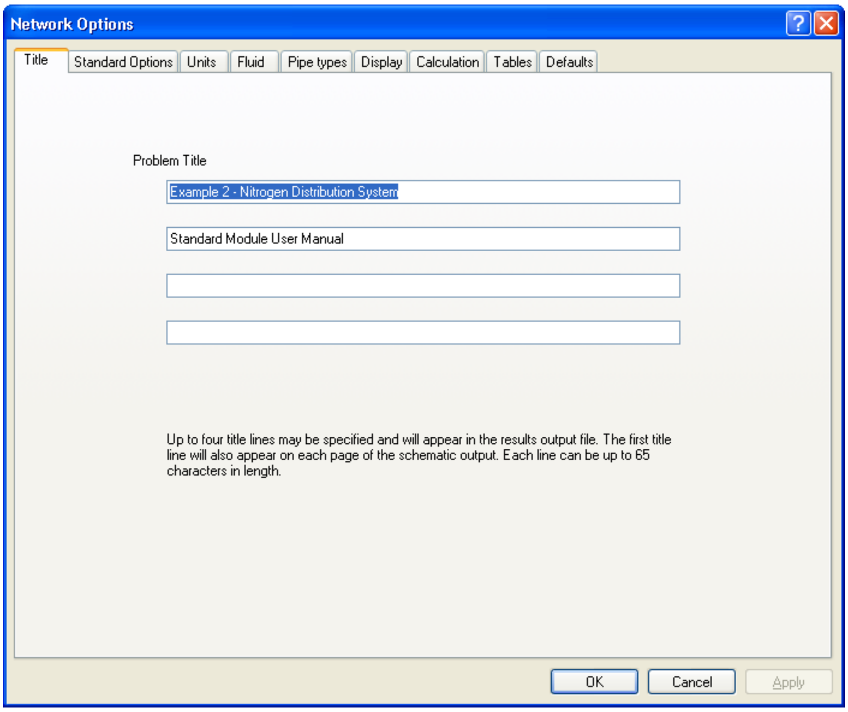
These four stages are outlined in the following sections.

5.2 Initialisation

All initialisation is perform via the options dialogs, select **Options | Title**

Title

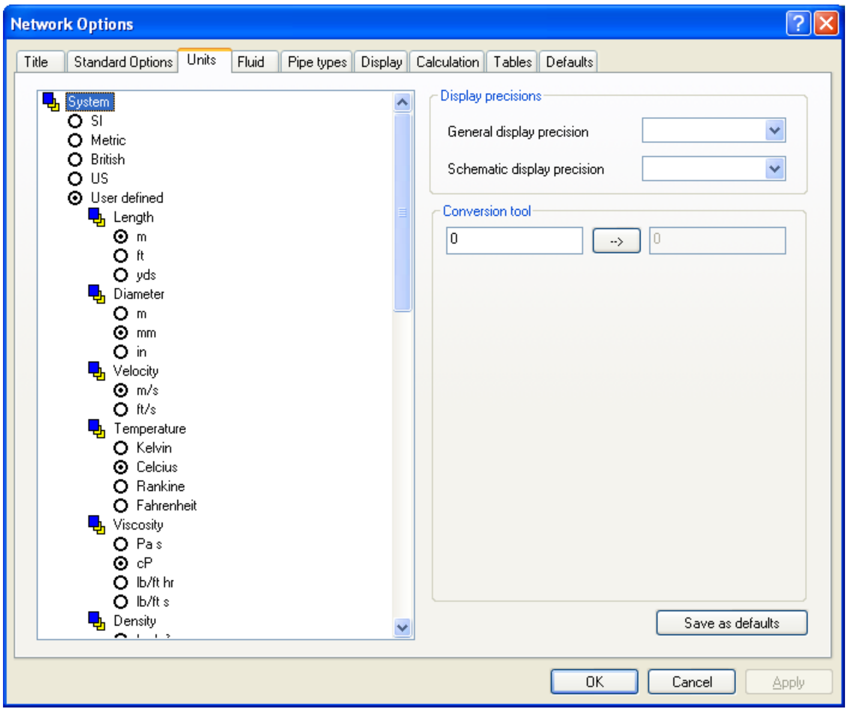
Enter title lines one and two as shown below:



Units

In this example we are going to use user-defined units. Select the menu option **Options | Units** and then from the dialog that appears select the unit system as **User defined** from the system options in the upper part of the left-hand window. Information on the other features available in this dialog are discussed in Specifying Options - Unit Options :

Select the Units tab and then **User defined** from the system options in the upper part of the left-hand window.



The left-hand window can be scrolled down to display more unit options. Make sure you select the following options:

Measure	Unit
Length	metres
Diameter	mm
Velocity	m/s
Temperature	Celcius
Viscosity	cP
Density	kg/m^3
Pressure	Bar gauge
Flow type	Mass flow
Mass flow units	kg/hour

Note in particular the selection of the flow rate type as **Mass** since changing between Mass and Volumetric can lead to problems later on if the fluid density cannot be determined.

Defaults

Since all our pipes have a roughness of 0.0457mm, it would ease data entry if we made this the default value for the roughness. This can be done by choosing the Defaults tab and entering 0.0457mm for the default roughness.

Fluid

The fluid in our network is nitrogen at 25°C and in order to model the gas, we will use the Ideal Gas equations. Select the Fluid tab and:

1. Select the fluid class as Ideal gas from the upper left-hand window.
2. From the lower left-hand window select the gas as Nitrogen.
3. Set the temperature to 25° C.
4. Finally, select **OK** to accept all values and leave the options dialogs.

5.3 Network data entry

The table below gives full details of the pipes used in the network:

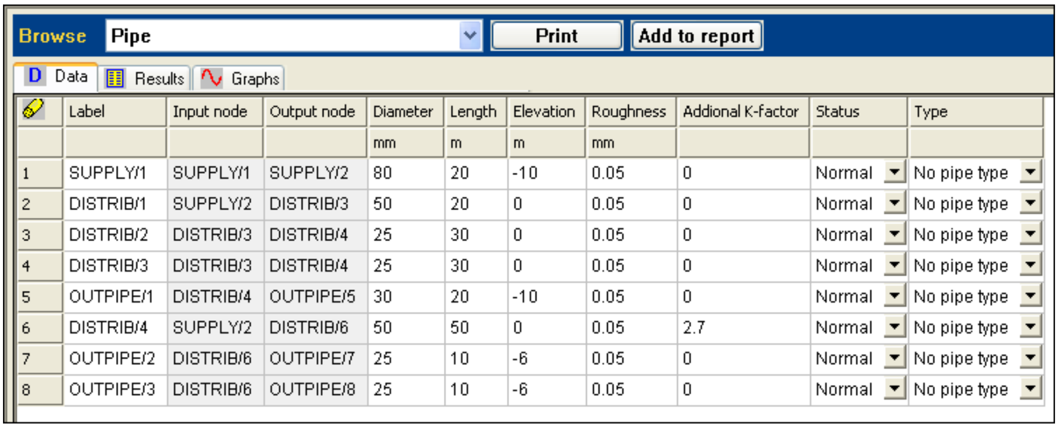
Pipe label	Input node	Output node	Diameter (mm)	Length (metres)	Elevation (metres)	Roughness (mm)	Fittings k-factor
SUPPLY/1 (1)	SUPPLY/1	SUPPLY/2	80	20	-10	0.0457	GLOBE
DISTRIB/1 (2)	SUPPLY/2	DISTRIB/3	50	20	0	0.0457	0.0
DISTRIB/2 (3)	DISTRIB/3	DISTRIB/4	25	30	0	0.0457	GLOBE ELBOW
DISTRIB/3 (4)	DISTRIB/3	DISTRIB/4	25	30	0	0.0457	GLOBE ELBOW
OUTPIPE/1 (5)	DISTRIB/4	OUTPIPE/5	30	20	-10	0.0457	GLOBE
DISTRIB/4 (6)	SUPPLY/2	DISTRIB/6	50	50	0	0.0457	2.7
OUTPIPE/2 (7)	DISTRIB/6	OUTPIPE/7	25	10	-6	0.0457	GLOBE
OUTPIPE/3 (8)	DISTRIB/6	OUTPIPE/8	25	10	-6	0.0457	GLOBE ELBOW

The built-in fittings GLOBE and ELBOW are used to model, respectively, the globe valves and 90º elbows in the network. The k-factor of 2.7 in pipe DISTRIB/4 is due to a diaphragm valve and a blanked-off junction on that pipe (not shown on the diagram).

Tags

The labels used in the table above require some explanation. If nodes and components are entered as in the previous example then they will automatically be assigned a unique numeric id. Note that a node can have the same numeric id as a pipe, a pump, or any other component but it will never be assigned the same numeric id as another node. Assuming we entered the pipes in the order shown above then the pipes would be assigned the numeric identifiers shown in parentheses in the first column. Similarly, nodes would be assigned the numeric id show following the character '/' in the second and third columns.

In the above table the parts of the label SUPPLY, DISTRIB and OUTPIPE are tags and the labels referred to as tagged labels. With a relatively small network like this it is probably easier to enter the network (using purely numeric labels) and then edit the labels in the Tabular view. Simply click on an item in the label column and change its label to the desired value. If you attempt to use a label which is already in use then the change will be rejected. Note if you proceed in this way then it is probably best to rename the nodes first.

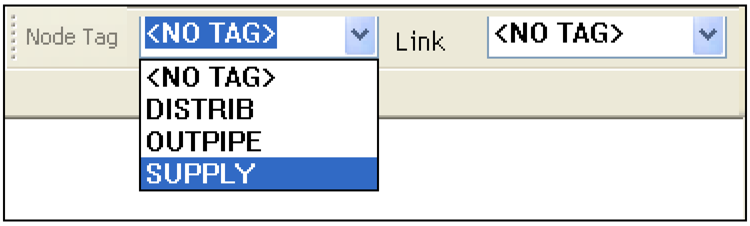


	Label	Input node	Output node	Diameter	Length	Elevation	Roughness	Addional K-factor	Status	Type
				mm	m	m	mm			
1	SUPPLY/1	SUPPLY/1	SUPPLY/2	80	20	-10	0.05	0	Normal	No pipe type
2	DISTRIB/1	SUPPLY/2	DISTRIB/3	50	20	0	0.05	0	Normal	No pipe type
3	DISTRIB/2	DISTRIB/3	DISTRIB/4	25	30	0	0.05	0	Normal	No pipe type
4	DISTRIB/3	DISTRIB/3	DISTRIB/4	25	30	0	0.05	0	Normal	No pipe type
5	OUTPIPE/1	DISTRIB/4	OUTPIPE/5	30	20	-10	0.05	0	Normal	No pipe type
6	DISTRIB/4	SUPPLY/2	DISTRIB/6	50	50	0	0.05	2.7	Normal	No pipe type
7	OUTPIPE/2	DISTRIB/6	OUTPIPE/7	25	10	-6	0.05	0	Normal	No pipe type
8	OUTPIPE/3	DISTRIB/6	OUTPIPE/8	25	10	-6	0.05	0	Normal	No pipe type

For larger networks there is a better way, using the Tag tool bar:

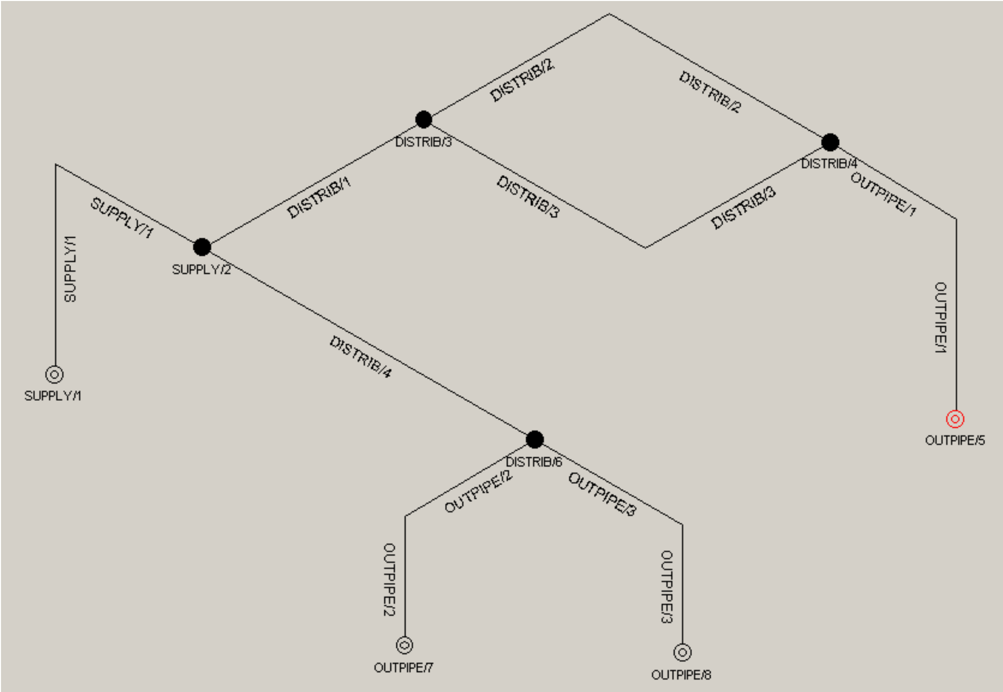
Tags can be assigned separately for nodes and components, by default both have a setting of <NO TAG>. you can either:

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



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

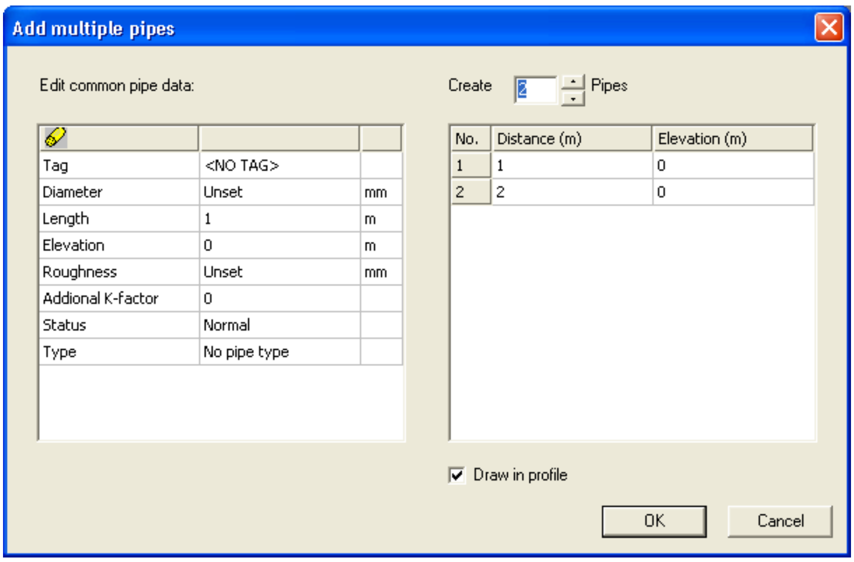
Of course there is no need to change the labels, as long as the entered network is topologically the same as the following the calculated results will be the same. The use of tags simply makes the identification of key points of interest easier.



Note in th layout above use has been made of waypoints and an isometric grid (the grid is not shown here for clarity).

Adding multiple pipes

For some networks (but not this one), large parts of the network may consist of long runs of pipes, typically with the same diameter and the Add multiple pipes tool (**Tools** menu) may help:



Here you can enter information for a single pipe, including a tag and then have PIPENET generate a number of connected copies of the pipe when OK is selected. Before clicking OK you can set the length and elevation for each individual pipe.

5.4 Specifications

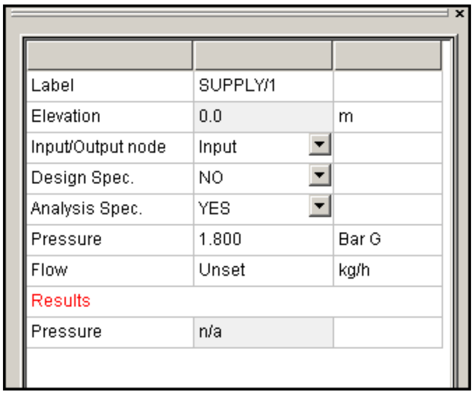
Outlined below is a summary of the specifications used in our network.

I/O nodes:
- The input node for the system is SUPPLY/1.
- The output nodes for the system are OUTPIPE/5, OUTPIPE/7 and OUTPIPE/8.

Pressure Specs:
- Node SUPPLY/1 is at 1.8 bar g.
- Node OUTPIPE/5 is at 1.0 bar g.

Flow rate Specs:
- Flow rate out of node OUTPIPE/7 is 100 kg/hr.
- Flow rate out of node OUTPIPE/8 is 100 kg/hr.

Specifications are entered in the properties window by selecting the appropriate node, making the node an I/O node if necessary and entering the specification data. The Property window for SUPPLY/1 is shown below:



Label	SUPPLY/1	
Elevation	0.0	m
Input/Output node	Input	
Design Spec.	NO	
Analysis Spec.	YES	
Pressure	1.800	Bar G
Flow	Unset	kg/h
Results		
Pressure	n/a	

5.5 Calculation and results

Having entered all the data, we can check the data by choosing the menu option **Calc| Check** (or alternatively the **check** button on the calculation toolbar). If there are no errors, we can run the simulation by choosing the option **Calc | Calculation** or the **calculation** button on the calculation toolbar. All of the results can be examined with the browser or via the Tabular view. In the tabular view select the component type of interest (for this example we only have results for pipes, and nodes), and then the Results tab. The results for the pipes are shown below:

Browse Pipe | Print | Add to report

Data | Results | Graphs

	Label	Input node	Output node	Diameter	Length	Elevation	Roughness	Addional K-factor	Status	Type
				mm	m	m	mm			
1	SUPPLY/1	SUPPLY/1	SUPPLY/2	80	20	-10	0.05	0	Normal	No pipe type
2	DISTRIB/1	SUPPLY/2	DISTRIB/3	50	20	0	0.05	0	Normal	No pipe type
3	DISTRIB/2	DISTRIB/3	DISTRIB/4	25	30	0	0.05	0	Normal	No pipe type
4	DISTRIB/3	DISTRIB/3	DISTRIB/4	25	30	0	0.05	0	Normal	No pipe type
5	OUTPIPE/1	DISTRIB/4	OUTPIPE/5	30	20	-10	0.05	0	Normal	No pipe type
6	DISTRIB/4	SUPPLY/2	DISTRIB/6	50	50	0	0.05	2.7	Normal	No pipe type
7	OUTPIPE/2	DISTRIB/6	OUTPIPE/7	25	10	-6	0.05	0	Normal	No pipe type
8	OUTPIPE/3	DISTRIB/6	OUTPIPE/8	25	10	-6	0.05	0	Normal	No pipe type

Example 3 - Machine Shop Air Extraction System

Part 6

6 Example 3 - Machine Shop Air Extraction System

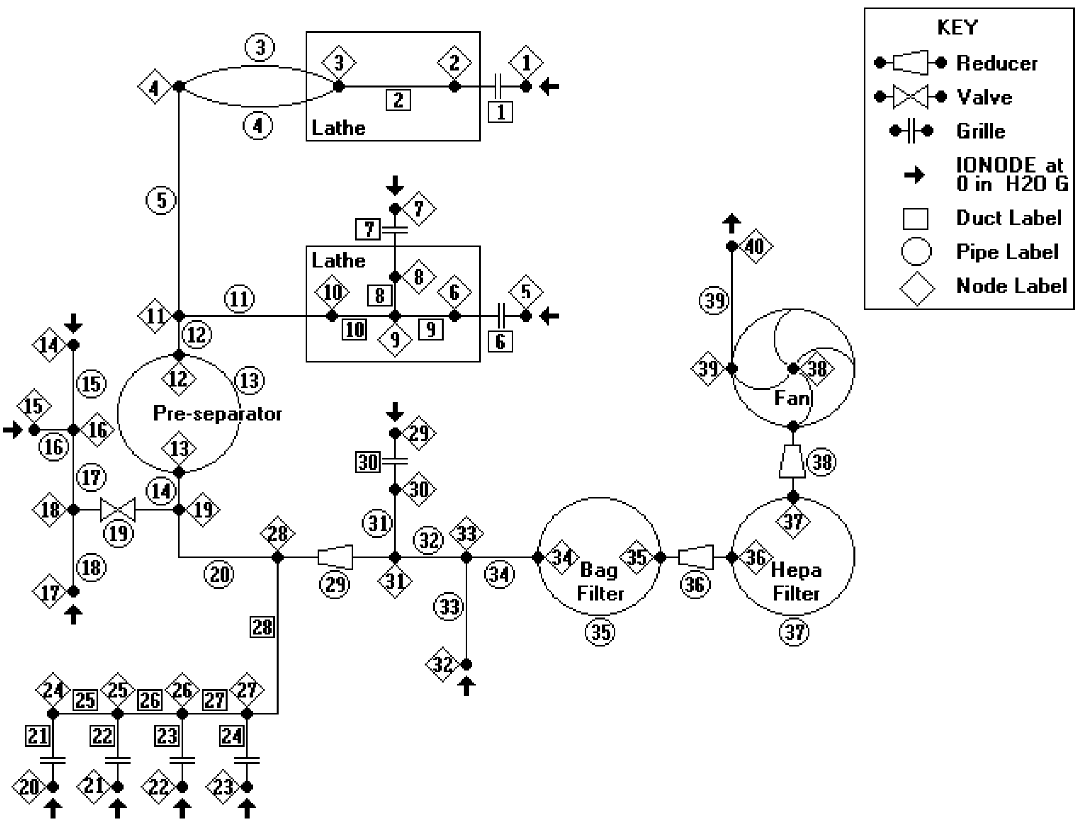
6.1 Machine extraction system

This example will look at an air extraction system for an industrial machine shop and it illustrates the following points:

- How to mix pipes and ducts in a network.
- The use of non-built-in fittings.
- The use of user-defined units.
- The use of a non-library fan.

The files relating to this example are supplied with PIPENET, and are: _newvent.dat, and _newvent.ufl.

The network



The system, as shown above, consists of a number of pipes and ducts of various sizes which remove contaminated air from several locations in a machine shop. Air from two lathes is passed through a pre-separator before joining other air streams. The combined streams are passed through two filters—a "bag filter" and a "hepa filter" before being vented to the atmosphere via the fan. The problem to be investigated is the behaviour of the system when the filters and separator are choked with dust so that they no longer allow a free flow of air.

Running the simulation

As before, we split it into Initialisation, Network data entry, Specifications data entry and Calculation and results.

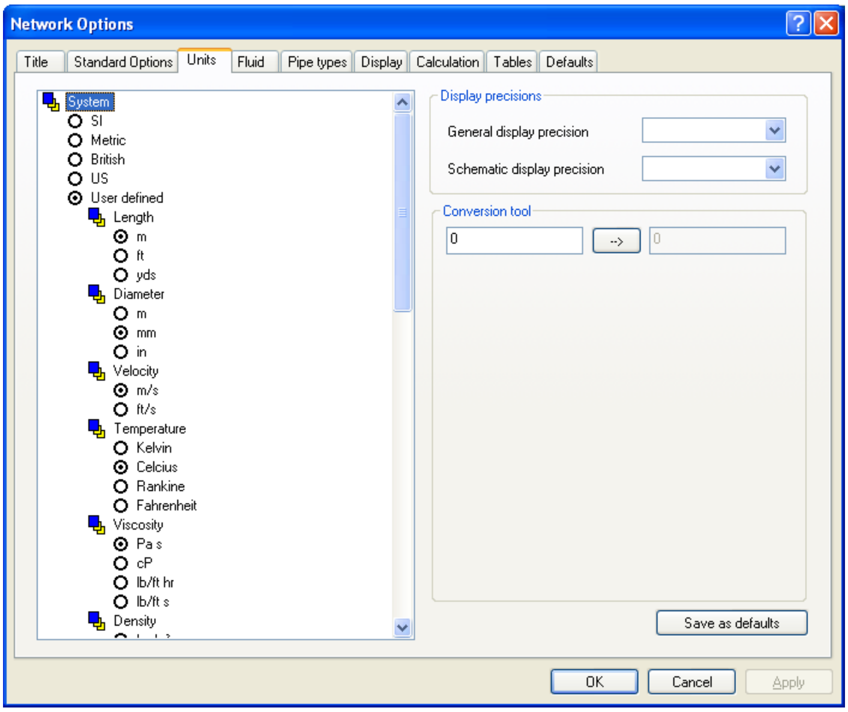
6.2 Initialisation

Title

Choose the menu option **Options | Title** and enter 'Example 3 - Machine Shop Air Extraction System' as the first and only title.

Units

Choose the **Units** tab, set the Unit System to **User-defined.**



Make sure that the unit options are selected as follows:

Measure	Unit
Length	metres
Diameter	mm
Velocity	m/s
Temperature	Celcius
Viscosity	Pa s
Density	kg/m^3
Pressure	inches of water gauge
Flow type	Volumetric flow

Volumetric flow units	m^3/s

Fluid

The fluid in the system is Air which is modelled as an ideal gas at a constant temperature of 15°C. Choose the Fluid tab and:

1. select from the upper left-hand window the fluid class as Ideal gas
2. from the lower left-hand window select the gas as Air
3. set the temperature to 15° C
4. finally select **OK** to accept all values and leave the options dialogs.

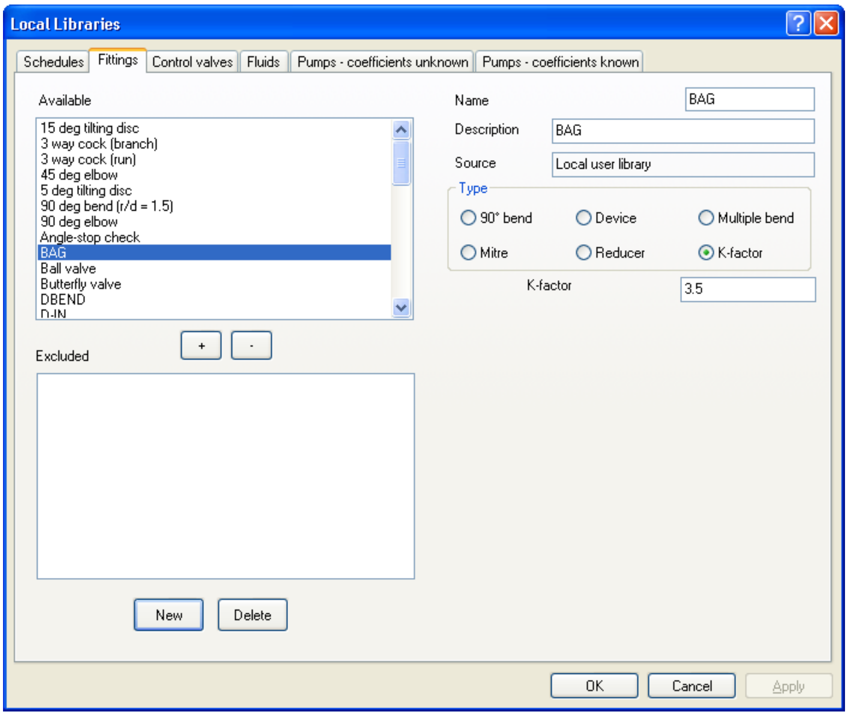
Fittings

All the fittings used in this example are characterised by velocity head loss (k-factor) and are defined as follows:

PBEND	0.2
D-IN	3.2
D-TEE	0.9
DBEND	0.27
FANIO	2.0
GRILL	5.0
P-IN	0.95
P-TEE	0.48
SEP	20 *
BAG	3.5 *
HEPA	3.0 *

In order to simulate the case where the filters and separator are choked with dust, we simply increase the values of velocity head loss associated with the last three fittings of the above list which are denoted by *. Thus we can very easily run several simulations to investigate the effect of different degrees of choking on the system. It is suggested that the user experiments by running several simulations with different values of velocity head loss (k-factor) associated with the last three fittings.

In order to enter the user-defined fittings data, we choose the menu option **Libraries | Fittings** to obtain a dialog box similar to the following:



In the upper left-hand window there is a list of available fittings. In the lower left-hand a set of excluded fittings. These are fittings we have elected to be made unavailable for this network. To make a fitting unavailable, select the fitting from the upper window followed by the - button. To make a fitting available again, select the fitting in the lower window and then select the + button.

Select the type of the fitting from the upper right-hand side of the dialog, provide a name and one or more attribute values. The attributes listed will depend on the fitting type selected. The example above is for the BAG fitting which is a K-factor device, as are all the fittings in this example.

When you have finished, select the **Apply** button to store the changes. Repeat the process for all fittings.

Default Values

Using the menu option **Options | Defaults** set the default pipe/duct roughness to 0.005 mm, the default elevation to 0 m and the default k-factor to 0.

6.3 Network data entry

Both pipes and ducts are required for this system. Pipes are circular, and have just a diameter. Ducts are rectangular and have both height and width. The table below provides details about the pipes and ducts used in the system. Note that elevations and roughness are only specified when they differ from the defaults values.

Enter pipe and duct data using the pipe and duct tools - note if the duct tool is not selectable (greyed out) this means you have not specified the fluid correctly since ducts can only be used with gases.

Note that for historical reasons pipes and ducts share a common number scheme. That is, pipes and ducts are assigned labels such that no pipe or duct has the same label.

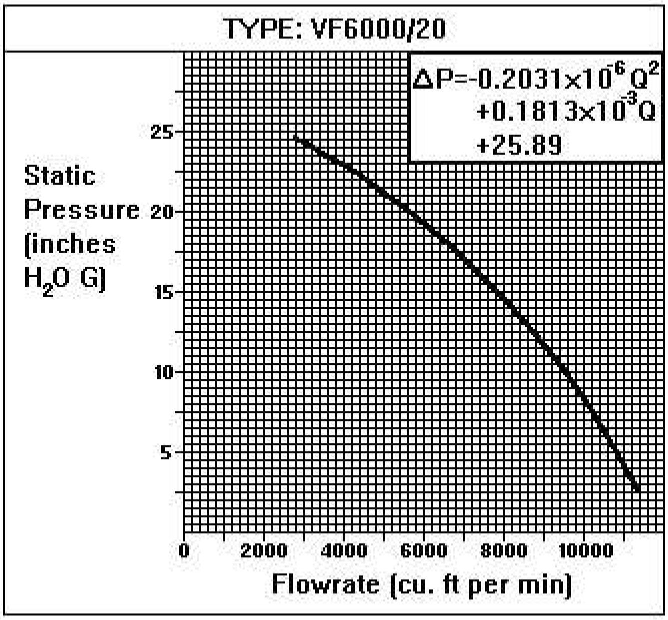
Pipe Duct	In	Out	Diameter or Width Height	Length (metres)	Elevation (metres)	Roughness (mm)	Fittings
1	1	2	1300 400	0.05		0.005	2.5
2	2	3	1800 1100	0.85		0.005	
3	3	4	250	1.0		0.005	0.5 + PBEND
4	3	4	250	1.0		0.005	0.5 + PBEND
5	4	11	330	7.0		0.005	P-TEE
6	5	6	390 390	0.05		0.005	D-IN
7	7	8	860 860	0.05		0.005	D-IN
8	8	9	1000 1000	1.0		0.005	
9	6	9	1300 1000	0.5		0.005	
10	9	10	1300 1000	0.5		0.005	
11	10	11	330	1.0		0.005	0.74
12	11	12	330	1.0		0.005	P-TEE
13	12	13	330	1.0		0.005	SEP
14	13	19	330	1.0	0.2	0.005	2 x PBEND
15	14	16	53.5	12.0		0.018	P-IN
16	15	16	53.5	6.0		0.018	P-IN
17	16	18	53.5	2.5		0.018	2.5
18	17	18	53.5	10		0.018	P-IN
19	18	19	53.5	2.75	1.75	0.018	6.0 + P-TEE
20	19	28	330	9.0		0.005	0.3 + PBEND

Pipe Duct	In	Out	Diameter or Width Height	Length (metres)	Elevation (metres)	Roughness (mm)	Fittings
21	20	24	300 300	0.05		0.005	GRIILL
22	21	25	300 300	0.05		0.005	GRIILL
23	22	26	300 300	0.05		0.005	GRIILL
24	23	27	300 300	0.05		0.005	GRIILL
25	24	25	380 150	1.0		0.005	D-TEE + 2 x DBEND
26	25	26	380 150	1.3		0.005	D-TEE
27	26	27	380 150	1.3		0.005	D-TEE
28	27	28	380 150	3.4		0.005	1.64 + D-TEE + DBEND
29	28	31	360	2.0		0.005	0.4
30	29	30	600 600	0.05		0.005	GRILL
31	30	31	150	8.75	1.0	0.005	0.24 + D-TEE +

							DBEND
32	31	33	380	0.5		0.005	0.3
33	32	33	53.5	2.15	1.0	0.005	P-IN
34	33	34	380	0.5		0.005	0.3
35	34	35	380	1.0		0.005	BAG
36	35	36	600	8.0	8.0	0.005	0.1 + PBEND
37	36	37	380	1.0		0.005	HEPA
38	37	38	600	1.2		0.005	1.4 + FANIO
39	39	40	500	9.2	9.2	0.005	1.25 + FANIO

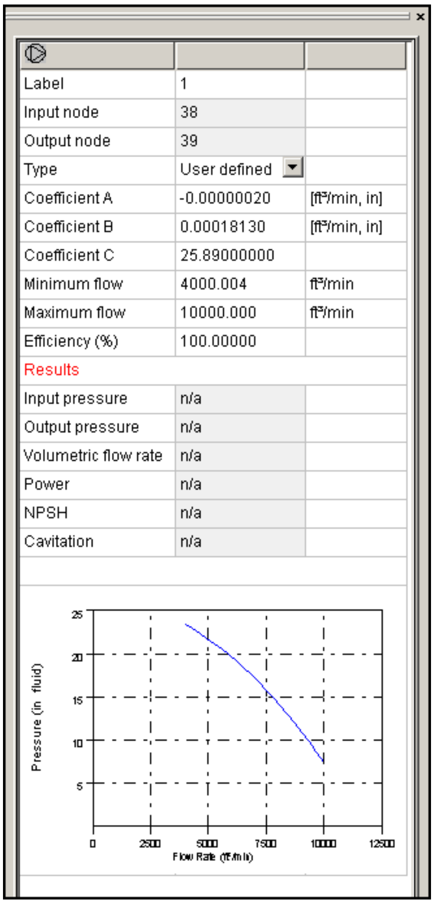
6.4 Fan data

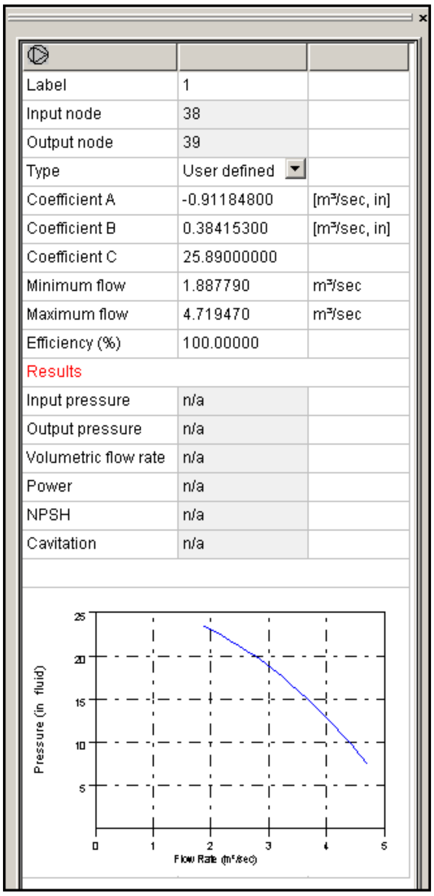
The characteristics of the fan used in this system are shown in the diagram below. It would be possible to read points off this curve and process these with the **Libraries | Pump - Coefficients Unknown**, but this is unnecessary as the coefficients are given. Note that the flow rates are given in ft3/min, and not our chosen units of m^3/s. Hence, before we enter the data for the Fan, choose the menu option **Options | Units** and set the flow rate units to ft^3/min.



To enter a fan select the Pump/fan tool from the tool bar and draw the fan between nodes 38 and 39. Note the arrow shown in the pump/fan symbol shows the direction, which should be from node 38 to node 39. If you draw the pump the wrong way around simply click on the pump and from the popup menu select the option **Reverse**.

In the properties window enter the coefficients, minimum and maximum flows and efficiency and the Properties window should appear as shown in the left-hand image.





Note the pump curve is displayed in the lower part of the Properties window.

Remember to go back to the **Options | Units** and set the units of flow rate back to m^3/s. The Properties window will now appear as shown in the right-hand image above.

6.5 Specifications

The I/O nodes for this problem are nodes 1,5,7,14,15,17,20,21,22,23,29,32 and 40. All the I/O nodes are at atmospheric pressure (i.e. 0 inches of water gauge). This is all we need for the specification data. Select each node in turn and add the specification. The specifications can be visually checked by opening the Tabular view:

Browse | Calculation spec. | Print

D Data | Results | Graphs

	Node	Pressure	Flow
		in w.g.	m³/sec
1	1	0	Unset
2	5	0	Unset
3	7	0	Unset
4	14	0	Unset
5	15	0	Unset
6	17	0	Unset
7	20	0	Unset
8	21	0	Unset
9	22	0	Unset
10	23	0	Unset
11	29	0	Unset
12	32	0	Unset
13	40	0	Unset

6.6 Calculation and results

Having entered all the data, we can check the data by choosing the menu option **Calc| Check** (or alternatively the **check** button on the calculation toolbar). If there are no errors, we can run the simulation by choosing the option **Calc | Calculation** or the **calculation** button on the calculation toolbar. All of the results can be examined with the browser or via the Tabular view. In the tabular view select the component type of interest (for this example we only have results for pipes, and nodes), and then the Results tab. The results for the pipes are shown below:

Browse | Pipe | Print | Add to report

D Data | Results | Graphs

	Label	Input node	Output node	Input pressure	Output pressure	Friction loss	Velocity	Flow rate
				in w.g.	in w.g.	in w.g./m	m/sec	m³/sec
1	3	3	4	-0.003	-0.02	0.002	3.41	0.2
2	4	3	4	-0.003	-0.02	0.002	3.41	0.2
3	5	4	11	-0.02	-0.06	0.002	3.91	0.3
4	11	10	11	-0.002	-0.06	0.004	5.31	0.5
5	12	11	12	-0.06	-0.2	0.01	9.23	0.8
6	13	12	13	-0.2	-4.4	0.01	9.28	0.8
7	14	13	19	-4.4	-4.5	0.01	9.33	0.8
8	15	14	16	0	-0.2	0.01	3.07	0.007
9	16	15	16	0	-0.2	0.02	4.2	0.009
10	17	16	18	-0.2	-0.6	0.06	7.28	0.02
11	18	17	18	0	-0.6	0.05	6.92	0.02
12	19	18	19	-0.6	-4.5	0.2	14.27	0.03
13	20	19	28	-4.5	-4.7	0.01	9.71	0.8
14	29	28	31	-4.7	-5.2	0.04	21.46	2.184467
15	31	30	31	-0.03	-5.2	0.2	30.72	0.5
16	32	31	33	-5.2	-5.7	0.05	24.11	2.733831
17	33	32	33	0	-5.7	1.13483	37.05	0.08
18	34	33	34	-5.7	-6.2	0.05	24.87	2.820903
19	35	34	35	-6.2	-11.5	0.05	25.05	2.841492
20	36	35	36	-11.5	-12	0.005	10.12	2.862447
21	37	36	37	-12	-16.6	0.05	25.4	2.881173
22	38	37	38	-16.6	-17.5	0.005	10.26	2.901482
23	39	39	40	2.2	0	0.01	14.12	2.772588

Example 4 - Closed Loop Cooling System

Part 7

7 Example 4 - Closed Loop Cooling System

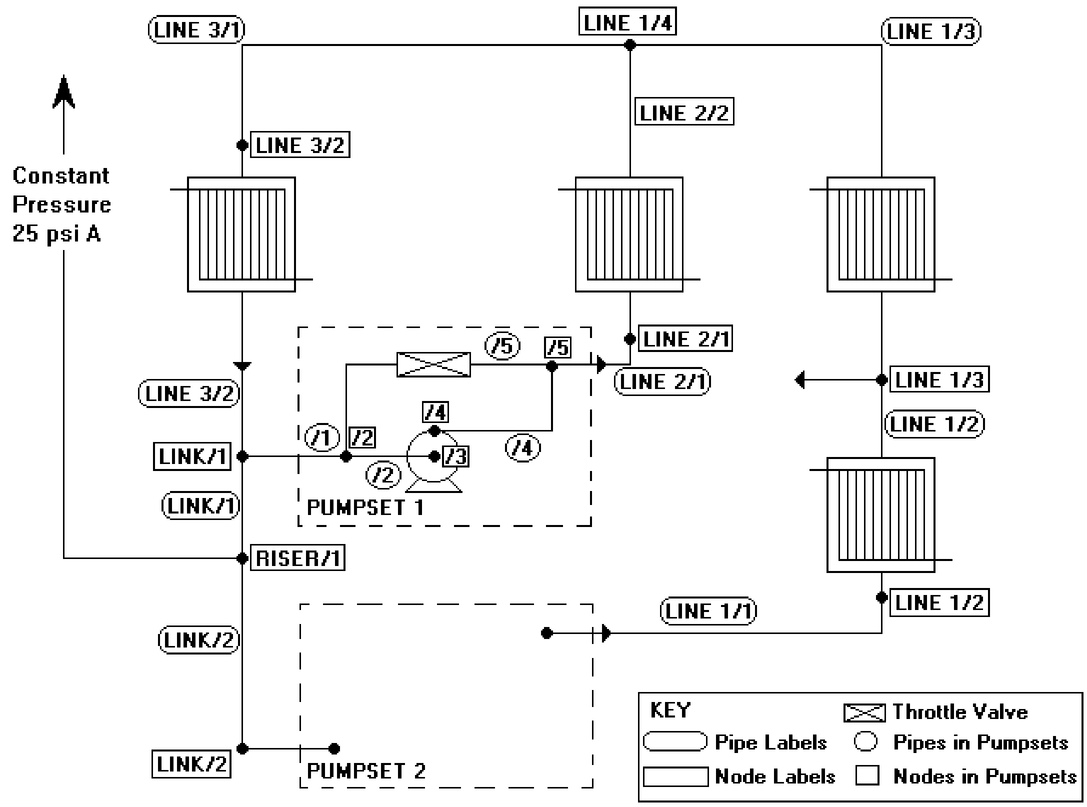
7.1 Closed loop cooling system

This example will consider a closed loop cooling system and it will illustrate the following points:

- How to deal with varying fluid temperature when this affects the transport properties.
- How to model miscellaneous network items such as Heat Exchangers.
- The use of pumps from a Pump/Fan Preprocessor Library File.
- Ways of modelling a closed loop system.

The files relating to this example are supplied with PIPENET, and are: _cooling.sdf, and the associated library file _cooling.slf.

The network



The network, as shown above, is a closed loop cooling system. The coolant is circulated through four heat-exchangers by two identical pump sets, each of which operates with local recycle, controlled by a throttle valve. After passing through the exchangers the coolant streams are combined, chilled and returned to the recycle pump inlets.

The components and nodes in PUMPSET 1 are labeled with the tag 'PS1' - this tag has been omitted from the diagram. PUMPSET 2 is identical to pump set 1 except all node and component labels have the tag PS2.

Running the simulation

We will split this example into Initialisation, Pump data, Network data entry, Specifications data entry and Calculation and results.

7.2 Initialisation

Units

The units used are user-defined and are as follows:

Measure	Unit
Length	feet
Diameter	inches
Velocity	ft/s
Temperature	Celcius
Viscosity	cP
Density	lb/ft3
Pressure	psi Absolute
Flow type	Mass flow
Mass flow units	lb/s

Fluid Type

The fluid (the coolant) is a glycol-water mixture at 2°C. Its density and viscosity vary with temperature, T, according to the correlation equations:

$$Density = AB^{-(1-\frac{T}{T_c})^{\frac{2}{7}}}$$
$$Viscosity = CT^{M}$$

where the coefficients ***A***, ***B***, ***C*** and ***M*** and the critical temperature, T_c, of the fluid are as follows:

A = 41.97 lb/ft^3
B = 0.6043
C = 2.10E18 Cp
M = -7.362
T_c = 328.2°C

Note that temperatures ***T*** and T_c are degrees Kelvin.

Exchanger and Fittings

For simplicity, each exchanger is represented as a fitting on the section of pipe downstream from the exchanger. A velocity head loss coefficient (k-factor) of 3.5 is assigned to the exchanger. The fittings used in the system are as follows:

Fitting type	K-factor	Fitting name
90° bend	0.75	B90
T junction	1.0	TJUNC
Throttle valve	50.0	THRT
Exchanger	3.5	EXCH

Default Values

All pipes in the network have roughness 0.0018 inches. Enter a Title, Units, Default Values, Fluid Type and Fittings using the appropriate menu options.

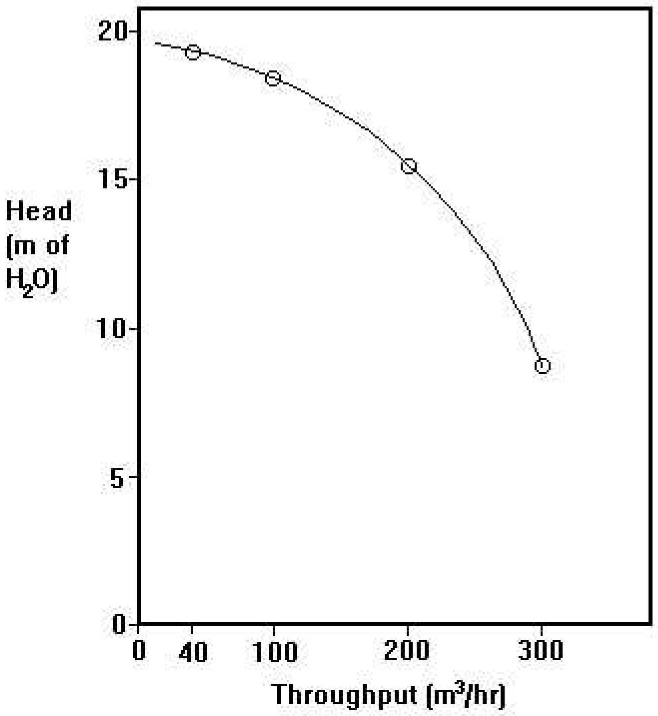
Properties

The default system temperature is 2°C as recorded in the menu option **Options | Fluids**. However, the temperature in pipe LINE1/2 is 20°C and in pipes LINE1/3, LINE2/2 and LINE3/1 the temperature is 40°C.

These temperatures are set by selecting the Property tool  from the toolbar and then placing the property on the appropriate pipe by simply pointing and clicking on the pipe. The position of the symbol representing the property can be changed using the Selection tool and click-dragging the property along the pipe. Only one property may be placed on each pipe.

7.3 Pump data and the library

The characteristic curve of the pump is as shown below.



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

Pump descriptor: TYPE-300
Minimum flow rate = 40 m^3/hr
Maximum flow rate = 300 m^3/hr

The data points are:

Flow rate (m3/hr)	Pressure (metres of water)
40	19.18
100	18.03
200	15.24
300	8.89

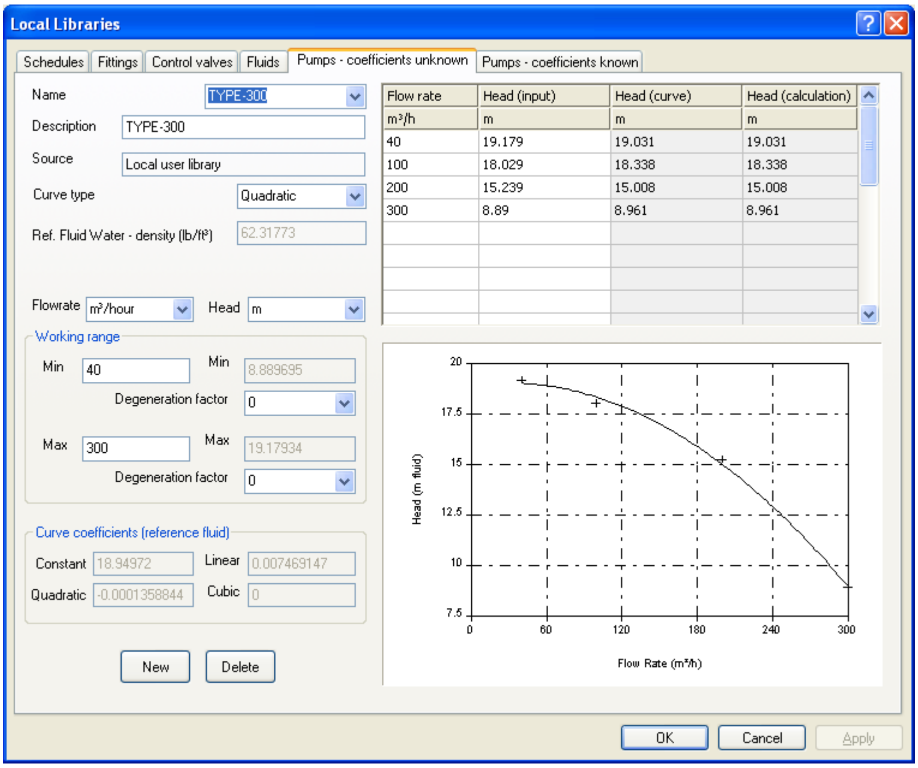
Adding the pump to the library

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

1. Select the **New** button
2. Provide the name for the pump as TYPE-300 (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 40m^3/hour and a maximum flow rate of 300m^3 /hour
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 40.0
6. Tab to the next field and enter the corresponding pressure of 19.18
7. Continue in this way entering the remaining data pairs, using tab to move from one cell to the next
8. When all the data pairs have been entered, select the **Apply** button to calculate and display the pump curve

The completed pump specification should appear as:



For further information on the pump dialog refer to the [libraries section](#).

7.4 Network data entry

The table below gives details about the pipes used in the system.

Pipe	In	Out	Diameter (in)	Length (feet)	Elevation (ft)	Fittings
PS/1	LINK/1	PS1/2	3.826	4.0	0.0	TJUNC
PS1/2	PS1/2	PS1/3	3.826	4.0	-1.0	
PS1/4	PSI/4	PS1/5	3.826	3.0	0.5	

PS1/5	PS1/2	PS1/5	3.826	11.0	-0.5	THRT
PS2/1	LINK/2	PS2/2	3.826	4.0	0.0	TJUNC
PS2/2	PS2/2	PS2/3	3.826	4.0	-1.0	
PS2/4	PS2/4	PS2/5	3.826	3.0	0.5	
PS2/5	PS2/2	PS2/5	3.826	11.0	-0.5	THRT
LINK/2	RISER/1	LINK/2	5.761	5.00.5	0.0	B90
LINE1/1	PS2/5	LINE1/2	5.761	16	2.0	TJUNC + B90
LINE1/2	LINE1/2	LINE1/3	5.761	6.0	4.0	EXCH
LINE1/3	LINE1/3	LINE1/4	5.761	14.0	-2.0	B90 + EXCH
LINE3/1	LINE1/4	LINE3/2	5.761	15.0	0.0	B90
LINE3/2	LINE3/2	LINK/1	5.761	35.0	-3.5	EXCH + 2 *B90
LINE2/1	PS1/514	LINE2/1	3.826	10.0	2.0	TJUNC + B90
LINE2/2	LINE2/1	LINE1/4	3.826	12.0	2.0	EXCH
LINK/1	LINK/1	RISER/1	5.761	5.02.5	5.0	

7.5 Specifications

The network is a closed-loop with a single opening to the atmosphere where a user places an expansion tank to connect to the loop. The pressure at that point is 25 psi absolute, which may be assumed constant whatever the recirculation rates are in the network. In order to analyse a network using PIPENET, there must be at least two I/O nodes. As the network stands, there is only one I/O node: RISER/1 which has a pressure specification of 25 psi A.

The second I/O node and a second specification may be provided in two different ways:

1. Make a gap in the loop at the point where the pressure in the system is known (i.e. at the point where the riser to the expansion tank joins the loop). This creates two free ends, which can then be made I/O nodes with identical pressure specifications of 25 psi A.
2. Declare an I/O node in the middle of the loop with a flow rate specification of zero (i.e. no draw-off). This may be thought of as a drainage tap that is shut. Note that connecting nodes may not be declared I/O nodes if using volumetric flow rate units and variable fluid properties. However, in this case we are using mass flow rate units, and so this method is feasible.

Both methods are equally effective, though the second method is usually better in that it maintains the loop and so allows the program to detect any elevation consistency errors which may be present. We will use the second method in this case.

Our specifications for this example are thus:

RISER/1 as an inlet with pressure 25 psi a.
LINE1/3 as an outlet with flow rate 0 lb/s.

7.6 Calculation and results

Having entered all the data, we can check the data by choosing the menu option **Calc| Check** (or alternatively the **check** button on the calculation toolbar). If there are no errors, we can run the simulation by choosing the option **Calc | Calculation** or the **calculation** button on the calculation toolbar. All of the results can be examined with the browser or via the Tabular view.

Example 5 - Cooling System Using Heat Exchangers

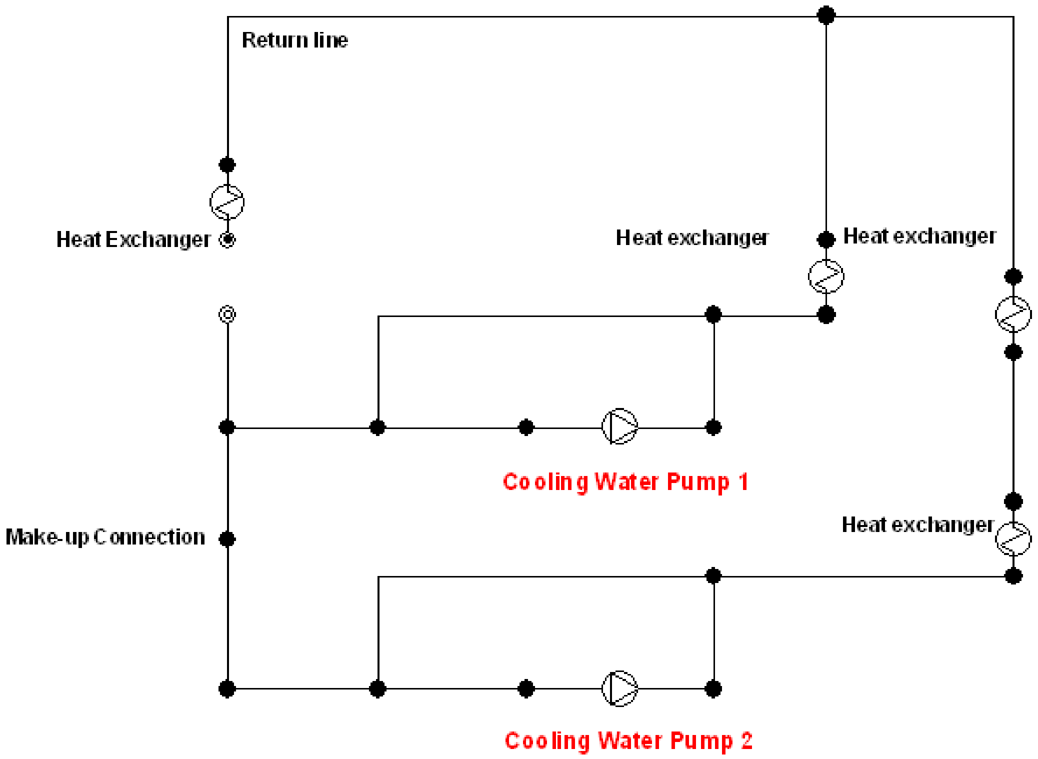
Part 8

AIR CONDITIONING COMPANY

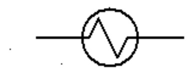
8 Example 5 - Cooling System Using Heat Exchangers

8.1 Cooling System Using Heat Exchangers

This example is essentially the same as the previous example, except that here we use heat exchangers in place of properties on pipes. The basic structure of the network closely resembles that of the previous example, as shown in the completed network diagram below.



with the heat exchangers shown using the component symbol:-



There are a few obvious changes to this network:-

1. The temperature properties have been removed and replaced by heat exchangers
2. The cooling fluid is water since heat exchanges currently only work when the fluid is water.
3. The heat transfer option must be explicitly enabled
4. To the left of the above network diagram observe that the network is no longer a closed loop, a break has been inserted where the coolant streams are chilled and returned to the recycle pump inlets.

The files relating to this example are supplied with PIPENET, and are: _coolingWithHXCH.sdf, and the associated library file _coolingWithHXCH.slf.

8.2 Initialisation

Units

The units used are user-defined and are as follows:

Measure	Unit
Length	feet
Diameter	inches
Velocity	ft/s
Temperature	Celcius
Viscosity	cP
Density	lb/ft3
Pressure	psi Absolute
Flow type	Mass flow
Mass flow units	lb/s
Power	kW

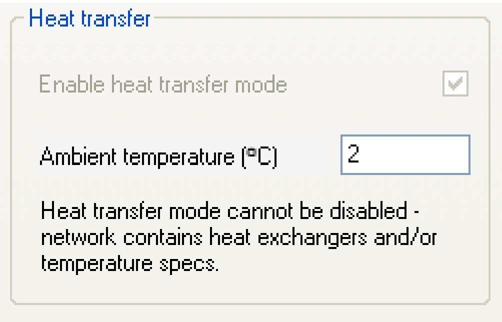
Note the addition of a specification for the units of power.

Fluid Type

The fluid type is set to water, this is necessary to subsequently be able to select the heat transfer mode. It is not necessary to specify the water temperature, leave it at its default value.

Module Options

In the Module Options dialog (selected via the Options | Module options menu option) it is necessary to select the heat transfer mode together with an ambient temperature, which here is et to 2° C:-



Fittings

The fittings used in the system are as follows:

Fitting type	K-factor	Fitting name
90° bend	0.75	B90
T junction	1.0	TJUNC
Throttle valve	50.0	THRT

Default Values
All pipes in the network have roughness 0.0018 inches. Enter a Title, Units, Default Values and Fittings using the appropriate menu options.

8.3 Pump data and the library

The characteristics of the pump and its specification are as described in the previous example.

8.4 Network data entry

Pipes
The table below gives details about the pipes used in the system.

Pipe	In	Out	Diameter (in)	Length (feet)	Elevation (ft)	Fittings
PS/1	LINK/1	PS1/2	3.826	4.0	0.0	TJUNC
PS1/2	PS1/2	PS1/3	3.826	4.0	-1.0	
PS1/4	PSI/4	PS1/5	3.826	3.0	0.5	
PS1/5	PS1/2	PS1/5	3.826	11.0	-0.5	THRT
PS2/1	LINK/2	PS2/2	3.826	4.0	0.0	TJUNC
PS2/2	PS2/2	PS2/3	3.826	4.0	-1.0	
PS2/4	PS2/4	PS2/5	3.826	3.0	0.5	
PS2/5	PS2/2	PS2/5	3.826	11.0	-0.5	THRT
LINK/2	RISER/1	LINK/2	5.761	5.00.5	0.0	B90
LINE1/1	PS2/5	LINE1/2	5.761	16	2.0	TJUNC + B90
LINE1/2	LINE1/2	LINE1/3	5.761	6.0	4.0	
LINE1/3	LINE1/3	LINE1/4	5.761	14.0	-2.0	B90
LINE3/1	LINE1/4	LINE3/2	5.761	15.0	0.0	B90
LINE3/2	LINE3/2	LINK/1	5.761	35.0	-3.5	2 *B90
LINE2/1	PS1/514	LINE2/1	3.826	10.0	2.0	TJUNC + B90
LINE2/2	LINE2/1	LINE1/4	3.826	12.0	2.0	
LINK/1	LINK/1	RISER/1	5.761	5.02.5	5.0	

.

Heat Exchangers
Add the four heat exchangers to the network as shown in the diagram of the introduction. The properties for all four exchangers are:-

- Heat transfer rate = 3000 kW
- Reference flow rate = 37.08 lb/s
- Reference pressure drop = 0.254 psi

So the Properties window for each exchanger appears as

Label	4	
Input node	LINE3/2	
Output node	5	
Heat transfer rate	3000	kW
Reference flow rate	37.08	lb/sec
Reference pressure drop	0.254	psi
Results		
Input pressure	n/a	
Output pressure	n/a	
Input flow rate	n/a	

8.5 Specifications

Unlike the previous example the system is no longer a closed loop, so we simply provide the two I/O nodes with the same pressure specification of 25 psi A. This is in fact the first option discussed in the Specification section of the previous example.

For this system we must also provide a temperature specification on the input node 4. In the option for temperature specification select Yes and then enter a temperature of 2° C in the temperature field, the Properties window appears as:-

Label	4	
Elevation	0	ft
Input/Output node	Input	
Design Spec.	NO	
Analysis Spec.	YES	
Pressure	25	psi A
Flow	Unset	lb/sec
Temperature Spec.	YES	
Temperature	2	°C
Results		
Pressure	n/a	
Flow Rate	n/a	
Outflow temperature	n/a	

In summary our specifications for this example are:

Node **4** as an inlet outlet with pressure 25 psi a and a temperature specification of 2° C.
Node **5** as an outlet with pressure 25 psi a.

8.6 Calculation and results

Having entered all the data, we can check the data by choosing the menu option **Calc| Check** (or alternatively the **check** button on the calculation toolbar). If there are no errors, we can run the simulation by choosing the option **Calc | Calculation** or the **calculation** button on the calculation toolbar. All of the results can be examined with the browser or via the Tabular view.

Note that we now have results returned for heat exchangers, available in the Dat View and in the Properties window. For example the results for the heat exchanger closest to the output are:-

Label	4	
Input node	LINE3/2	
Output node	5	
Heat transfer rate	3000	kW
Reference flow rate	37.08	lb/sec
Reference pressure drop	0.254	psi
Results		
Input pressure	29.52818	psi A
Output pressure	25.00001	psi A
Input flow rate	156.5612	lb/sec
Output flow rate	156.5612	lb/sec
Input temperature	32.3	°C
Output temperature	42.4	°C
Temperature change	10.1	°C
Av. specific heat capacity	4178.4614	J/kg K

Example 6 - Design of a Steam Network

Part 9

9 Example 6 - Design of a Steam Network

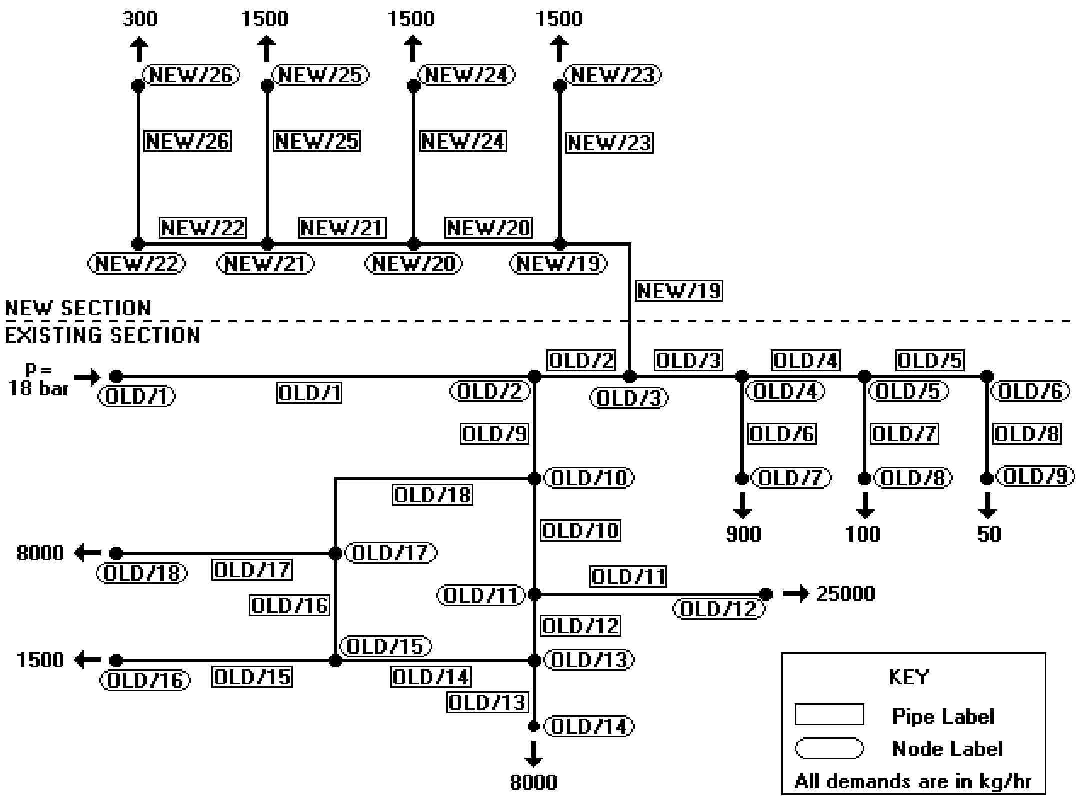
9.1 Design of a steam network

In this example we look at the design of part of the high pressure section of a steam utility system. The example illustrates the following:

- Production of a Private Data File.
- Use of PIPENET's Design facility to find optimum pipe sizes.

The files relating to this example are: _steam.dat and steam.pdf

The network



The network under consideration is the high pressure section of a steam utility system. The existing system is to be extended and pipe work to supply four new outlets is to be added. The above diagram shows our proposed new network. The existing network has labels with the tag 'OLD', whilst the proposed new section is labeled with the tag 'NEW'. We will use PIPENET's Design Facility to help us size the new pipes in the system. Steam is available at the header inlet at 18 Bar G and 230°C. The outlet demands are shown on the diagram (in units of kg/hr).

9.2 Initialisation

Title

Enter a suitable title, for example "Example 5 - Design of a Steam Network**"**

Units

The units used are user-defined and are as follows:

Measure	Unit
Length	metres
Diameter	mm
Velocity	m/s
Temperature	Celcius
Viscosity	cP
Density	kg/m^3
Pressure	Bar gauge
Flow type	Mass flow
Mass flow units	kg/hr

Fluid

The fluid is steam at a constant temperature of 230°C.

Design data

Given a list of available pipe sizes and the maximum permitted velocity of fluid in the network, PIPENET will choose an appropriate size for the pipes in the new part of the network. The user must supply data about the pipe schedule in use. The pipe schedule used in this example is steel piping with a roughness of 0.0457, and comes in the following sizes:

Nominal size (mm)	Actual internal diameter (mm)
25	24.31
40	38.10
50	49.25
80	73.66
100	97.18
150	146.33
200	193.68
250	242.87
300	288.90
350	317.50
400	363.52

Design Velocity

When defining a pipe type a maximum **design velocity** can be provided for each pipe size.

If not specified (value is shown as *Unset* - see Pipe Type dialog in the following section) a default maximum velocity will be assumed. For gases (including Steam) this default maximum is 20.4 m/s, for liquids it is 2.25 m/s.

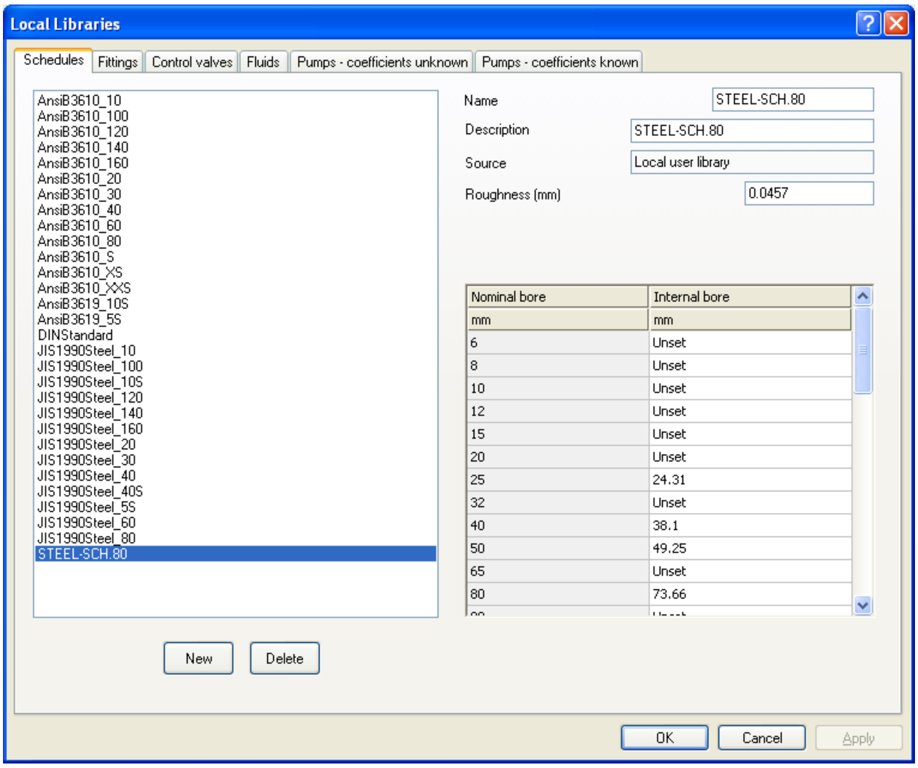
Design phase specifications

PIPENET must also be given a full set of specifications for the design stage of the calculation. These design specifications must satisfy the same conditions as the calculation specifications, with the additional constraint that there must be exactly one pressure specification in the network. The design specifications for this example are the same as the calculation specifications.

9.3 Creating a pipe type

Entering the Pipe Schedule

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



In the left-hand window is a list of all currently defined schedules; selecting an item in this list displays the schedules properties on the right-hand side of the dialog. Data for 29 pipe schedules are built into the PIPENET Standard module. A non-built-in schedule in the list can be edited by simply selecting it in the left-hand window and editing the various attributes that appear in the right-hand side. Note that built-in schedules cannot be edited.

To add data for a schedule select the New button and provide:

- The schedule name, Schedule 80 (this is the name that will appear in the left-hand

window when the data entry is complete).
- An optional description (if not specified the name will be the same as the schedule name).
- A roughness value of 0.0457 mm.

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

Now place the cursor in the internal bore field corresponding to a nominal bore of 25.00 mm and enter the value 24.003 (from table above). Selecting tab moves to the next field which we can skip (leaving the value as unset). Selecting tab again moves to the next field where we enter the value 38.10. Continue in this way until the last diameter has been entered. Select the **Apply** button - changes are accepted and the new schedule appears in the list in the left-hand window.

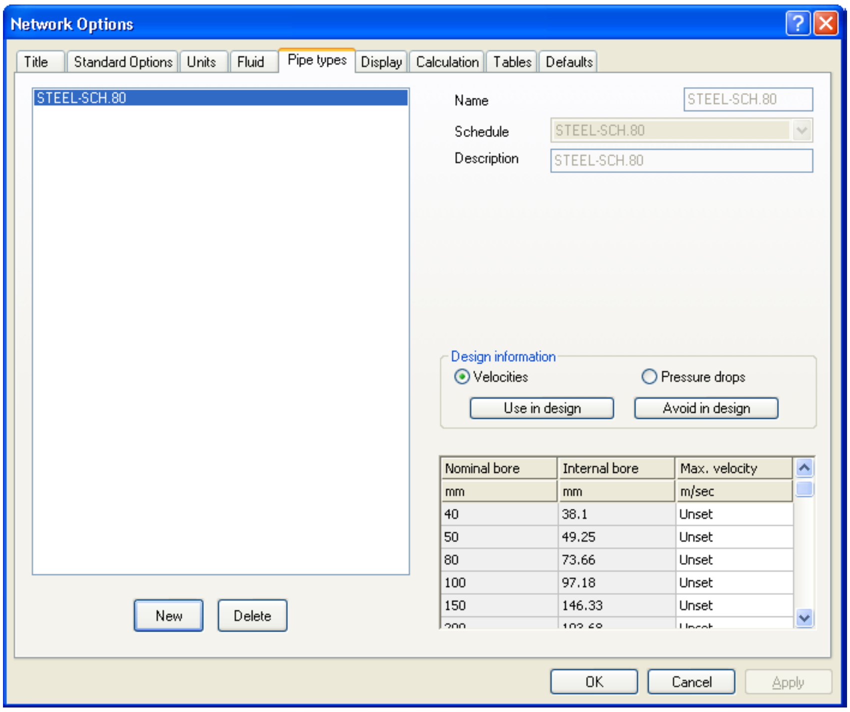
If you are happy with the displayed results you can either select **OK** to quit the dialogs or, since we going to define other library items, simply select another tab, specifically the Nozzles tab since we are going to define a library nozzle in the next section.

Creating the pipe type

Before we can create any pipes we must first create a pipe type. This is done using the menu option **Options| Pipe types** . Proceed as follows:

1. Select **New.**
2. Select the corresponding schedule from the schedule drop-down, this is the first editable box on the right-hand side of the dialog - the pipe type name becomes the schedule name.
3. Select the **Apply** button to accept the pipe type.

The dialog should appear as:



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.

9.4 Network data entry

The table below gives details about the pipes used in the system:

Pipe label	Input node	Output node	Nominal bore (mm)	Length (metres)	Fittings (head loss)
OLD/1	OLD/1	OLD/2	300	15.0	
OLD/2	OLD/2	OLD/3	300	60.0	
OLD/3	OLD/3	OLD/4	300	9.0	
OLD/4	OLD/4	OLD/5	300	6.0	
OLD/5	OLD/5	OLD/6	300	6.0	
OLD/6	OLD/4	OLD/7	50	9.0	1.0
OLD/7	OLD/5	OLD/8	25	9.0	1.0
OLD/8	OLD/6	OLD/9	25	9.0	0.5
OLD/9	OLD/2	OLD/10	250	18.0	1.0
OLD/10	OLD/10	OLD/11	250	9.0	
OLD/11	OLD/11	OLD/12	200	9.0	1.0
OLD/12	OLD/11	OLD/13	250	9.0	
OLD/13	OLD/13	OLD/14	150	9.0	1.0
OLD/14	OLD/15	OLD/13	250	9.0	1.0
OLD/15	OLD/15	OLD/16	50	9.0	1.0
OLD/16	OLD/17	OLD/15	250	9.0	

OLD/17	OLD/17	OLD/18	150	9.0	1.0
OLD/18	OLD/10	OLD/17	250	18.0	1.5
NEW/1	OLD/3	NEW/19		60.0	1.5
NEW/20	NEW/19	NEW/20		6.0	
NEW/21	NEW/20	NEW/21		6.0	
NEW/22	NEW/21	NEW/22		6.0	
NEW/23	NEW/19	NEW/23		9.0	1.0
NEW/24	NEW/20	NEW/24		9.0	1.0
NEW/25	NEW/21	NEW/25		9.0	1.0
NEW/26	NEW/22	NEW/26		9.0	1.0

The pipe data for the network is given in the above table. Note that all pipes have a roughness of 0.0457 (from the pipe schedule data) and an elevation of 0m.

Note the following:

1. Where a pipe diameter is left unset PIPENET will calculate suitable sizes for these pipes in the Design phase of the simulation.
2. Where a pipe's diameter is given it must be a nominal diameter from the pipe schedule used.

9.5 Specifications

The specifications for both the design and calculation phases are as shown below. Enter the specification data as in previous examples via the properties window. Remember that the same specifications must be made for both the calculation phase and design phase of the simulation.

Node label	Pressure (Bar G)	Flow rate (kg/hour)	I/O
OLD/1	18.00		In
OLD/7		900	Out
OLD/8		100	Out
OLD/9		50	Out
OLD/12		25000	Out
OLD/14		8000	Out
OLD/16		1500	Out
OLD/18		8000	Out
NEW/23		1500	Out
NEW/24		1500	Out
NEW/25		1500	Out
NEW/26		300	Out

9.6 Calculation and results

Having entered all the data, we can check the data by choosing the menu option **Calc| Check** (or alternatively the check button on the calculation toolbar). If there are no errors, we can run the simulation by choosing the option **Calc | Calculation** or the calculation button on the calculation toolbar. All of the results can be examined with the browser or via the Tabular view.

Part 10

10 The Schematic

10.1 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.

10.2 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.

10.3 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

10.4 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.

10.5 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 whist 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.

10.6 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

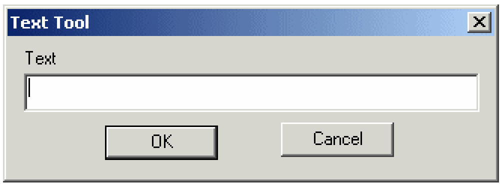
The polygon and all selected items within the polygon can be moved by selecting the left mouse button and whist it is down dragging the polygon to a new position. On releasing the mouse button the components will be drawn at the new position.

Copying selected items

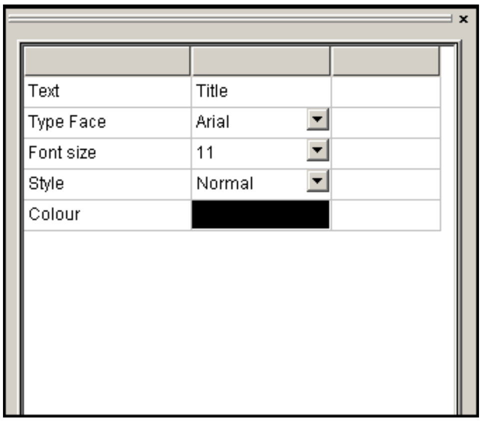
Items selected within the polygon can be copied to the clipboard, from whence they can be pasted onto the same network, or onto another network in a separate instance of the same module. Copy/paste are activated via the **Edit** menu options, the **Copy** and **Paste** buttons on the toolbar, or by right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

10.7 Text Tool

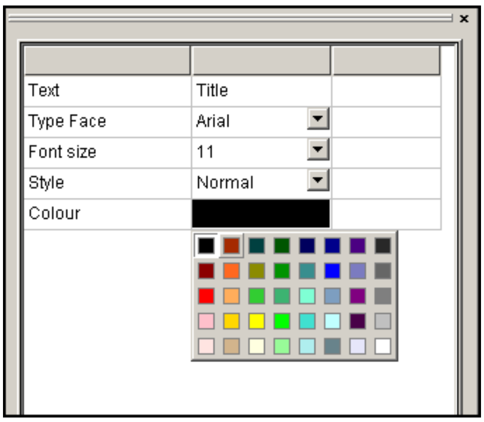
The Text Tool is used to place text on the schematic, for example, as titles and additional labeling information. To place a text item on the schematic click the left mouse at the approximate point at which the first character is to appear; the following dialog appears:



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 right click in the cell and a small selection of colours will be displayed:



Moving and editing text

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

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

10.8 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 and ducts.
5. If you left click without holding down the shift key then the output node is created.

6. If, after creating the pipe or duct, you want to add additional waypoints or to move waypoints, this can be done using the selection tool.
7. The creation of a pipe or duct 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 or ducts then the associated symbol will be displayed on each segment of the pipe or duct.

Undefined or invalid components

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

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

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

10.9 Pipe and Duct Component Tools

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

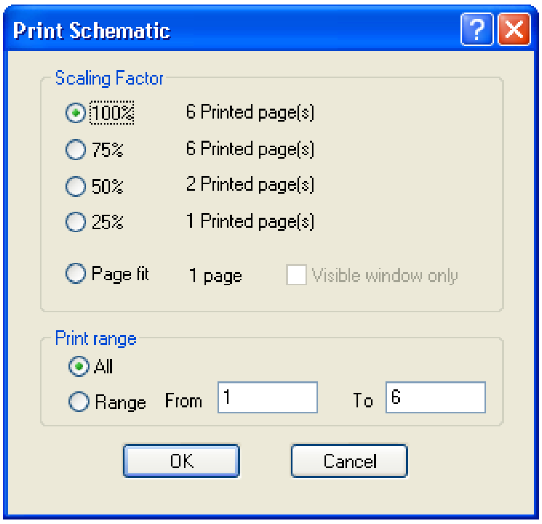
1. Place the cursor on the pipe.
2. Left click to add the attribute element to the pipe.
3. The component may subsequently be moved along the length of the pipe using the selection tool.
4. Attributes for the added component are displayed in the Properties Window, as for link components.

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

10.10 Schematic Printing

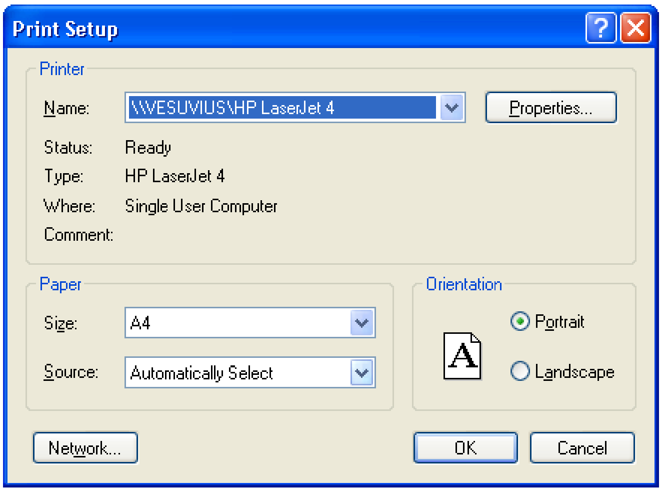
A schematic may be printed by selecting the **File | Print** option. The schematic may be printed to any supported Windows' printer on a single page or across multiple pages. Note however, that printing to a large plotter using a Windows' printer driver may be very slow since some Windows' drivers will 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.

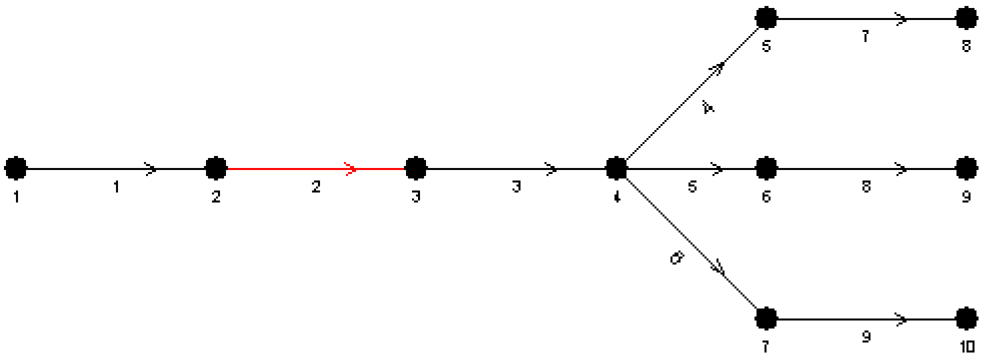
10.11 Exporting the Schematic

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.

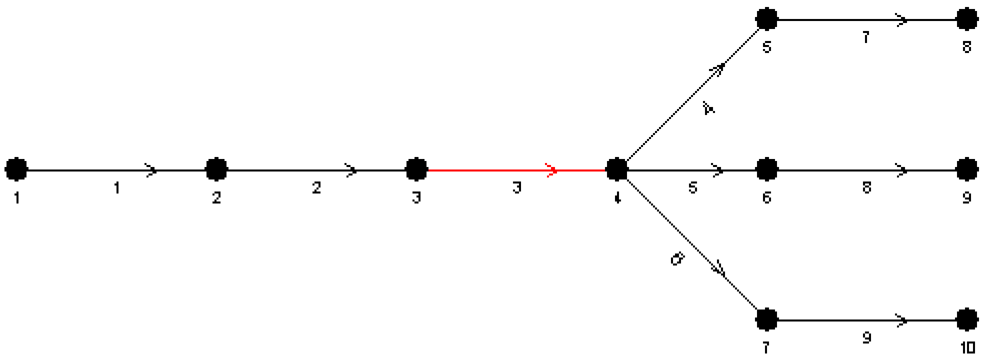
10.12 Moving around the network

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).

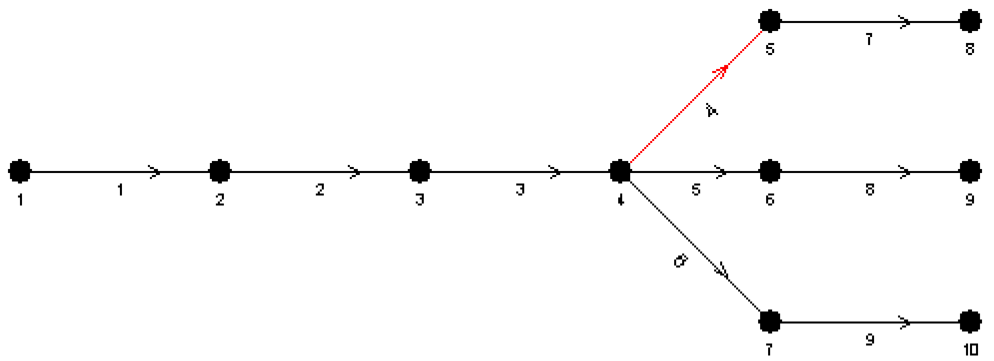
There is another way to move around the network, highlighting components in turn, using the four cursor keys on the keyboard. This is best illustrated by the following example. With pipe 2 the currently selected component and all component directions being from left to right, selecting the right cursor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



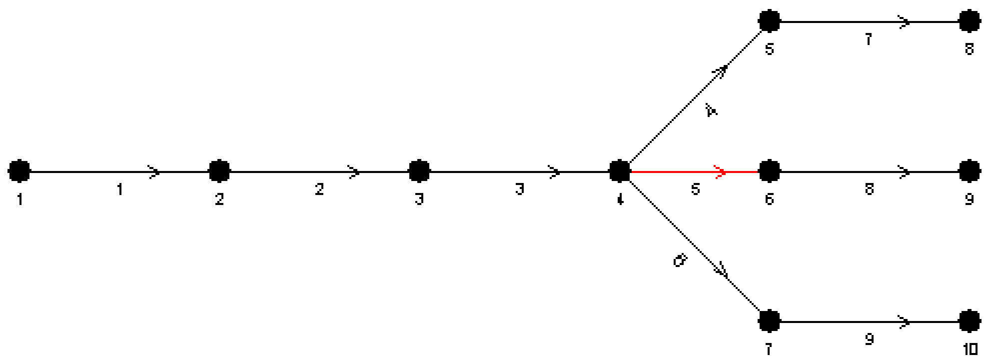
Now consider what happens when we are on pipe 3:



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.

10.13 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 whist holding down the mouse wheel, move the mouse to pan the network.

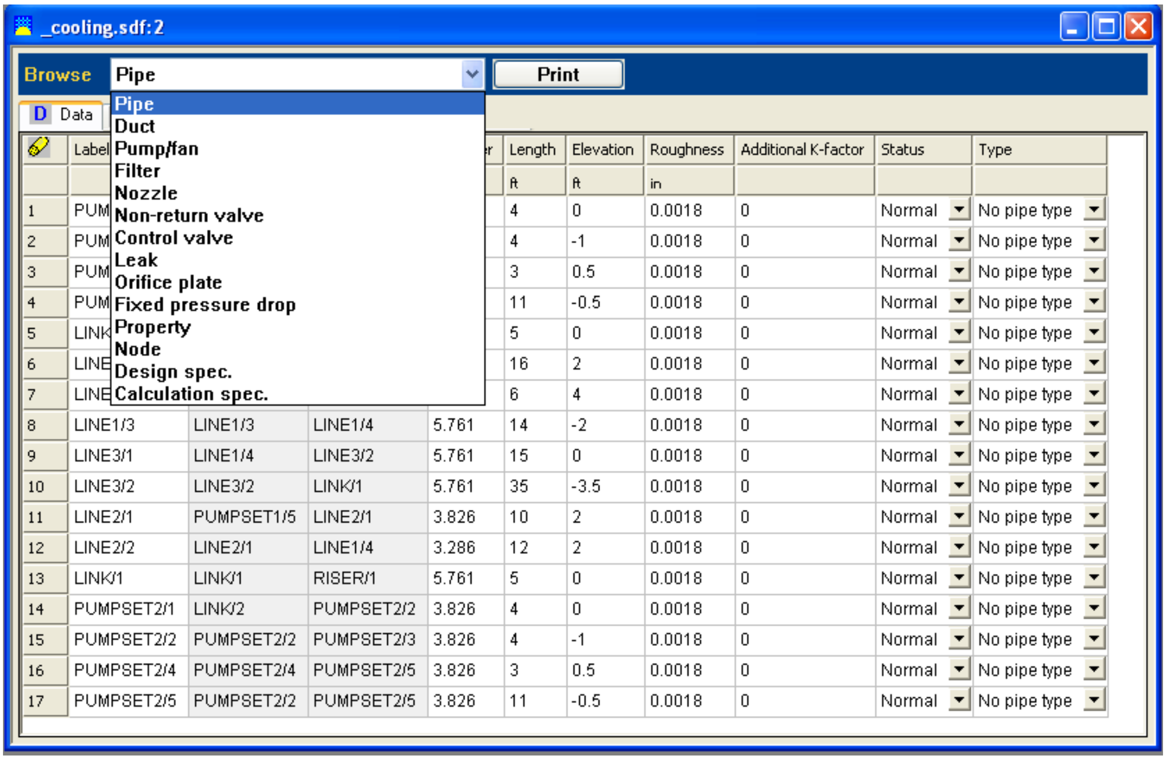
Part 11

11 The Tabular View

11.1 Tabular View

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

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



Sorting

Components are initially displayed in data entry order, however, rows may be 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 an entire row.

Editing a cell

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

Cell edits can be undone using the Undo/Redo facilities.

Printing

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

11.2 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.

11.3 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.

Part 12

12 Specifications

12.1 Introduction to Specifications

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

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

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

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

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

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

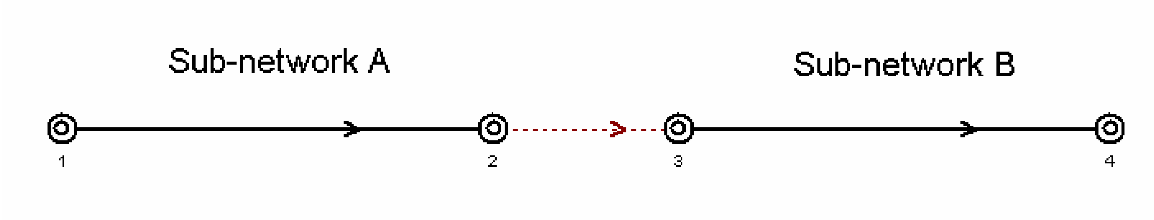
The number of specifications must be equal to the total number of input and output nodes, and at least one of the specifications must be a pressure specification. See Specification Rules for further details and the special considerations which apply to the Design Phase, nozzles and remote specifications.

Disjoint Network

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

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

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



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

12.2 Specification Rules

Assumptions

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

Design phase

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

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

Calculation Phase

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

12.3 Breaks and Blocks

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

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

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

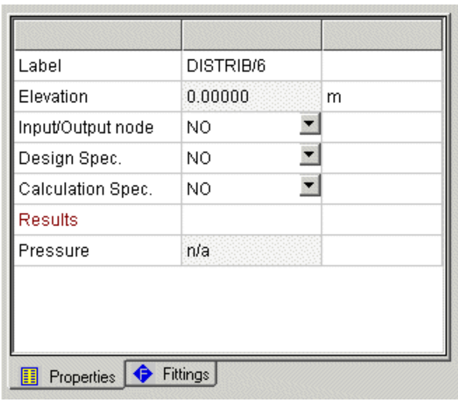
Each break or block may separate a single network into two sub-networks, and since specifications must be valid in each sub-network, problems can occur. The most likely problem to arise is that the presence of a blocked pipe breaks a valid network into two sub-networks with one of the sub-networks having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

12.4 User Interface

Viewing and editing specifications

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

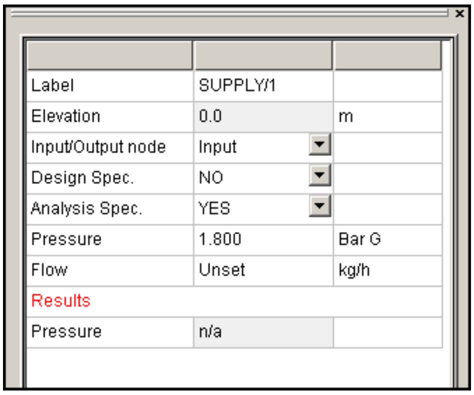
1. by clicking on a node, the details of the specification appear in the Properties window where they may be edited. If no specifications have been attached to the node then the properties window appears as:



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

Adding specifications

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



where the pressure and/or flow can be provided.

Removing specifications

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

Checking specifications

Specifications are checked during the performance of a check operation along with height checking and general consistency checking. This can be initiated by a user selecting the **Calculation | Check** menu option.

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

The errors and warnings are as follows:

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

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

Node has no inputs and is not an I/O node - warning
Taking into account component directions, a node has been found that has no inputs. It may have been the user's intention to associate a specification with the node.

Node has no outputs and is not an I/O node - warning
Taking into account component directions, a node has been found that has no outputs. It may have been the user's intention to associate a specification with the node.

No design pressure specification - error
One, and only one, design pressure must be provided for the design phase.

No design flow specification -error
At least one flow specification must be provided in each sub-network, together with a single pressure specification.

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

No analysis pressure specification - error
At least one design pressure must be provided for each sub-network. See also blocked pipe warning above.

There must be at least two specifications - error
At least two specifications must be provided for each phase.

Network is over specified in analysis phase
More specifications than are necessary have been supplied, that is:

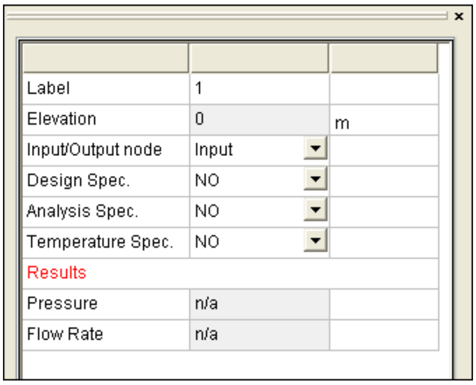
> number of flow specifications + number of Pressure specifications > number of I/O nodes.

Network is under specified in analysis phase
Insufficient specifications have been supplied, that is:

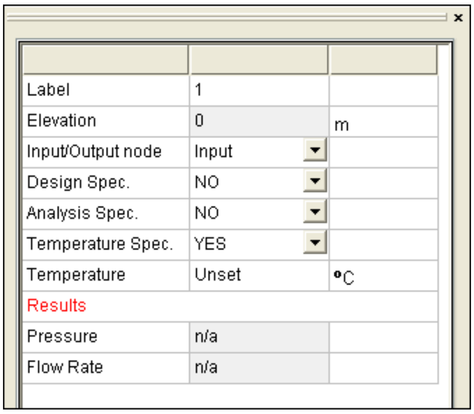
> number of flow specifications + number of Pressure specifications < number of I/O nodes.

12.5 Temperature specifications

These only apply when the heat transfer mode has been selected and are not included when applying the specification rules that apply to flow and pressure specifications. Temperature specifications can only be attached to I/O nodes. If the heat transfer mode has been selected then selecting an I/O node will display the following properties:-



Changing the Temperature spec. option from **No** to **Yes** then results in the following display:-



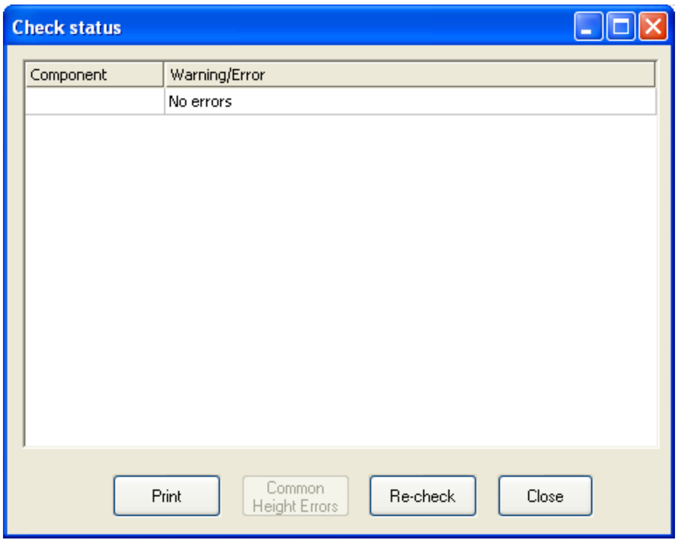
and the temperature at the node can be provided.

Part 13

13 Status Checking

13.1 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.

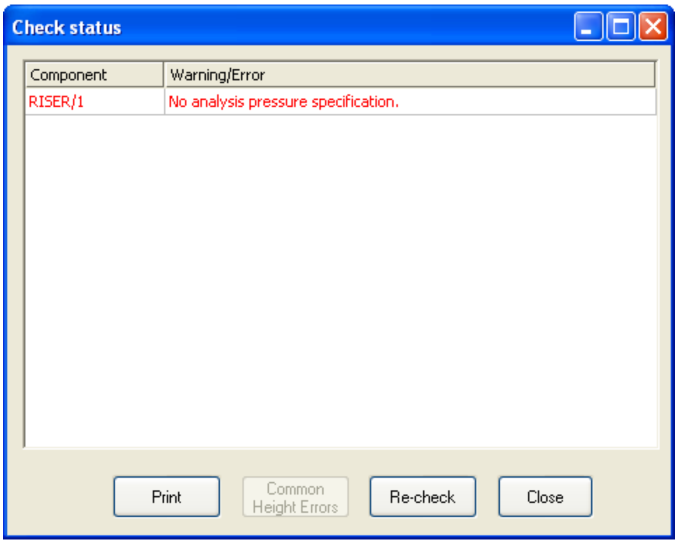
Warnings are shown in back text, and indicate possible problems with the network. Errors are shown in red text and indicate problems that may prevent a successful calculation.

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.

13.2 Specification Checks

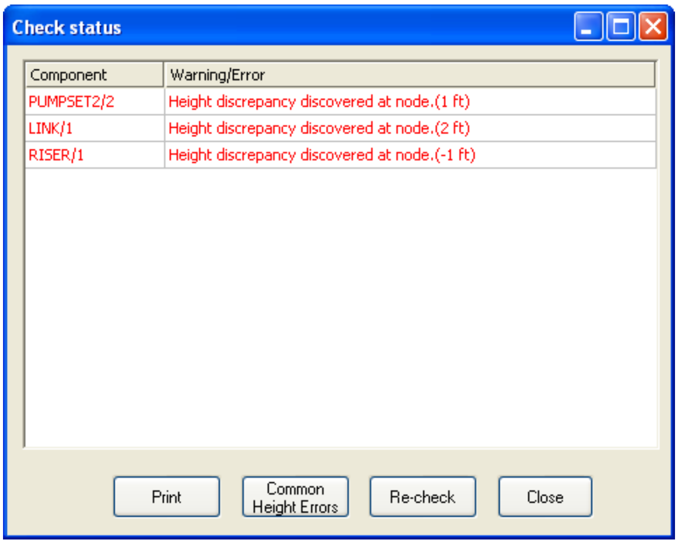
When a check is activated the number of specifications and their type is validated against the rules defined in Specification Rules. The following is the check window displaying a specification error.



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

13.3 Height Checking

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



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

Part 14

14 Colour Schemes

14.1 Colour Schemes

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

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

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

Each colour scheme can assign one of six colours to a component: RED, ORANGE, GREEN, CYAN, BLUE and MAGENTA. 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.

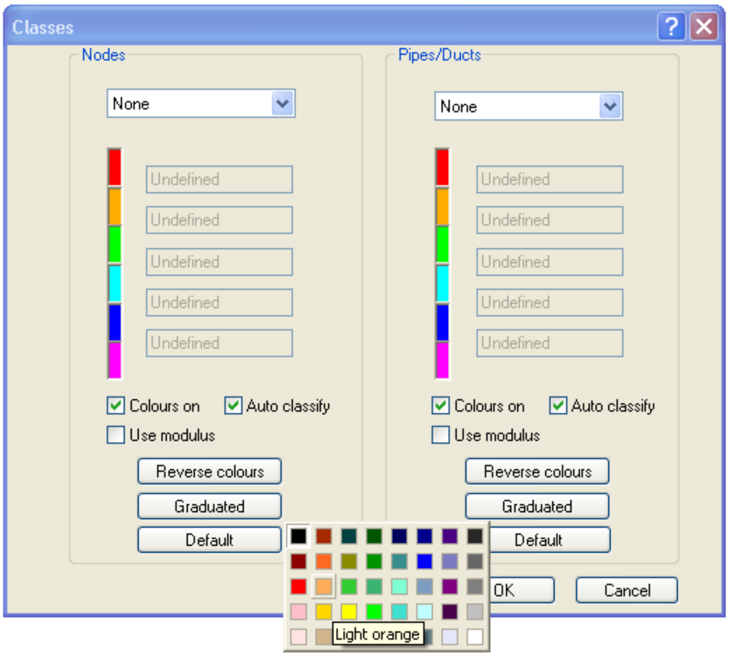
Simple Colour schemes

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 v1, v2, v3, v4, v5, the intervals being:

< v1	RED
ừv1 & < v2	ORANGE
ừv2 & < v3	GREEN
ừv3 & < v4	CYAN
ừv4 & < v5	BLUE
ừv5	MAGENTA

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.

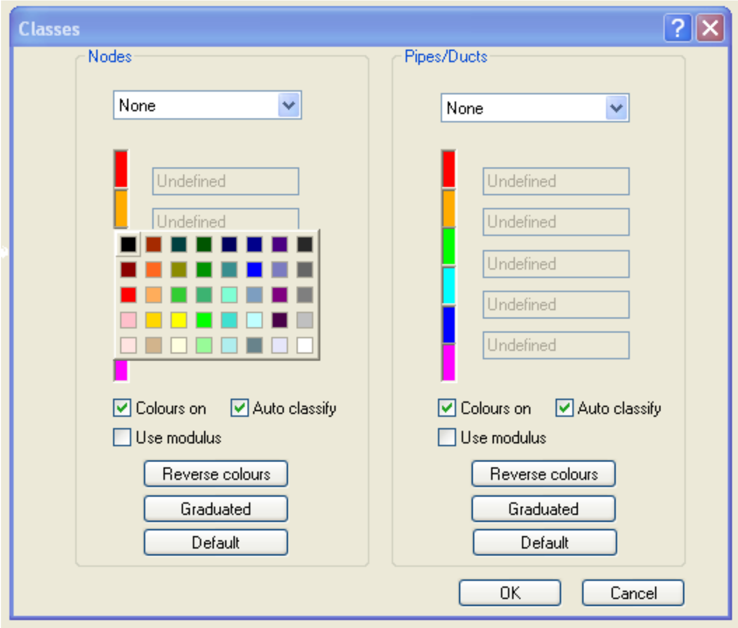
Below the combo box are five edit boxes used for defining the six intervals. The coloured boxes alongside the edit boxes define the colours for the six intervals. The edit boxes will be grayed out if the selected colouring scheme does not use intervals.

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.

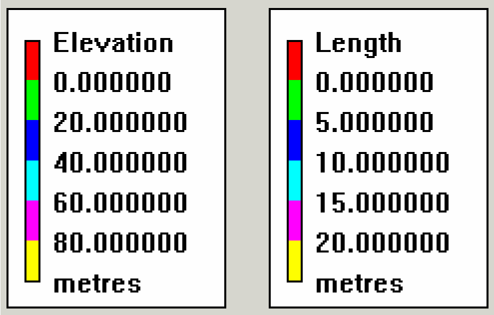
The final three buttons are used to select the way in which the intervals are coloured, the default being the six colours ordered as RED, ORANGE, GREEN, CYAN, BLUE and MAGENTA. The ordering of the colours may be reversed by selecting the Reverse colours button. As an alternative to the six colours a 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/ducts) on the schematic. For example:

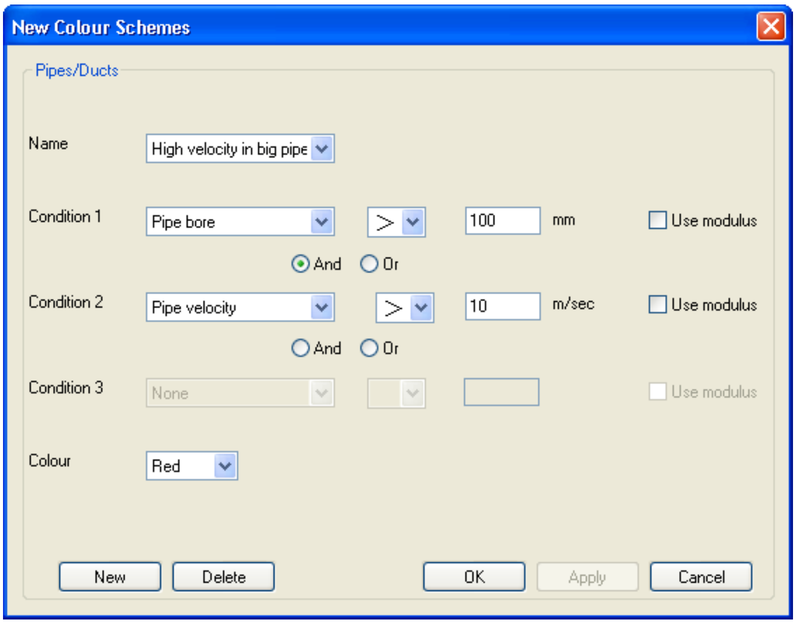


The legends can be moved around by clicking and dragging with the mouse.

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.

Complex rules

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

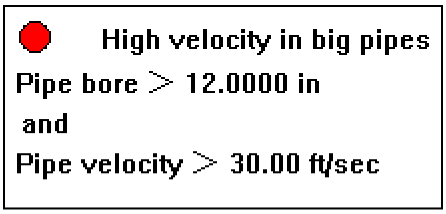


This shows the dialog for entering a complex rule for a pipe or duct, that for a node has the same layout. It comprises:

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), a value, and a Use modulus check box (if the box is checked then the absolute value of the attribute must satisfy the condition).
3. Logical And or Or operators relating the conditions.
4. The colour to be used for display.

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

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



14.2 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** and selecting a colour from the resulting popup menu.
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 of a certain colour can be made untagged by selecting the menu option **Tools | Remove Tags** and selecting the colour from the resulting popup menu, or selecting All to remove all tags.

14.3 Background Colours

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

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

Elevation Profile and Hydraulic Grade Line

Part 15

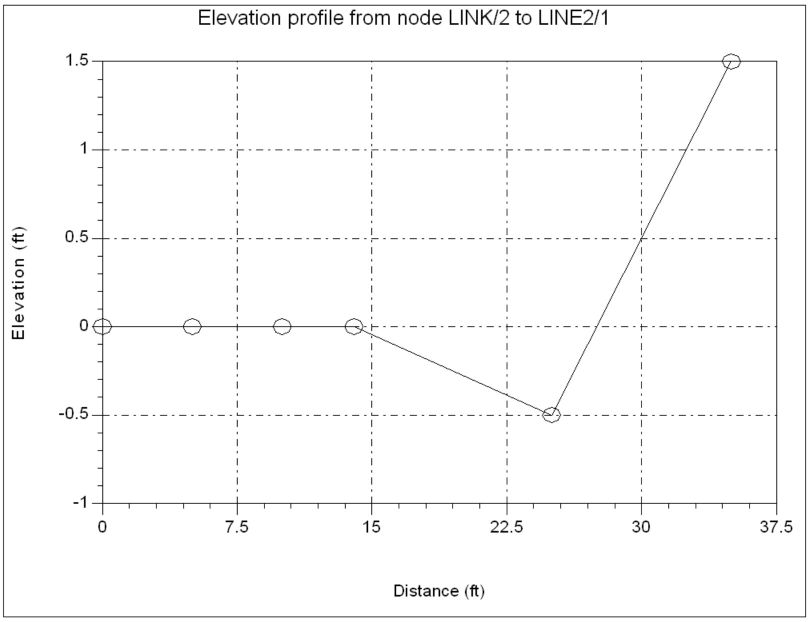
azarnasim

AIR CONDITIONING COMPANY

15 Elevation Profile and Hydraulic Grade Line

15.1 Elevation Profile Graph

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 whist selecting the nodes.

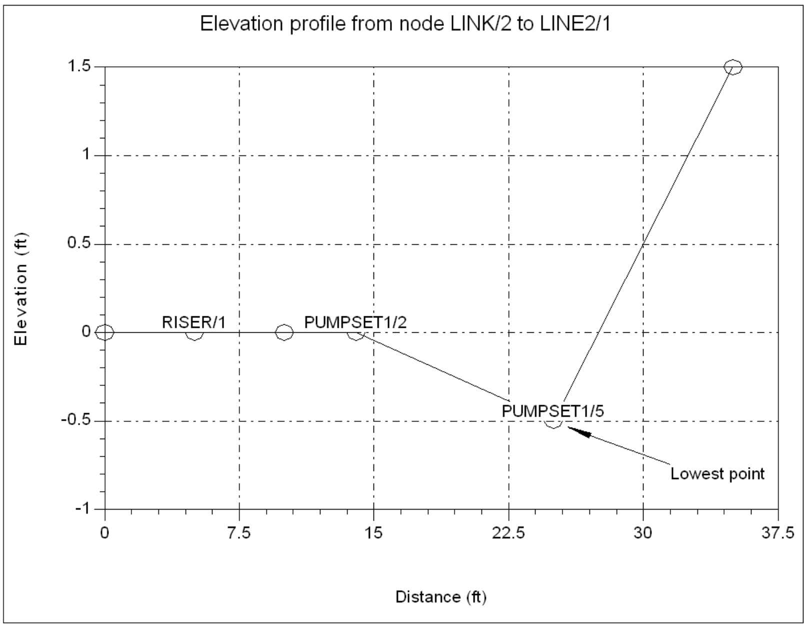
The path found is the shortest path, where shortest means the smallest number of nodes. Having created a path, the elevation profile will be displayed with the nodes plotted from the leftmost of the two nodes selected to the second, rightmost node. The vertical axis displays an elevation scale in the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

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

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

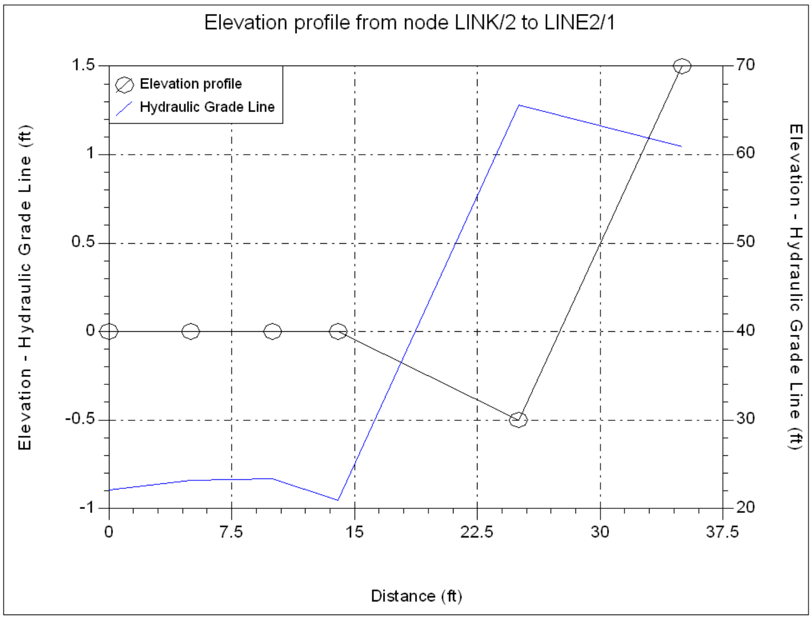
possible to edit the title, labels, styles and the axes. These properties can be saved as a template.

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

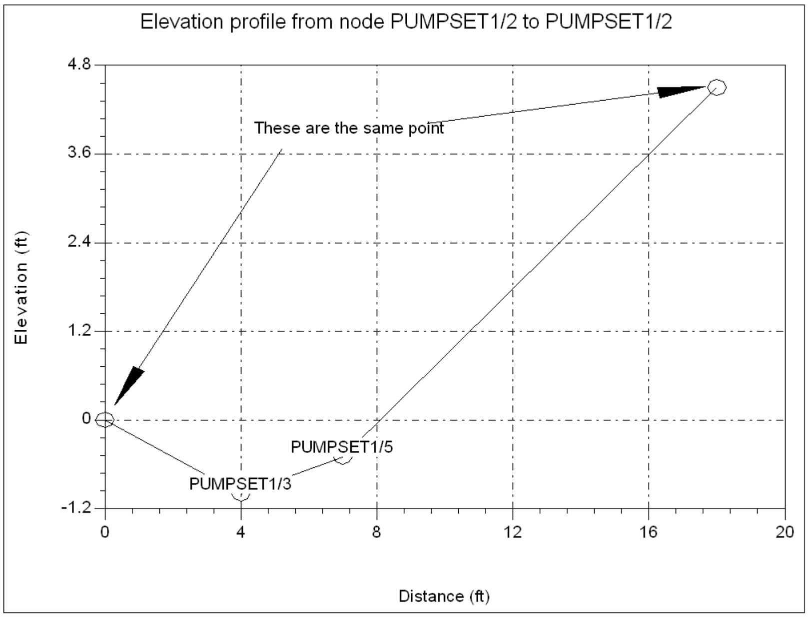


The Elevation profile has a number of uses, the major two being:

1. The display of a Hydraulic Grade Line, the following shows an example of a hydraulic grade line plot (blue) with the elevation profile in black:



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



15.2 Hydraulic Grade Line

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

HGL = SH + EH {with units of length}

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

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

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

The Hydraulic grade line can be displayed in the elevation profile window following a calculation by creating a path between two nodes in the network, using the **Tools | Make path** facility.

Note that for a hydraulic grade line to produced the fluid must either be a liquid with no temperature items in the network or a gas and the temperature unit is other than Kelvin.

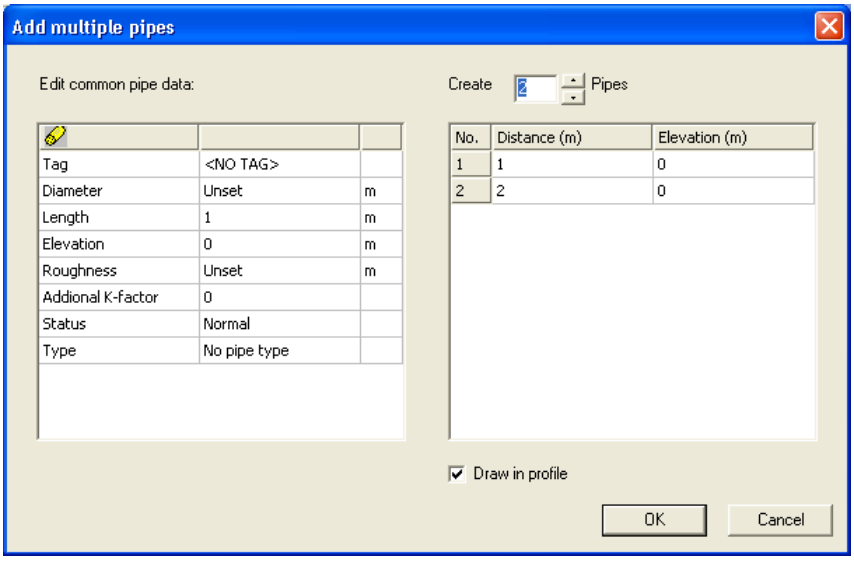
Part 16

16 Add Multiple Pipes

16.1 Add multiple pipes dialog

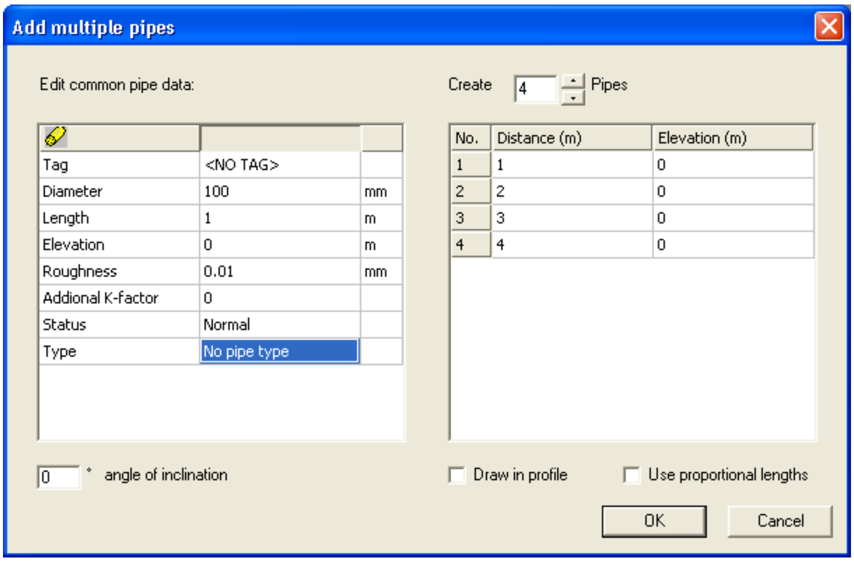
This facility is activated via the **Tool** menu and provides a quick and simple method of generating pipe runs, typically as used in pipeline applications. Here the user may start with a set of distance and elevation pairs, possibly in a spreadsheet form.

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



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

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



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 17

17 Libraries

17.1 Libraries

Two types of library are used in PIPENET modules:

Local User Library

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

- pipe schedules
- fittings
- valves
- fluids
- pumps
- linings

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

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

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

System Library

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

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

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

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

Opening Libraries

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

Importing libraries

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

Exporting Libraries

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

Libraries - example 1 - new data and library file

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

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

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

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

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

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

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

17.2 Library Editor

The Library Editor comprises a number of pages, one for each type of library item, for the Standard module these are:

Schedules - Define or edit pipe schedules.

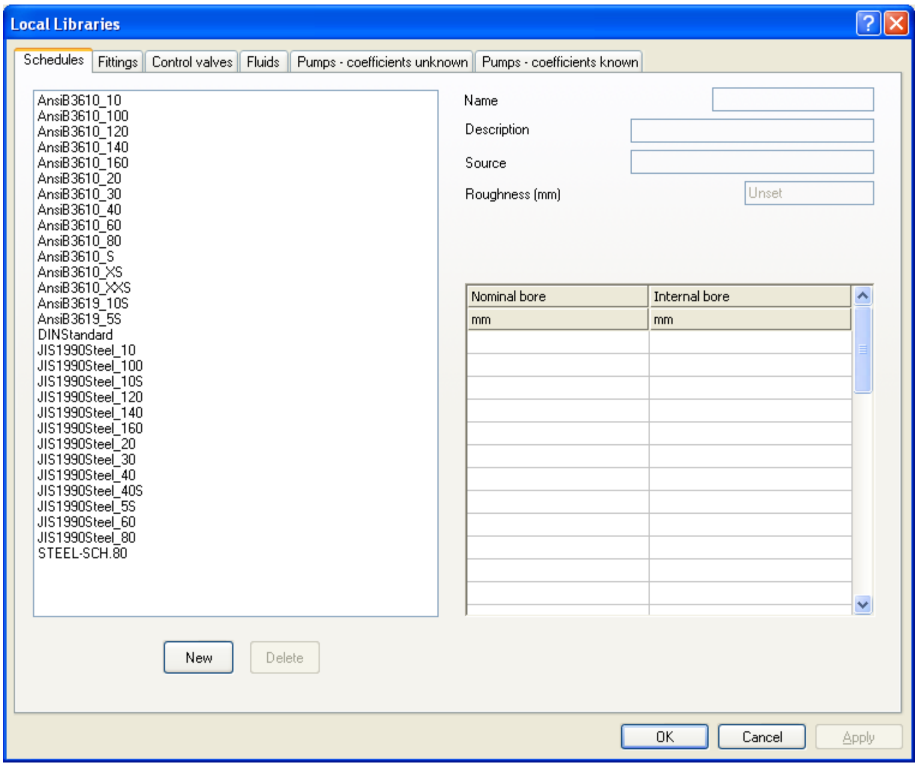
Fittings - Define or edit pipe fittings.

Control valves - Define or edit pipe control valves.

Fluids - Define or edit fluids.

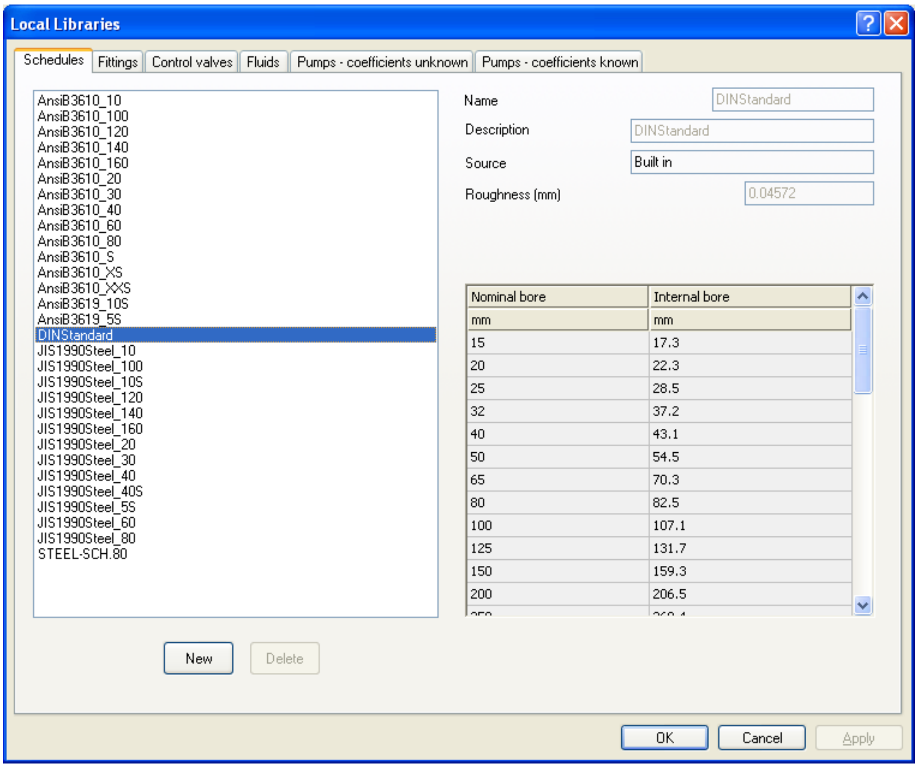
Pumps - coefficients unknown - Define or edit pumps with unknown coefficients.

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



17.3 Pipe Schedules

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



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

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

1. The name of the schedule as it appears in pop-up menus, the length of this name is limited to 20 characters.
2. An optional longer description.
3. The source of the schedule, which may be one of the following:
   - Built-in schedule provided with the 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 using the External System Library Editor.

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

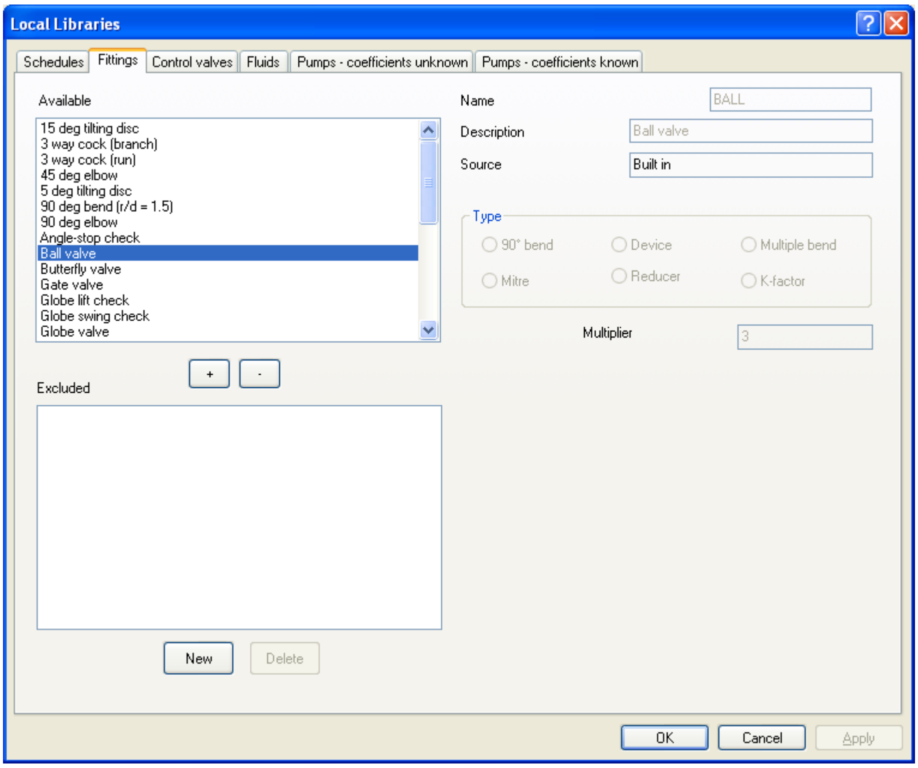
Select the schedule to be edited from the left-hand window, enter the desired changes and then select the **Apply** button to accept the changes.

Deleting an existing schedule

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

17.4 Fittings Library

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 right-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 optional 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.

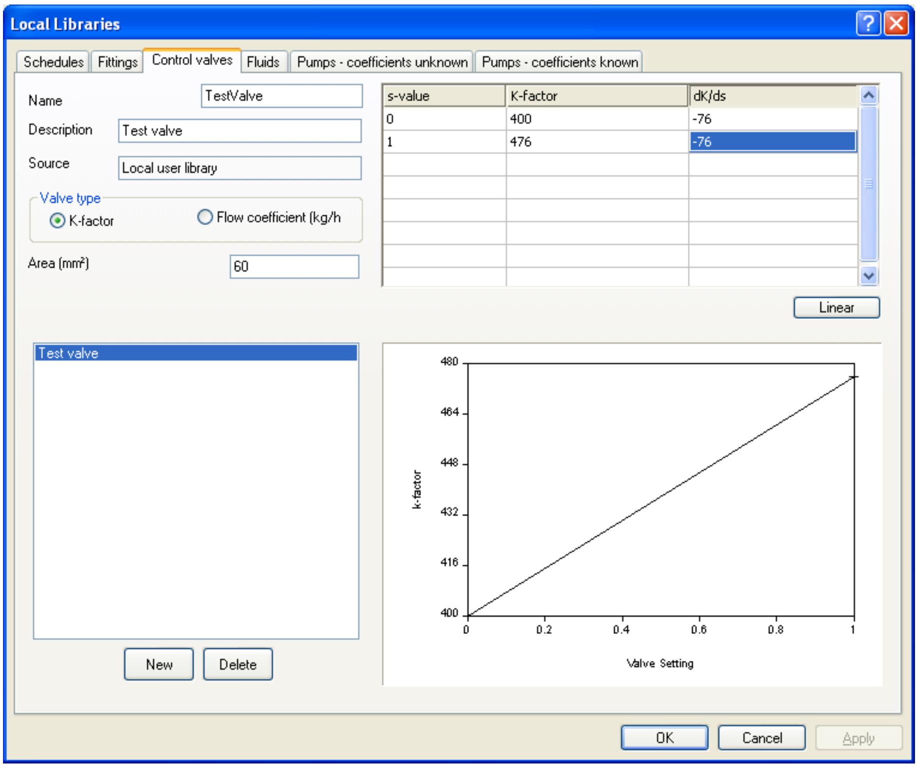
Deleting a fitting

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

17.5 Control Valves

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

1. Select the **New** button.
2. Provide a unique name for the control valve and an optional description.
3. Select the valve type.
4. If K-factor valve type selected enter the value for the area.
5. At least two entries must be provided in the grid.
   - For K-factor valves these consist of a valve setting (between 0.0 and 1.0), a K-factor (which must be positive) and a value for the gradient at each point dK/ds (must be less than or equal to zero).
   - For Flow coefficient valves these consist of a valve setting (between 0.0 and 1.0), a Flow coefficient (which must be positive) and a value for the gradient at each point dCv/ds (must be positive).
6. Select the **Apply** button to add the control valve to the library.



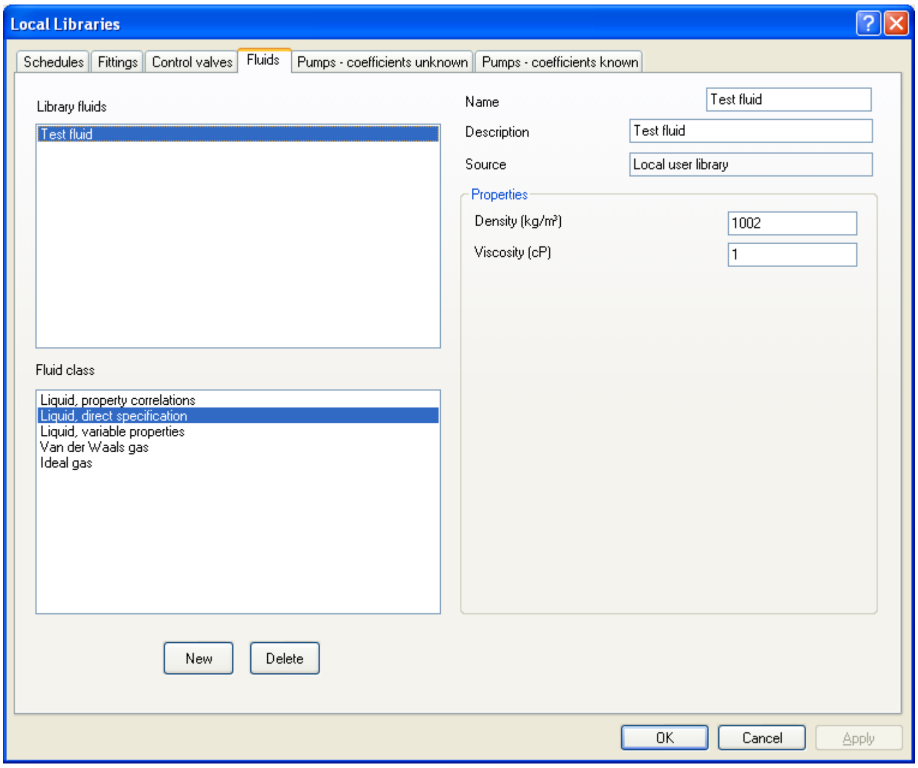
Selecting the **Linear** button (immediately below the grid) will fill in the values for dK/ds or dCv/ds using a linear gradient between each two successive points.

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.

17.6 Fluids library

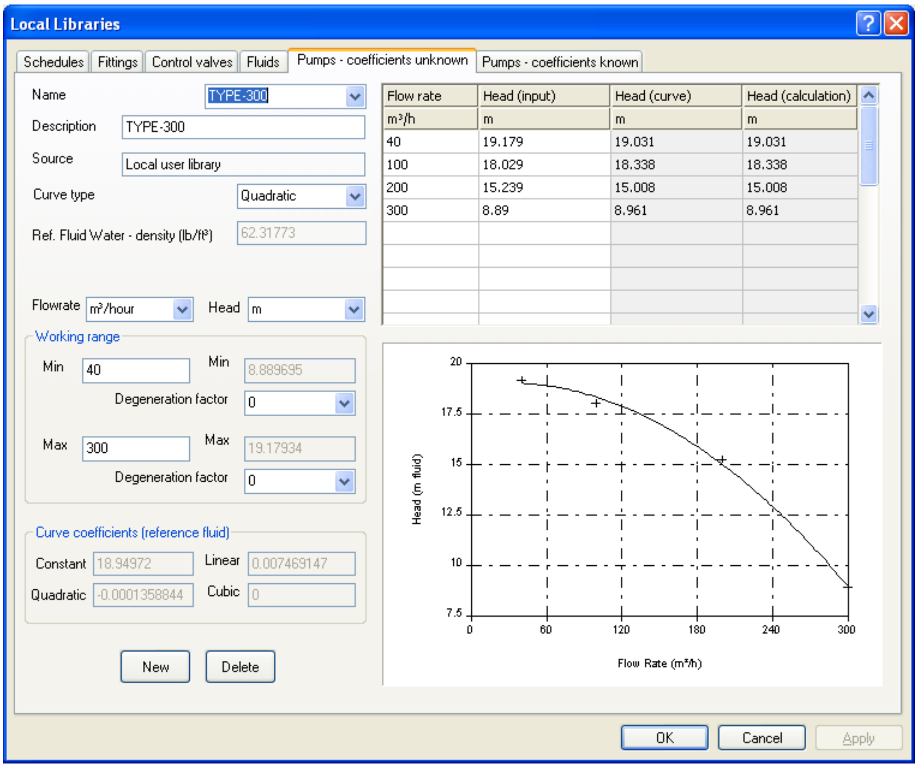
To view a library fluid, select the required fluid from the top left-hand window and its properties are displayed on the right-hand side. To delete a fluid select the fluid in the left-hand window and then select the **Delete** button. To add a new fluid:

1. Select the **New** button.
2. Provide a unique name for the fluid and an optional description.
3. Select the fluid class from the bottom left-hand window, the required properties appear on the right-hand side.
4. Depending on the fluid class selected, enter the required parameters.
5. Select the **Apply** button to add the fluid to the library.



17.7 Pumps - Coefficients Unknown

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



Creating a new pump

To create a new pump:

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

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

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

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

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

Editing an existing pump

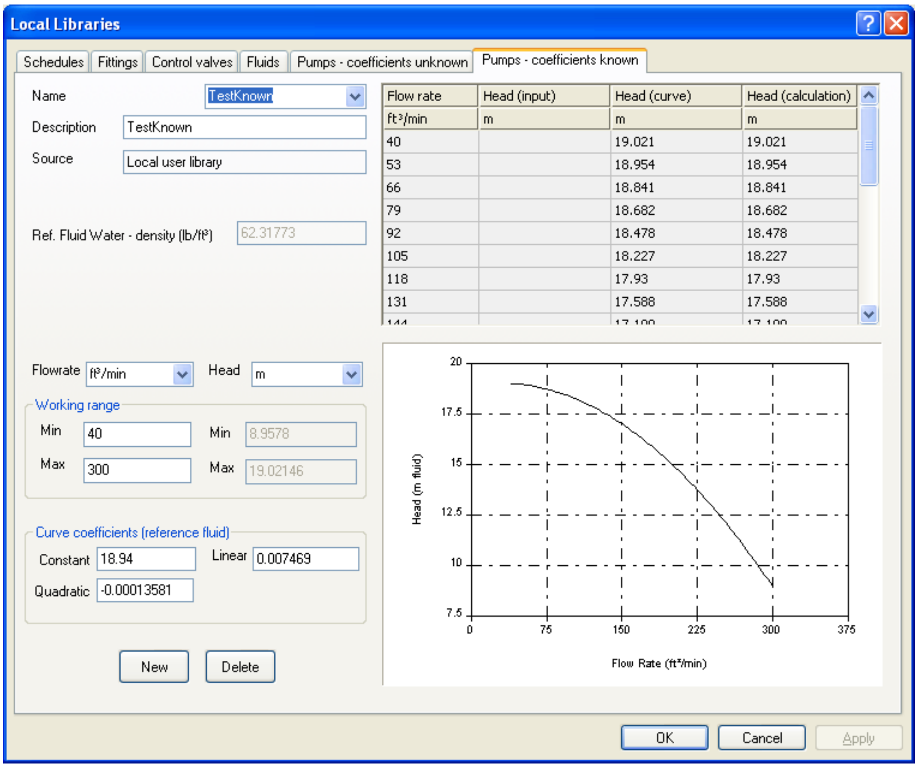
1. Select the pump from the drop-down list presented in the pump name box.
2. Make any changes required to the pump parameters.
3. Select **Apply** to commit the changes.

Deleting a pump

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

17.8 Pumps - Coefficients Known

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



Creating a new pump

To create a new pump:

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

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

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

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

Editing an existing pump

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

Deleting a pump

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

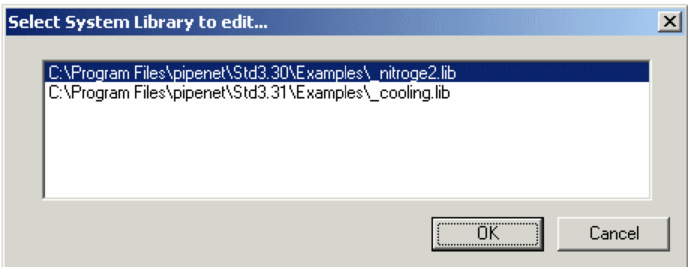
17.9 Editing System Libraries

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

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

Selecting this option will either:

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

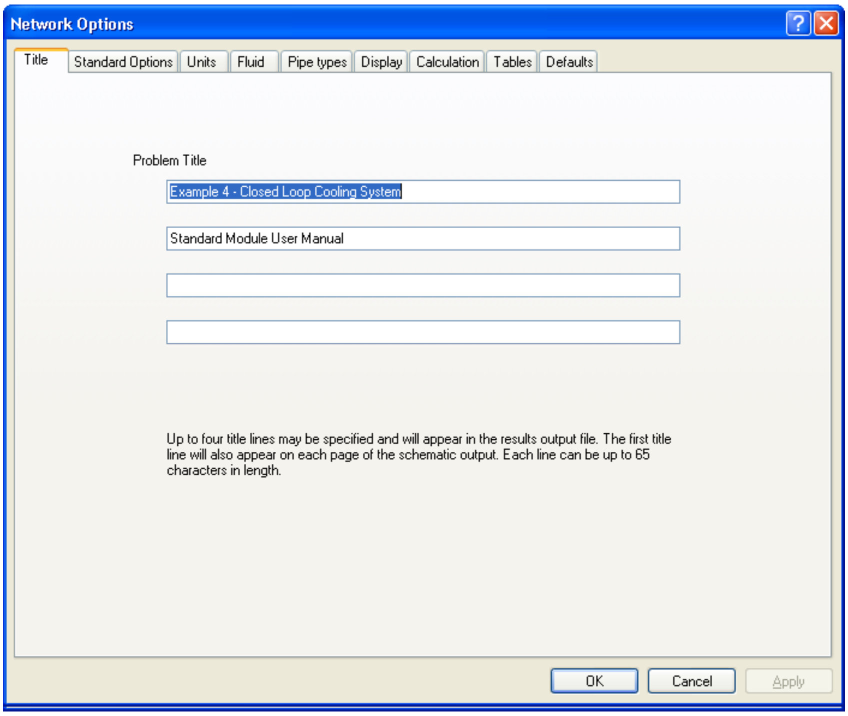


Part 18

18 Specifying options

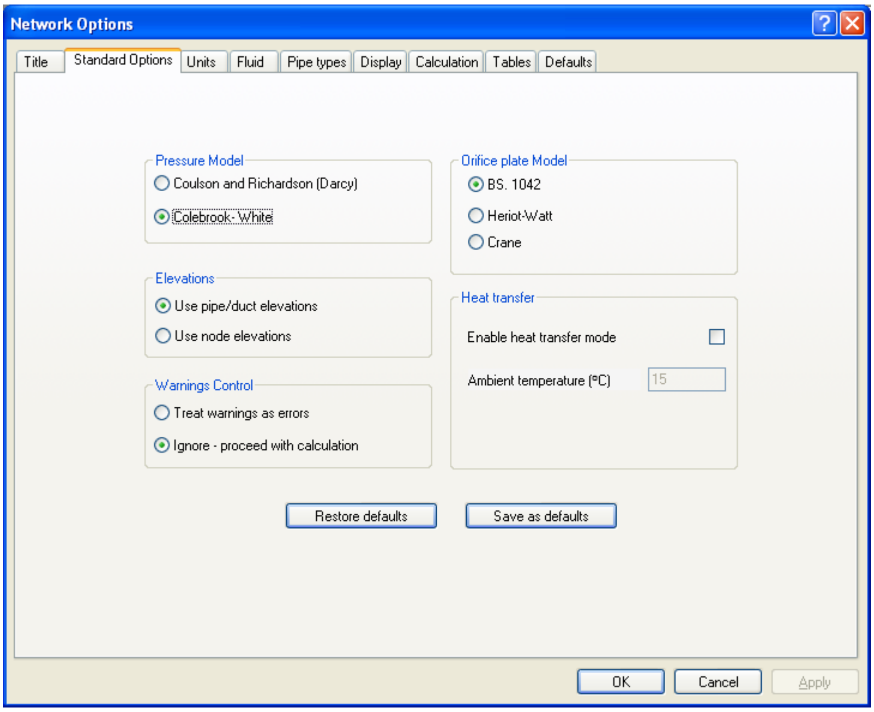
18.1 Title

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



18.2 Module Options

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



Pressure model

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

Elevations

Specify means by which node heights are derived:

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

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

Warnings control

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

Orifice plate model

Select one of the available options:

BS1042
If the BS1042 model is used, the restrictions of that standard are applied so plates may only be used in pipes with diameters in the range 2–14 inches (50.8–355.6mm). Furthermore the ratio of the orifice diameter to the pipe diameter must be in the range 0.1 – 0.748 for larger pipes.

Heriot-Watt
If the "Heriot-Watt Orange Book" orifice plate model is used the ratio of the orifice diameter to the pipe diameter should be less than 0.8 (see also Modelling - Orifice plates).

Crane
Orifice plates are modelled according to [CRANE]

Heat transfer

The heat transfer mode is disabled by default, but may be enabled here if the following are true:-

- The fluid is water
- There are no property (temperature) items present in the network

When enabled, a default ambient temperature of 15° C is assumed, but may be changed.

Once enabled, the heat transfer mode cannot be de-selected if:-

- there are any heat exchanger components present in the network, or -
- if any I/O node has an attached temperature specification.

18.3 Units options

A wide selection of unit systems are provided, including:

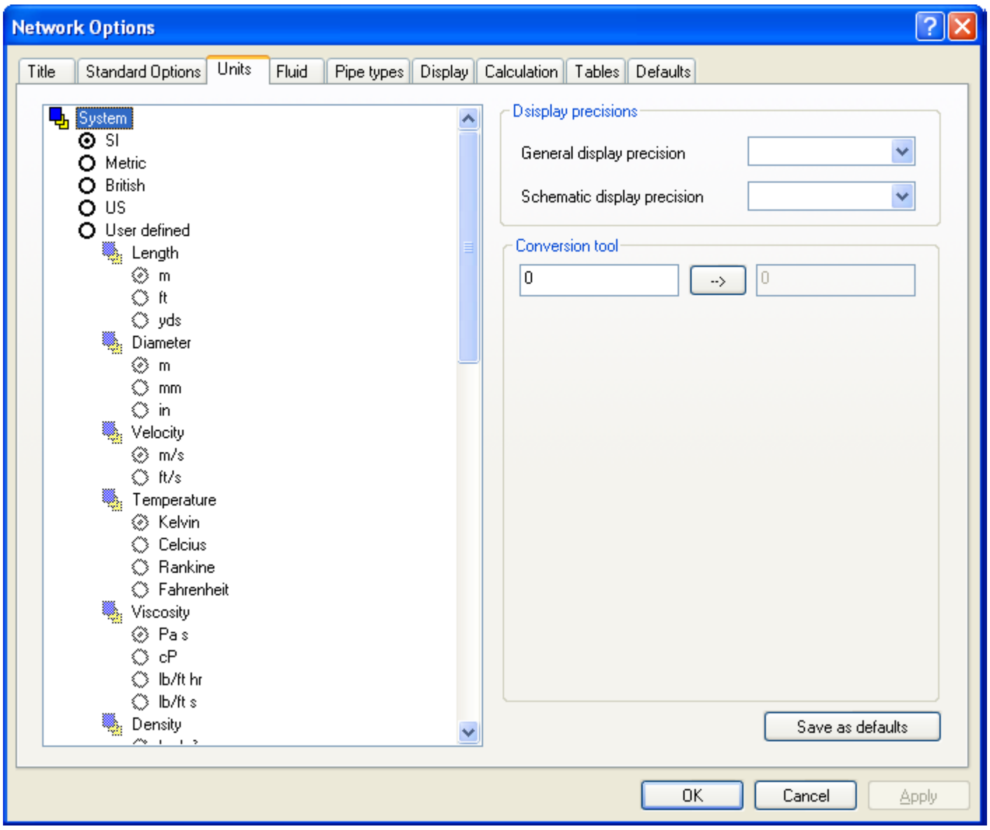
- SI
- Metric
- US
- Imperial
- User Defined

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.

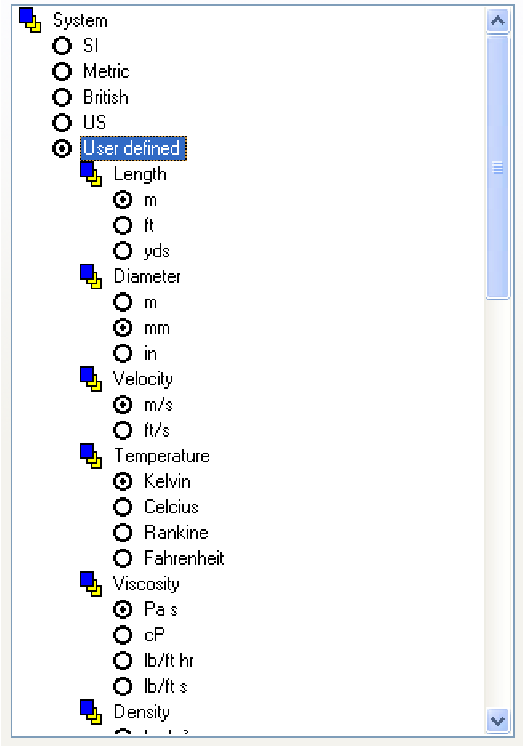
The Units dialog

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:

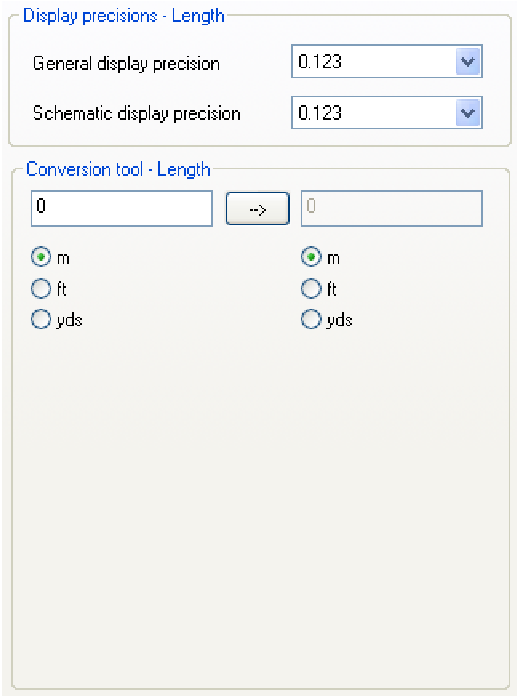


Notes

1. Individual unit options can only be set if the User-defined unit system has been selected.
2. There are three options for flow: Volumetric flow, Mass flow and Standard Volumetric. The Standard Volumetric option can only be used when the fluid is a gas.

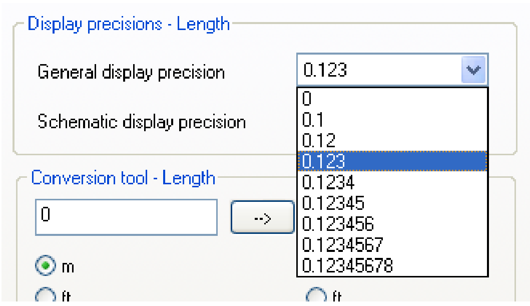
Display Precisions

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



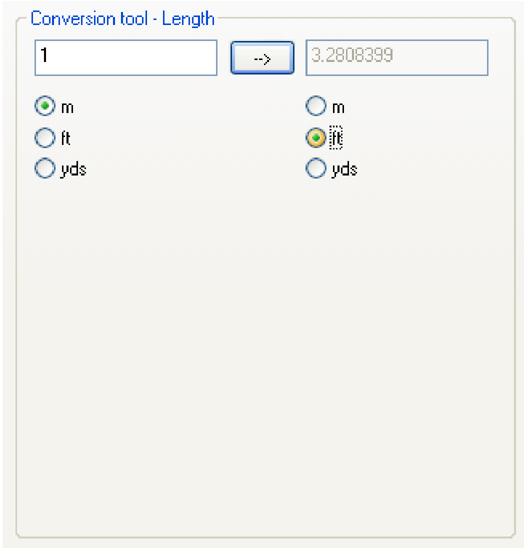
Here **Length** has been selected. The values displayed for the precision **[0.123]** indicate that three decimal places will be used for the display of length in the selected unit of length (metres in this example).

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

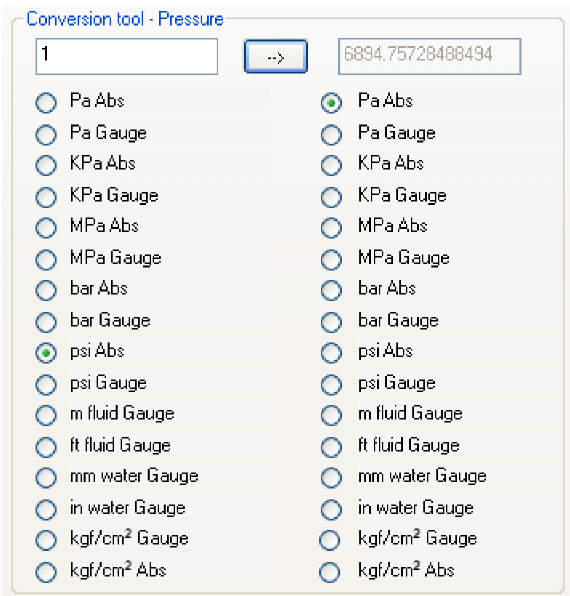


When a network is saved the preferred precisions are saved and they are reloaded when the file is re-opened. Selecting the **Save As Defaults** button will save the current settings in the registry and these will be the defaults used when new networks are created.

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

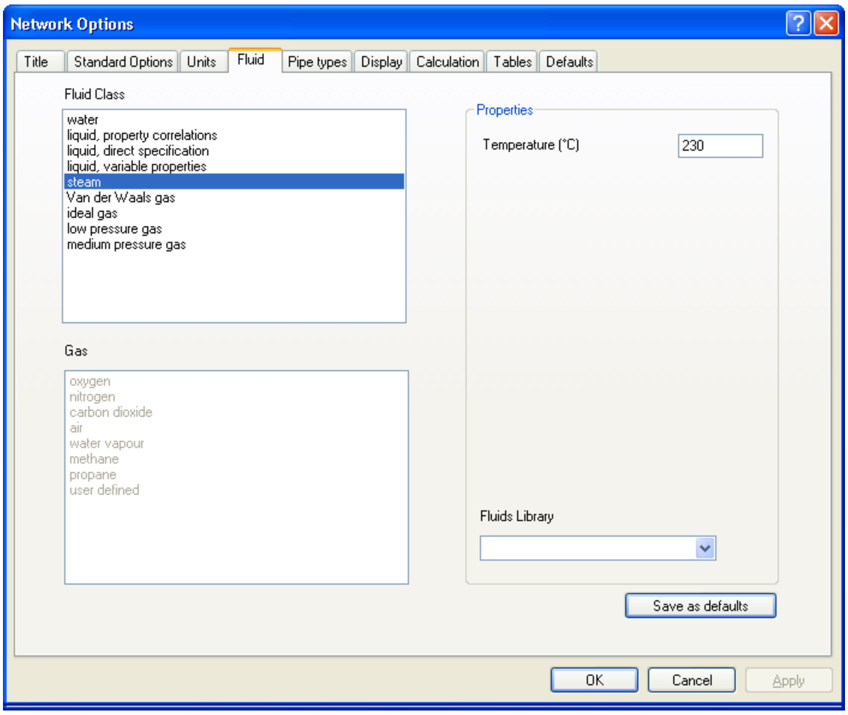


To use the tool simply enter a value in the left-hand field, select the unit that the value represents from the left-hand column (*from* unit) and the unit to which to convert to from the right-hand column (*to* unit) and the result is displayed in the right-hand field. This example illustrates that 1 metre is converted to 3.2808399 ft. The example below illustrates the use of the tool to convert a pressure, specifically 1.0 psi Abs converts to 6894.75... Pa Abs.



18.4 Fluid

The fluid used in the simulation is specified via this options page:



This page consists of three main areas:

Top-left - generic type of fluid, for a description of the parameters required see the properties description below. Note that the Van der Waals equation is an attempt to improve the ideal gas law by including repulsive and attractive molecular interactions and the non-zero volume taken up by the molecules themselves.

Bottom-left - if the fluid type is a gas, then this area is enabled and the specific gas can be selected.

Right-hand - properties for selected fluid, the parameters displayed here will be dependent on the fluid class.

- Water or steam - temperature.
- Liquid, direct specification - density, viscosity and vapour pressure.
- Liquid, property correlations - temperature, critical temperature, A, B, C and M coefficients. For a description of these coefficients refer to the Fluid Specification section of the Modelling chapter
- Liquid, variable properties - at least two sets of density, viscosity and temperature.
- Van der Waal's gas or ideal gas - temperature, critical properties (temperature, pressure and volume), ratio of specific heat capacities (Gamma) and molecular weight.

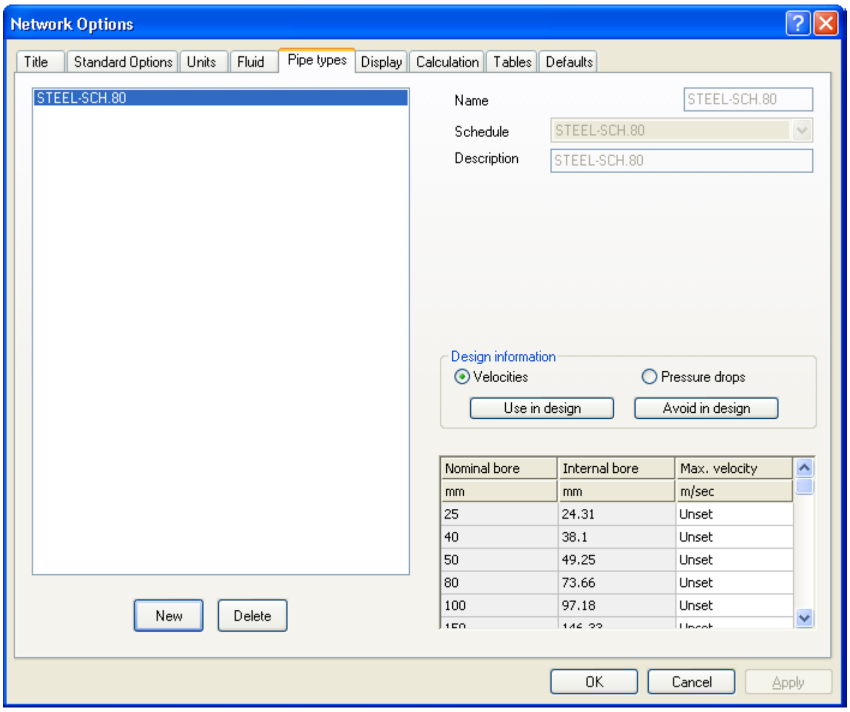
- Low or medium pressure natural gas - gas gravity and correlation equation to be used.

Notes

1. Standard Volumetric units can only be used when the fluid is a gas.
2. If the Heat Transfer mode has been selected, then the fluid will be water and thus the fluid class cannot be changed, neither can the water temperature.

18.5 Pipe Types

This dialog is used to provide a pipe type:



In the standard 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 - name and description provided automatically.
2. Select the associated schedule from the drop-down list provided.
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.

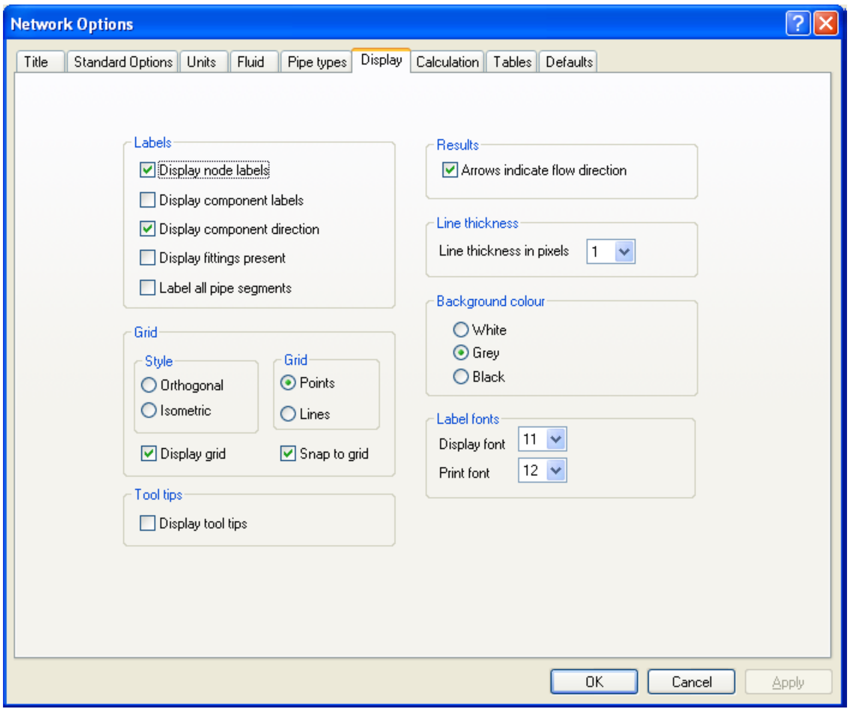
Deleting a pipe type

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 in use, that is one or more pipes reference the type.

18.6 Display options

All schematic related display options are displayed on the **Display** tab. Selecting this option will display a dialog box:

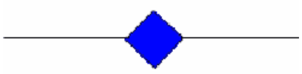


Label Options
The label options allow nodes to be labeled, links to be labeled, component direction to be indicated and the presence of any fittings on a pipe to be shown.

For pipes an additional option is available indicating whether or not all segments of a multi-segment pipe or duct (that is one containing way points) are to be labeled, or only the central segment. Labeling only the central segment may improve the appearance of printed schematics.

An arrowhead pointing from the input towards the output indicates component direction. As with the standard PIPENET definition, component direction does not necessarily correspond to the direction of flow.

The presence of one or more fittings on a pipe is indicated by a blue diamond symbol centred along the length of the pipe.



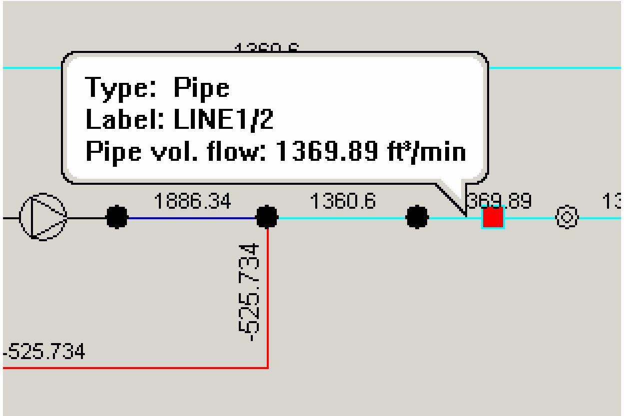
Results Options
Selecting this option allows flow direction to be indicated on all links once a calculation has been performed.

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
These options are used to select the background colour and the font size for node and link labels.

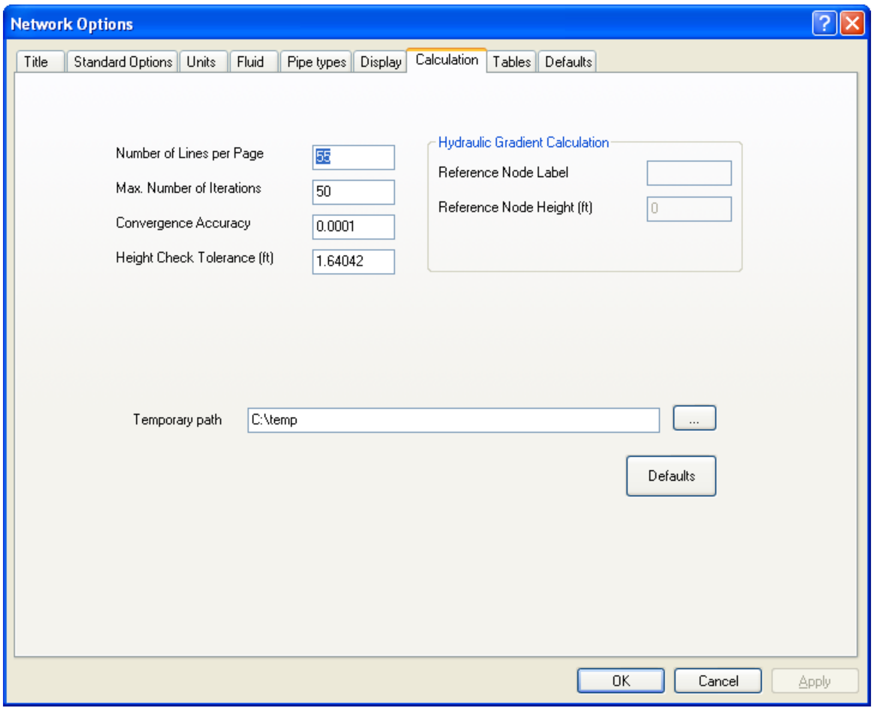
Tool Tips
Selecting this option will cause tool tips to be displayed when the mouse cursor is on or near a component. the tool tip will display the component type, its label and the current parameter selected from the Options toolbar. The information is displayed for a few seconds and then will disappear. For example, moving the mouse to a pipe will display:



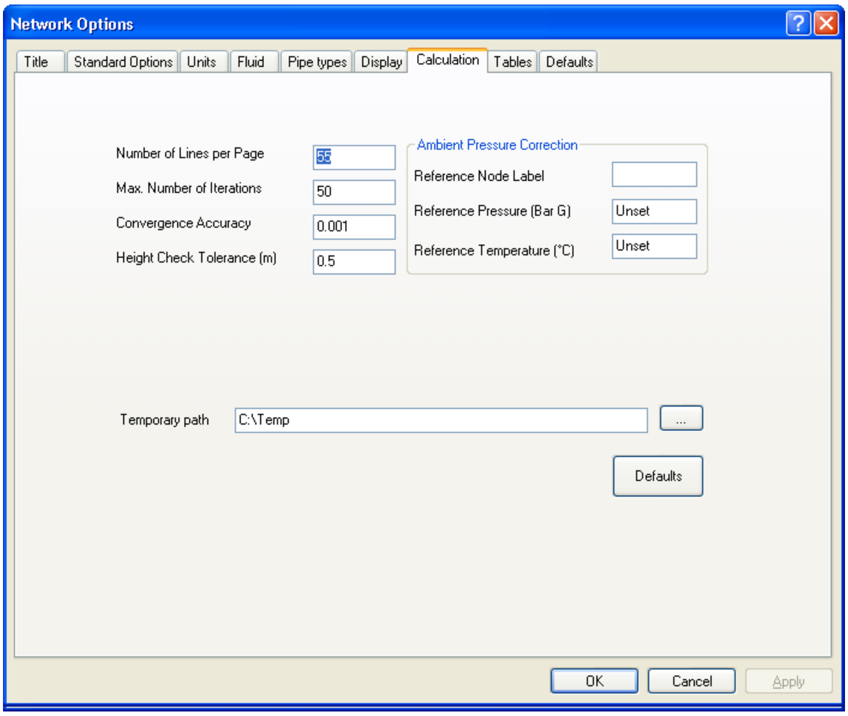
Here the tool tip is displaying the calculated volumetric flow.

18.7 Calculation Options

Calculation related options are displayed on the **Calculation** tab. Selecting this option will display the dialog box (assuming the fluid being used is a liquid):



If the fluid being used is a gas then the dialog will display ambient pressure correction settings instead of hydraulic gradient settings



This page contains a number of calculation related options:

Number of lines per page
This is the number of lines per page used in the traditional output file produced by the calculator.

Maximum number of iterations
The default value of 50 should be sufficient for almost all circumstances. If you see the error message "Failed to Converge" then increasing this value may produce results.

Convergence Accuracy
This parameter defines the convergence accuracy used to determine when two iterations are sufficiently close to assume a solution has been reached.

Height Check Tolerance
If pipe elevations are used this is the acceptable error in resolving node heights in loops.

Temporary Path
Enter the path to be used for the storage of all intermediate temporary files required by the calculator.

Hydraulic Gradient Calculation
This option is only available when the fluid is a liquid and there are no properties present in the network. In order to calculate the hydraulic gradient at every node in the network, PIPENET must be supplied with the absolute elevation of one

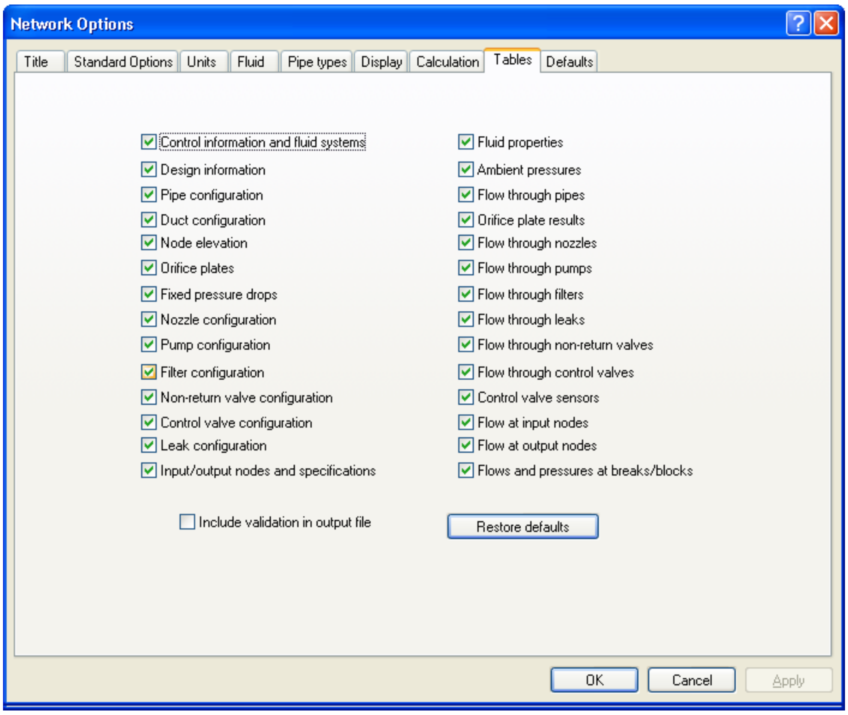
reference node with respect to a datum line.

Ambient Pressure Correction

This option is only available when the fluid is a gas and gauge pressure units are in use. To use the ambient pressure correction facility the user must specify the absolute atmospheric pressure and temperature at a reference node.

18.8 Standard Tables

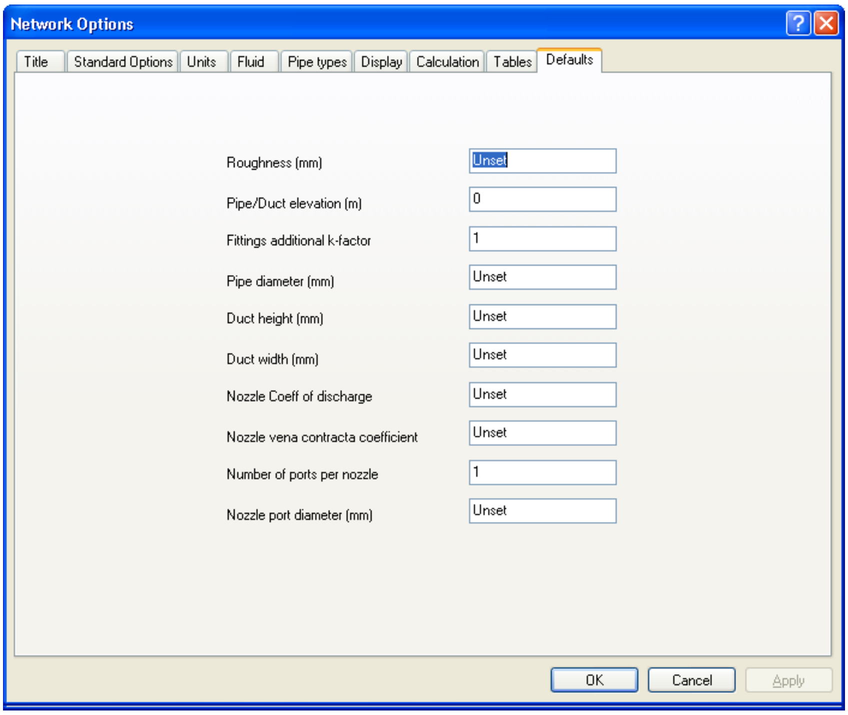
This dialog allows the user to select the tables to appear in the calculation output: for the Standard module:



If the include validation box (lower left-hand corner) is selected, a listing of the original input file will be included at the beginning of the output file. This will display any errors or warnings arising from the input file.

18.9 Defaults

Via this dialog the user can specify the defaults to be applied when creating new pipes, ducts and nozzles:



Part 19

19 Modelling

19.1 Fluid specification

The properties of the fluid being used have a large influence on the way in which the system behaves. PIPENET recognises the following fluid classes:

- **Class 1**: Liquids such as water, brine, crude and fuel oils and most process liquids.
- **Class 2**: Saturated or super-heated steam.
- **Class 3**: Gases modelled using van der Waal's equation.
- **Class 4**: Gases modelled using the Ideal Gas equation.
- **Class 5**: Low and Medium Pressure Natural Gas.

The user can either make use of the fluids which are built into PIPENET or define his own liquid (Class 1) or gas (Class 3 or 4).

The data for a fluid type depends on the fluid class. This can either be liquid (property correlations, direct specification or variable properties), van der Waals gas or ideal gas. For all these cases, a fluid name must be provided.

User-Defined fluids

For a user-defined fluid the density and viscosity must be defined. They may be given either as constants or as varying with temperature.

In the case where density and viscosity vary with temperature, PIPENET uses the correlation formulae:

$$Density = A(B)^{-(1-\frac{T}{T_c})^{\frac{2}{7}}}$$

$$Viscosity = C(T)^{M}$$

where:

T is the temperature (K),
T_c is the critical temperature (K)
A, B, C and M are constants for the fluid.

In order to define the fluid the user must provide values for A, B, C, M, T and T_c.

The density equation is the Rackett equation, which can also be formulated as:

$$V_s = V_c Z_c^{-(1-\frac{T}{T_c})^{\frac{2}{7}}}$$

where:

V_s is the saturated liquid volume at temperature T
V_c is the critical volume
Z_c is the critical compressibility factor

One suggested source of critical properties and the viscosity equation is [POLING].

User-defined gases

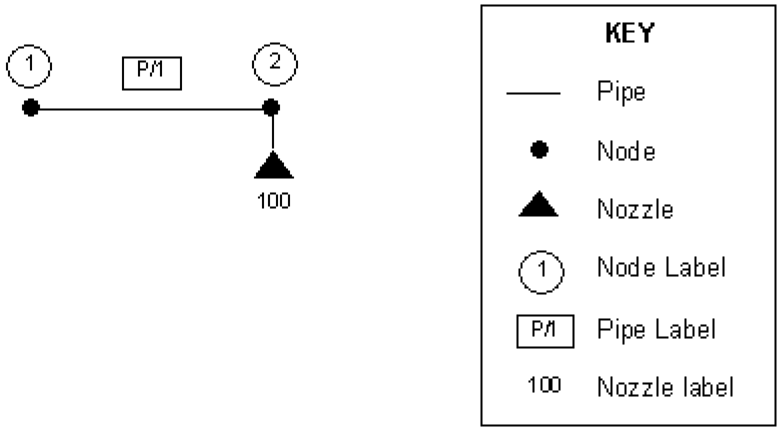
A gas may be defined by the user either as a Van der Waals gas, or as an ideal gas. In either case the user must provide:

- Molecular weight
- Critical properties (temperature, pressure and volume)
- Ratio of specific heat capacities (Gamma).

19.2 Design Facility

PIPENET's Design Facility helps the user to design safe networks, which meet given supply demands whilst ensuring that the fluid velocity in each pipe does not exceed a given maximum value (known as the design velocity of the pipe), or that the pressure drop per unit length of pipe does not exceed a given maximum value (the design pressure drop).

As a simple example of this type of calculation, consider once again the example network:



Suppose that the nozzle is required to supply water at a rate of 600 lit/min, but that for safety reasons the velocity of fluid in the pipe must not exceed 4m/s. (i.e. the design velocity is 4.0 m/s). Then we have:

Flow rate through pipe = Fluid velocity × Pipe cross-sectional area

and

Fluid velocity $\leq$ Pipe Design velocity

Using 600 lit/min = 10 lit/s = 0.01 m^3/s:

$0.01 \leq 4.0 \times (0.25 \times \pi \times d^2)$

or

$d^2 \geq 0.01/\pi = 0.003183$
$d \geq 0.056411$ m = 56.4 mm.

Thus in this case the diameter of the pipe should be larger than 56.4 mm. The actual diameter of the pipe used will depend on what sizes of pipe are available, and this is determined by the pipe schedule being used.

Given the flow rates required in and out of the network, PIPENET will calculate the flow rates through every pipe in the network. PIPENET will then perform a calculation similar to the one given above, and select a suitable diameter for each pipe.

19.3 Ambient pressure correction

While PIPENET performs its calculations using absolute pressure units of Pascals, it allows use of a variety of units for pressure including gauge units such as Bar G and psi G. Normally the program converts pressures in gauge units to absolute units by adding a constant value of one standard atmosphere (101325 Pa).
In networks with large elevation changes, users may wish to take account of the variation in atmospheric pressure with height when converting from gauge to absolute pressures. Atmospheric pressure Pi, at a height hi above a datum point is given by the equation:

$$\ln \frac{P_0}{P_i} = \frac{\rho h_i g}{P_0}$$

where:

P_0 is the absolute atmospheric pressure at the datum point.
g is the acceleration due to gravity.
ρ is the fluid density.

To use this correction facility the user must simply supply the absolute atmospheric pressure at a given reference node.

Ambient pressure correction is permitted by PIPENET only when gauge pressure units are being used and the fluid in the network is a gas.

19.4 Pipe Modelling

Modelling Equation

Pipes are assumed to be of uniform circular cross-section.

$$P = P_{fric} + P_{elev} + P_{plat}$$

where:

P_{fric} = Pressure loss due to friction and fittings.
P_{elev} = Pressure loss due to elevation change.
P_{plat} = Pressure loss due to any orifice plate fitted.

Friction losses - Coulson & Richardson (Darcy)

P_{fric} is found using the Bernoulli equation method. The Bernoulli equation is a theoretical equation which gives the pressure in pipes, ignoring frictional effects. By comparing the theoretical results obtained using the Bernoulli 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:

$$P_{fric} = \frac{2 f L \rho u^2}{D}$$

where:

D is the internal diameter of the pipe.
L is the pipe length.
f is the Fanning friction factor.
u is the fluid velocity.
ρ is the fluid density.

The Fanning friction factor ***f*** depends on Reynold's number (Re = uDρ/μ where μ 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):
$f = 16/Re$

Transitional flow (2000 < Re < 3000):
f is found by interpolating between laminar value for Re = 2000 and turbulent value at Re = 3000.

Turbulent flow (Re > 3000):
$$1/\sqrt{f} = -1.768 \ln (0.27r/D + 1.252/Re\sqrt{f})$$

Friction losses - Colebrook-White method

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

19.5 Heat Transfer and Heat Exchangers

Heat transfer

At I/O nodes, temperature specifications give the temperature for the flow into the network. If the temperature specification is not given, then the ambient temperature, as specified in the Standard options dialog, will be used. Temperature change is not considered for the components other than heat exchangers.

Density and specific heat capacity in the heat transfer calculation are calculated using [IAPWS].

Please note:

1. If a temperature specification is given to a node, but the flow is coming out from the system at this node, then the temperature result of this node is decided by the upstream temperature rather than the given temperature specification.
2. If the heat transfer rate is unreasonably high or the flow rate is unreasonably low, the calculation may fail.

Heat exchangers

Heat exchangers can only be added when the heat transfer mode is selected in the Standard options dialog.

User Input Parameters

E heat transfer rate
Q_{ref} reference flow rate
ΔP_{ref} reference pressure drop

Modelling Equations

Heat transfer equation

$$E = C Q_m (T_2 - T_1)$$

where

- Q_m mass flow rate
- c average fluid specific heat capacity in the heat exchanger
- T_1 inlet temperature
- T_2 outlet temperature

Resistance equation

$$P_1 - P_2 = K Q_m |Q_m|$$

where

- P_1 inlet pressure
- P_2 outlet pressure
- K resistance factor

Resistance factor equation

If the mass flow rate unit is selected

$$K = \frac{\Delta P_{ref}}{Q_{ref}^2}$$

If the volumetric flow rate unit is selected

$$K = \frac{\Delta P_{ref}}{Q_{ref}^2 \rho^2}$$

where

- ρ average fluid density in the heat exchanger

Inlet flow rate equation

If a mass flow rate unit is selected

$$Q_1 = Q_m$$

If a volumetric flow rate unit is selected

$$Q_1 = \frac{Q_m}{\rho_1}$$

where

Q_1 inlet flow rate

ρ_1 fluid density in the inlet of the heat exchanger

Outlet flow rate equation

If a mass flow rate unit is selected

$$Q_2 = Q_m$$

If a volumetric flow rate unit is selected

$$Q_2 = \frac{Q_m}{\rho_2}$$

where

$Q2$ outlet flow rate

ρ_2 fluid density in the outlet of the heat exchanger

The average fluid specific heat capacity and the average fluid density are calculated based on the arithmetic mean temperature and the arithmetic mean pressure of the heat exchanger.

19.6 Ducts

Ducts are very similar to pipes except for the obvious difference that ducts have a rectangular cross-section.

Modelling Equation

Ducts are modelled using the same equations as pipes. In order to do this PIPENET calculates a *mean hydraulic diameter*, D_H, for the duct using:

$$D_H = \frac{2HW}{(H+W)}$$

where:

H is the duct height and
W is the duct width.

Data Required
Supplied in the data file:

- Height and Width.
- Duct length.
- Increase in elevation from inlet to outlet.
- A list of fittings on the duct.

Notes
1. Ducts cannot be used with PIPENET's Design Facility.
2. Ducts can only be used when the fluid is a gas.
3. Increase in elevation may be given directly or by supplying the elevations of the input and output nodes.

19.7 Pumps

A pump provides a pressure increase which depends on the pump speed and performance curve. The pump performance curve is entered as a library.

User Input Parameters
Pump Type - selected from a list of defined library pumps (with either unknown coefficients or known coefficients).

Quadratic Modelling Equations

The pressure increase produced by the pump is:

$$P = AQ^2 + BsQ + Cs^2$$

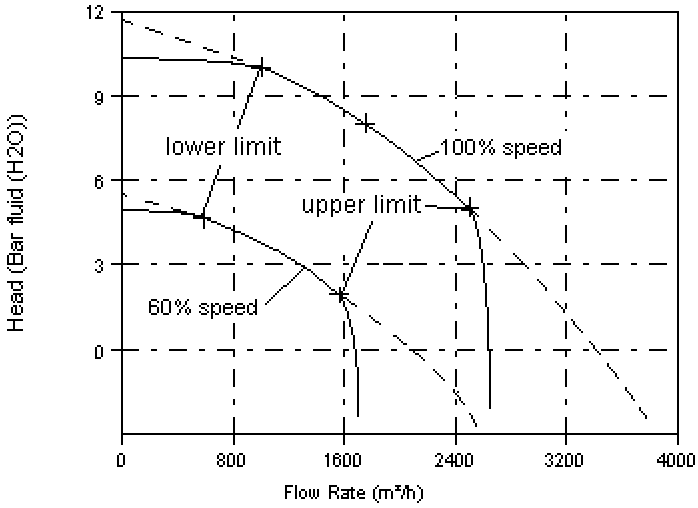
where:

s is the pump setting.
Q is the flow rate through the pump.
A, ***B***, ***C*** are pump performance coefficients.

The pump performance coefficients A, B and C are entered directly, or calculated by PIPENET for a specified pump curve, in pump libraries. The user also specifies an operating range for flow rates through the pump. It is recommended that the zero flow rate point is given when defining a pump curve.

The pump setting, **s**, must be between 0.0 and 1.0. A setting **s** = 0.0 represents a shut down pump, while **s** = 1.0 represents a pump at full speed.

The quadratic curve will, in general only apply to the specified working range of the pump, however degeneration factors can be used to specify the behaviour of the curve below the specified working range and/or above the specified working range.



Within the work region – Q_{LL} to Q_{UL}

PIPENET calculates a binomial correlation to fit the input data, see the solid line in the above figure:

$$P = AQ^2 + BsQ + Cs^2$$

Below the minimum flow rate Q_{LL}

We introduce a degeneration factor **n**, which is an integer 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:

$$0 \leqslant n \leqslant 10$$
$$\alpha = 2\mathrm{AQ}_{LL} + Bs$$
$$\beta = AQ_{LL}^2 + BsQ_{LL} + Cs^2$$
$$Q_{LL} = Q_{min} s$$

Above the maximum flow rate Q_{UL}

We introduce a degeneration factor **m**, which is an integer in the range 0 to 10. The larger the value of m the quicker of the pump curve degenerates..

$$\Delta P = AQ^2 + BsQ + Cs^2 - (2^m - 1)|A|(Q - Q_{UL})^2$$

where:

$$0 \leqslant m \leqslant 10$$
$$Q_{UL} = Q_{max}\, s$$

Notes

If the performance coefficients for a pump are known, then the pump can be defined as a non-library pump by giving the values of ***A***, ***B***, ***C***, Q_{min} and Q_{max}.

In order for the calculator to function correctly, it is necessary to ensure that there is only one flow rate corresponding to each pressure gain, and so the following restrictions are applied:

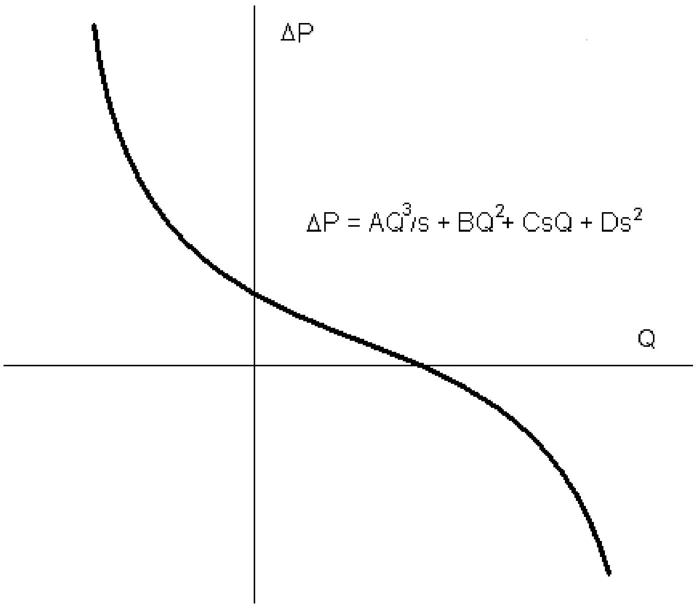
- For flow rates between Q_{min} and Q_{max}, the slope of the performance curve must be negative or zero.
- For flow rates between Q_{min} and Q_{max}, there must be no flow rate that gives no pressure change. (That is, the performance curve must not cross the horizontal axis.)

For flow rates outside the range Q_{min} to Q_{max}, PIPENET extrapolates the performance curve using the tangent to the curve at the minimum or maximum flow rate, and issues an appropriate warning message.

PIPENET will calculate the power required by a pump based on specified efficiency. The power calculation will assume that the pump is 100% efficient if the efficiency is not specified

Cubic Modelling Equations

A simple quadratic curve can deviate considerably from the supplied pump data, especially when the flow rate exceeds the pump capacity, or is less than the allowed minimum flow rate. The cubic curve can provide better results.

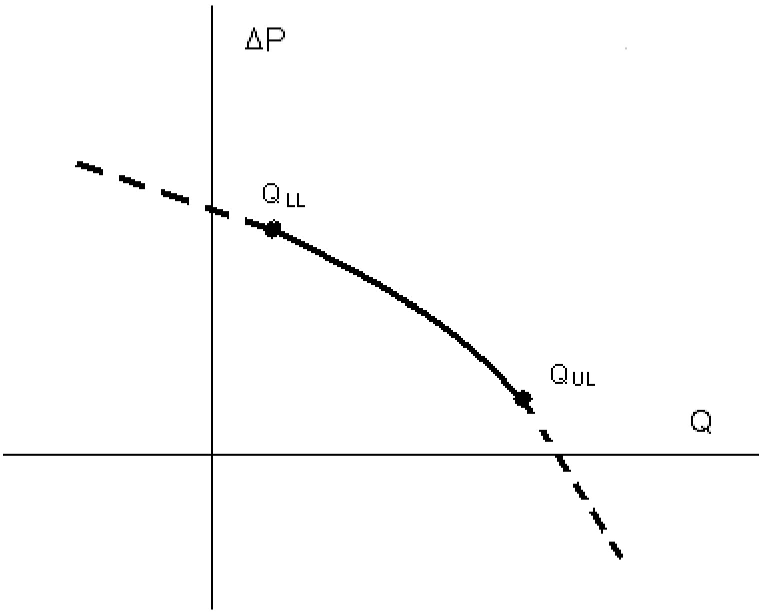


There is a potential problem with the cubic curve in that it may lead to multiple solutions. Therefore, the above function must satisfy the following condition:

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

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 $[Q_i, Q_{i+1}]$

$$\Delta P = A_i Q^3 / s + B_i Q^2 + C_i sQ + D_i s^2$$

Below the lower limit, with $\boldsymbol{Q} < \boldsymbol{Q_{LL}}$ and along the tangent direction at the point [$\boldsymbol{Q_{LL}}$, $\Delta \boldsymbol{P_{LL}}$]

$$\Delta P = \alpha (Q - Q_{LL}) + \beta$$

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 $\boldsymbol{Q} > \boldsymbol{Q_{UL}}$ and along the tangent direction at the point [$\boldsymbol{Q_{UL}}$, $\Delta \boldsymbol{P_{UL}}$]

$$\Delta P = \alpha (Q - Q_{UL}) + \beta$$

where:

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

NPSH

Net Positive Suction Head (NPSH) is an analysis of the energy conditions on the suction side of the pump to determine if the liquid will vaporize at the lowest pressure point in the pump. NPSH is measured in the units of length.

There are two components that must be considered for NPSH:

- **NPSHR** is defined as Net Positive Suction Head Required. NPSHR is a function of the design of the pump, and is determined by the pump manufacturers via testing.
- **NPSHA** is defined as Net Positive Suction Head Available, which can be calculated as follows:

$$NPSHA = (P_i - P_{vap}) / \rho g$$
$$= \frac{P_{atm}}{\rho g} - H_S - \frac{P_{vap}}{\rho g}$$

Where,

P_i is the (absolute static) pressure at pump inlet;

P_{vap} is the vapour pressure;

ρ is the liquid density;

g is the gravitational acceleration;

P_{atm} is the atmospheric pressure;

H_S is the suction head which includes not only the suction lift but also the sum of the losses in the inlet pipe and the velocity head.

If NPSHA < NPSHR, then **cavitation** will happen.

The **cavitation parameter** is a dimensionless ratio used to relate the conditions which inhibit cavitation to the conditions which cause cavitation. There are several common forms of the cavitation parameter. In PIPENET, the cavitation parameter uses the definition:

cavitation parameter = ***NPSHA*** / ***H***

where ***H*** is the total head of the pump.

NPSHA and the cavitation parameter are only available in the output report for pumps when water or a directly specified liquid is being used.

19.8 Non-return valve

Non-return valves allow unrestricted flow of fluid in a positive direction, and prevent all flow in a reverse direction. ***Positive flow*** is taken to mean from the valve's input node to its output node, in which case there will be no pressure drop across the component. Caution should be exercised not to position a non-return valve such that it would isolate a portion of the network. If this was to happen, the calculator could report an error: " network cannot be solved"

Note that even if the solution to the problem has the valve open, PIPENET can still generate this message. Occasionally, a non-zero "leakage flow" may be reported through a closed valve. This arises from rounding errors in the calculation which are smaller than the requested convergence accuracy, and is therefore usually negligible.

User Input Parameters

None

Modelling Equations

Open valve
$P_1 = P_2$

Fully closed valve
$Q_1 = 0$

Continuity Equation
$Q_1 = Q_2$

where:

P_1 is the inlet pressure.
P_2 is the outlet pressure.
Q_1 is the inlet flow rate.
Q_2 is the outlet flow rate.

19.9 Control Valves

The valve may be characterised by one of three built-in models which require either a K-factor and a port area, or a flow coefficient, or by a control valve type. Select the appropriate choice from the Valve type combo-box and radio buttons and enter the required data, if any, in the valve characteristics boxes below. Note that the flow coefficient is that for water at 20ºC.

Modelling Equation

$$P = \frac{\rho Q^2}{\rho_0 C_v^2(s)}$$

Or

$$P = \frac{K \rho Q^2}{2 s^2 A^2}$$

where:

P is the pressure drop across the valve.
Q is the (volumetric) flow rate through the valve.
ρ is the fluid density.
ρ_0 is the density of water at 20ºC.

s is the valve setting, $0 < s < 1$.
K is the K-factor for the valve.
A is the cross-sectional area of the valve port.
C_v (s) is the valve flow coefficient for water at 20ºC.

Data Required

In the input:

Either: A constant K-factor and cross-sectional area - ***k-factor valve.***
or: A constant flow coefficient ($C_v(s) = C_v.s$) - ***flow coefficient valve.***
or: A control valve type.

Either: Sensor details.
or: A setting, **s**, entered as a percentage; 0% is fully closed and 100% fully open.

In the valve library:

Either: A cross-sectional area and at least two points from the K(s) curve, and the gradients at those points.
or: At least two points from the $C_v(s)$ curve, and the gradients at those points.

The gradients at points on a K(s) curve must be negative. Those on a $C_v(s)$ curve must be positive.

Control Valve Considerations

A control valve regulates flow or pressure in a network. The pressure drop across the control valve is dependent on the valve setting, **s**, and its physical characteristics.

The valve setting can either be specified directly by the User, or be determined by PIPENET such that a particular sensor reading is satisfied. Three sensor types are available:

- Pressure at a node, P_n.
- Flowrate Q through a particular pipe.
- Pressure Difference between two nodes, $P_D = P_m - P_n$

PIPENET calculates a control valve's setting such that the associated sensor reading is attained. In some scenarios this is not possible. For example, a control valve monitoring flowrate in an adjacent pipe may be unable to achieve the sensor reading even with a fully open setting. In such cases the valve setting will be fully open or closed, whichever gives the closest result for the sensor reading.

Some care is required when using control valves that use a sensor to calculate the valve position. A fully closed control valve behaves like a closed non-return valve and can isolate parts of the network resulting in an unsolvable system. In particular, this can sometimes occur if blocked pipes and control valves are both present in a scenario.

Three built in control valve characteristics are available: Linear, Equal Percentage or Quick opening. Alternatively a library control valve type can be created by specifying the valve characteristic curve of k-factor or flow coefficient against **s**. PIPENET then uses cubic interpolation to determine intermediate points on the characteristic curve.

The linear control valve option provides the same model as used in earlier versions of PIPENET, which did not offer equal percentage or quick-opening valves.

19.10 ElastomericValve

The Elastomeric valve enables the user to achieve the required input pressure, output pressure, pressure drop or flow rate without the need to input the valve characteristic data. The elastomeric valve is representative of a valve type which fulfils the above roles; however, the model described below can be used to model any valve, including elastomeric valves, with the specified characteristics.

Note:
i) The use of unreasonable target values might make the network unsolvable.
ii) A warning message will be given in the calculation report when the output pressure is higher than the input pressure or the flow rate is negative.

User input parameters

- Type - Input pressure, Output pressure, Pressure drop or Flow rate
- Target value

Modelling equations

Design phase

$$P_1 = P_2$$

Analysis phase

Input pressure type - aim to control the input pressure of the valve to be the given target value:

$$P_1 = P_1^*$$

Output pressure type - aim to control the output pressure of the valve to be the given target value:

$$P_2 = P_2^*$$

Pressure drop type - aim to control the pressure drop of the valve to be the given target value:

$$P_1 - P_2 = \Delta P^*$$

Flow rate type - aim to control the flow rate of the valve to be the given target value:

$$Q = Q^*$$

where:-

P_1	- input pressure
P_2	- output pressure
P_1^*	- target input pressure
P_2^*	- target output pressure
Q	- flow rate
Q^*	- target flow rate

U

19.11 Filters

Modelling Equation

$$P = AQ|Q| + BQ$$

where:

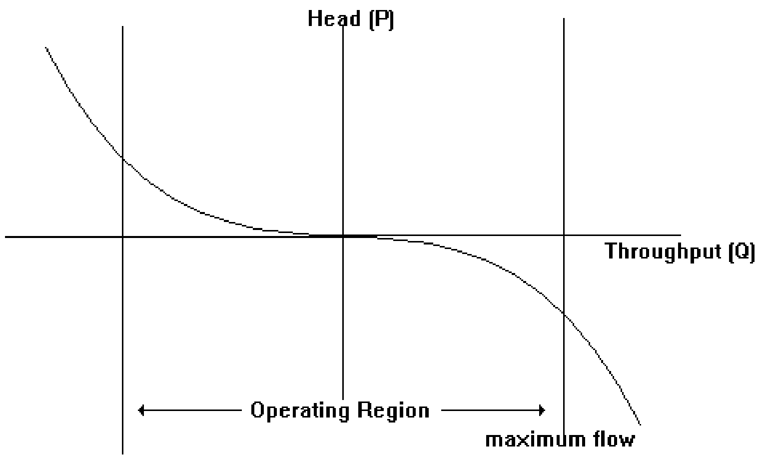
P is the pressure increase from inlet to outlet.
Q is the (volumetric) flow rate through the filter.
A is a coefficient less than or equal to zero.
B is a coefficient less than zero.

This is operative for values of Q whose modulus is less than a given maximum flow Q_{max}. Note that filters are reversible (Q may be negative), and that as

$$A \leq 0 \text{ and } B < 0$$

the pressure drops in the direction of the flow.

Typical Performance Curve



19.12 Nozzles

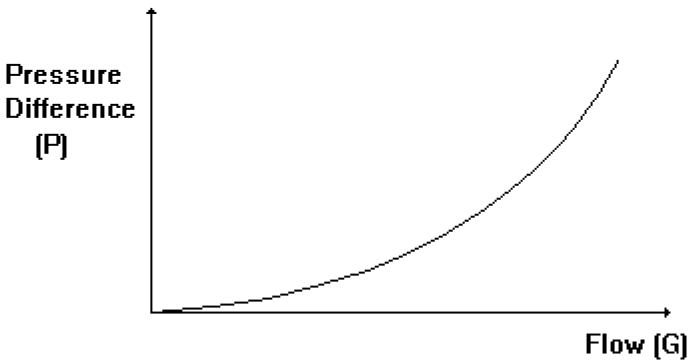
Modelling Equation

$$P = \frac{G^2}{2(C_v C_d A)^2 \rho}$$

where:

$\boldsymbol{A}$ is the cross-sectional area of a port.
$\boldsymbol{G}$ is mass flow rate through a single port.
$\boldsymbol{C_d}$ is the coefficient of discharge (0.0 – 1.0).
$\boldsymbol{C_v}$ s the coefficient of vena contracta (0.0 – 1.0)..
ρ is the fluid density.

Typical Performance Curve



Data Required

- The coefficient of discharge.
- The vena-contracta coefficient - this is the ratio between the area of the jet at the vena-contracta to the area of the orifice
- The number of ports.

- The port diameter.

Notes

Nozzles may only be used when the fluid is a liquid.

19.13 Leaks

This models a leak in a pipe and leaks may only be used when the fluid type is a gas.

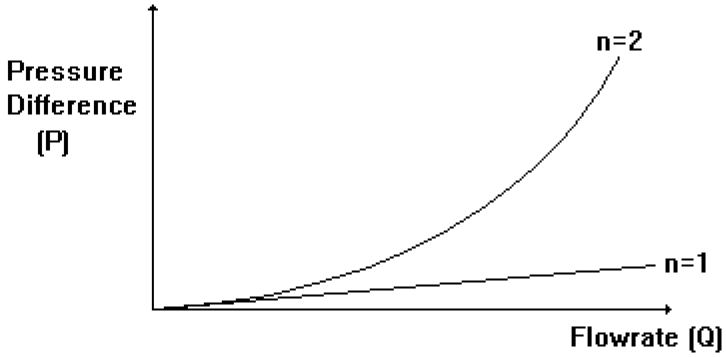
Modelling Equation

$$P = (Q/kA)^n$$

where:

P is the pressure drop across the leak.
Q is the (volumetric) flow rate through the leak.
A is the area of the leak.
n is an index in the range 1–2. (the Power Factor).
k is a units dependent constant (0.827 for SI units).

Typical Performance Curve



Data Required

- Area of Leak.
- Power factor of Leak.

19.14 Properties

PIPENET allows the properties of the fluid to be constant or to vary in the system. The default constant fluid properties are set up using the menu option **Options | Fluid Type**. If the fluid properties are to vary then this can be done in one of two ways:

By allowing the density and viscosity to vary with temperature

This method can only be used if the default fluid class is not liquid, direct specification. See Options | Fluid Type for more details on choosing a fluid class. This allows density and viscosity of the fluid in a given pipe depend on its temperature. Hence, the present menu option allows the user to set the temperature of the fluid in a pipe.

By specifying the density and viscosity directly
This method can only be used if the default fluid class is liquid, direct specification. See Options | Fluid Type for more details on choosing a fluid class. The present menu option will allow the user to input the value of the fluid density and viscosity for a pipe where the default properties are not to be used. This method is particularly useful for simulating the mixing of liquids within the network.

Notes
If a pipe does not have fluid properties defined using this command then the program uses the default fluid properties determined by the menu option Options | Fluid Type.

19.15 Orifice Plates

Orifice plates may be inserted into any pipes in the network and produce an additional pressure drop across the pipe. In order to define an orifice plate the user must specify:

1. A label for the plate.
2. The label of the pipe to which the plate is attached.
3. Either the diameter of the orifice plate or the pressure drop required across the plate.

Modelling - Pressure drop due to an Orifice Plate
Orifice plates may be inserted in any pipe, and are modelled using one of three models as selected by the user:

- Plates with flange tappings in accordance with BS1042, taking into account pressure recovery downstream. The restrictions of BS1042 are applied so plates may only be used in pipes with diameters in the range 2 – 14 inches (50.8 – 355.6 mm). Furthermore the ratio of the orifice diameter to the pipe diameter must be in the range 0.1 – 0.748 for pipes over 4 inches in diameter.

- The "Heriot-Watt Orange Book" orifice plate model [CRANE]. This model derives from curves for laminar flow given by Miller in "Internal Flow Systems" and for turbulent flow given by the ESDU in "Flow of liquids. Pressure losses across orifice plates, perforated plates and thick orifice plates in ducts." The ratio of the orifice diameter to the pipe diameter should be less than 0.8.

- Sharp-edged orifice plates as described in Crane T.P. 410M. The ratio of the orifice diameter to the pipe diameter should be in the range 0.2 – 0.75. In addition, the Reynolds number of flow in the pipe should be greater than 100, as the accuracy of the model decreases at lower values.

The pressure drop due to an Orifice Plate may be found in one of two ways: either it may be specified directly by the user (in which case PIPENET will calculate the orifice diameter necessary to produce this pressure drop), or it can be calculated by PIPENET from the orifice diameter given by the user. Users should exercise caution when using this facility as the calculation of pressure drop from plate diameter is unstable in that a small change in orifice diameter may result in a very large change in the pressure drop.

19.16 Fixed pressure drops

A single fixed Pressure Drop may be added to any pipe in the network not containing an orifice plate, to produce an additional pressure drop across the pipe which is independent of flow through the pipe. In order to define a fixed pressure drop, the user must specify the constant pressure drop.

19.17 Fittings

Tees and other fittings must be added manually. In principle PIPENET could attempt to deduce the placement of fittings from the network topology, but in practice it is far better to give the user complete control over this process, which is very difficult to automate in any useful way since it is often a matter of engineering judgment and different users will have different needs.

Each fitting, both built in and user defined, is modelled as an additional k-factor to be added to a pipe, the k-factor being calculated according to the type of fitting and the diameter of the pipe to which it is attached, K-factors are calculated according to the rules in [CRANE].

Device type fittings

All k-factors, for both built-in and user-defined fittings are reliably derived during a calculation and correctly displayed in the browser output. The user-interface also displays the calculated k-factors in the Properties window (Fittings tab) for user reference. However, if the fluid density is unknown prior to the calculation, as will happen of the fluid is one of the following:-

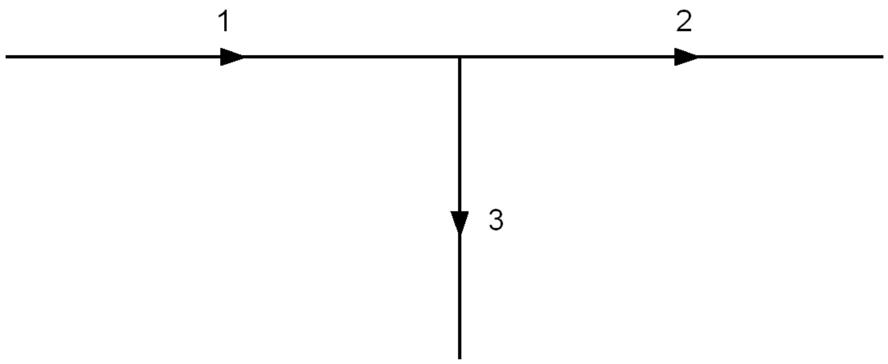
- Liquid - variable properties
- Steam
- Low pressure gas
- Medium pressure gas

then the K-factor for a user-defined **Device** type fitting cannot be displayed since it depends on the fluid density. In these cases the K-factor for Device fittings and the total K-factor will be displayed as **n/a** (not available).

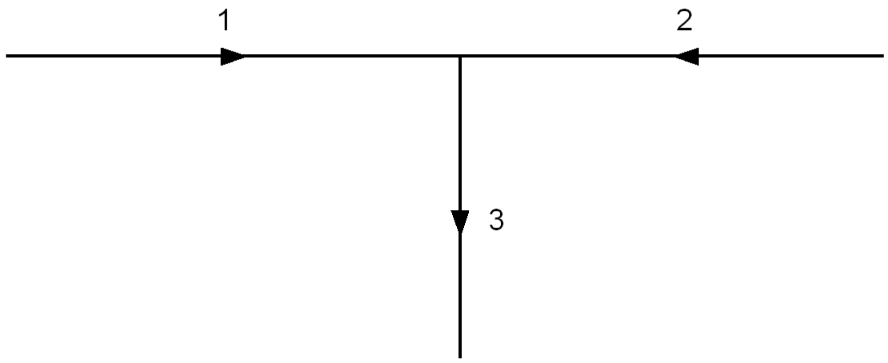
Tees

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.

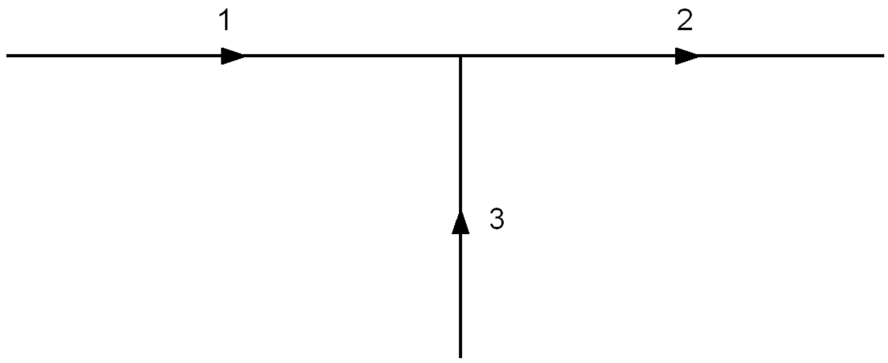
Example 1: tee(run) on pipe 2; tee(branch) on pipe 3.



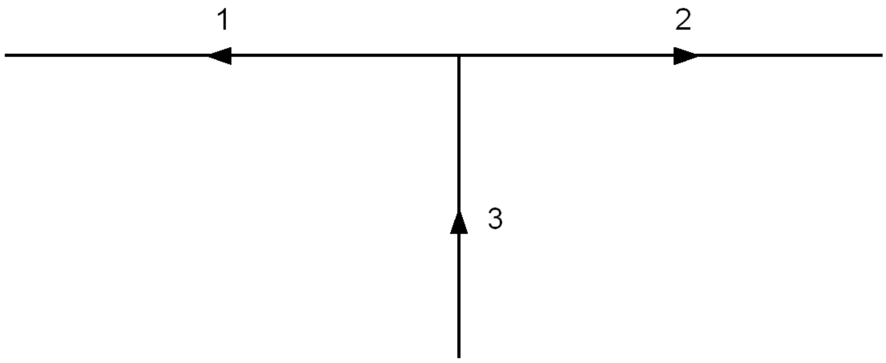
Example 2: tee(branch) on pipe 1; tee(branch) on pipe 2.



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



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


1
2
3

Exporting the Schematic

Part 20

AIR CONDITIONING COMPANY

20 Exporting the Schematic

20.1 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].

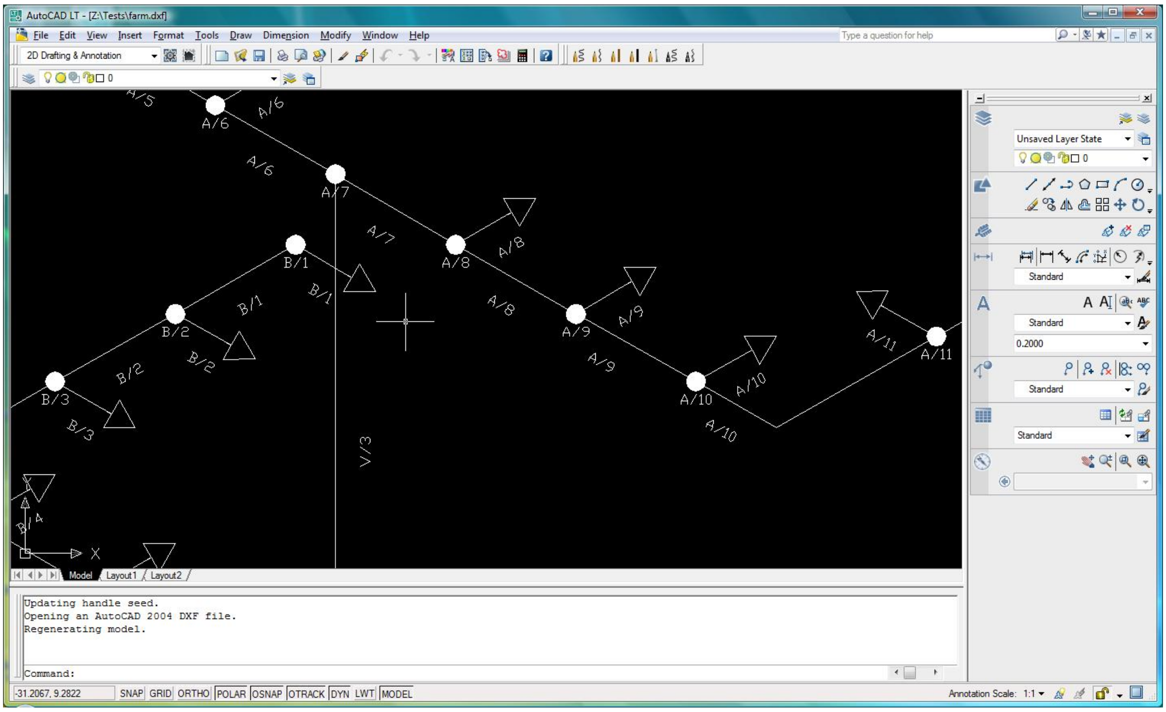
20.2 DXF Ouput

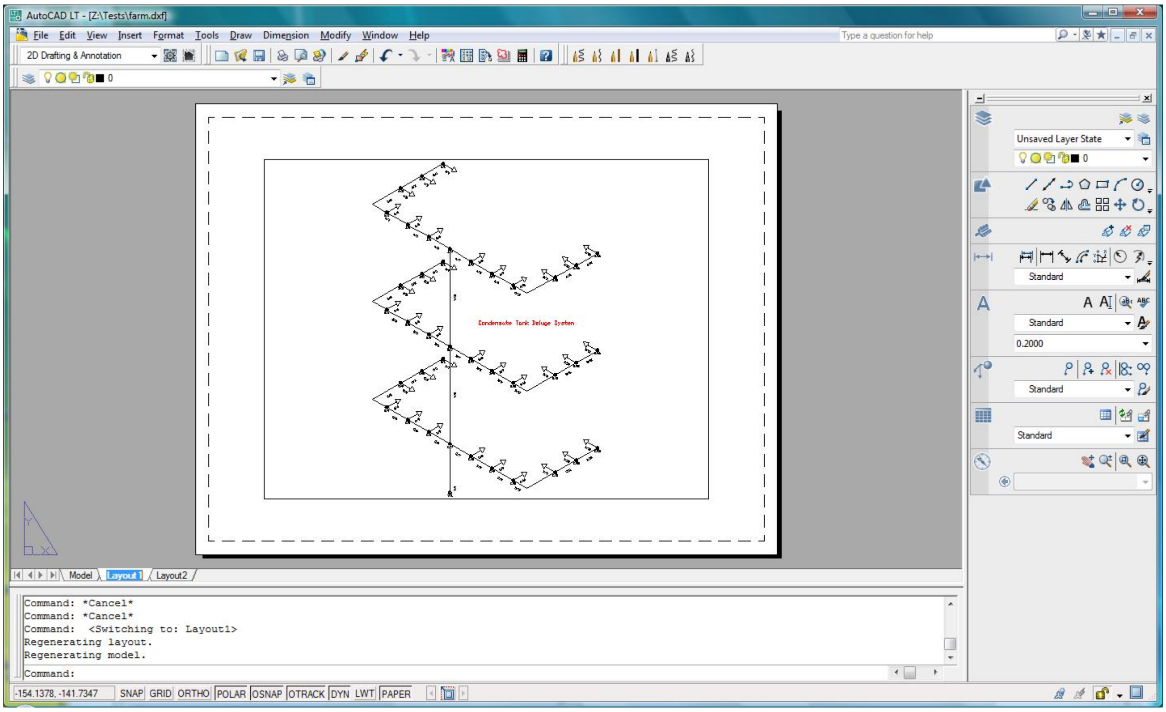
Via the **Tools | Export DXF file** menu option the schematic can be output as an AutoCAD DXF ASCII file. The format of the DXF files produced is according to the Autodesk DXF Reference March 2006 [ACAD06]. The DXF format is a complex format and one which can change with releases of AutoCAD. The output produced by PIPENET has been verified by loading the files into AutoCAD 2008, but is also know 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:


AutoCAD LT - [Z:\Tests\farm.dxf]
Updating handle seed.
Opening an AutoCAD 2004 DXF file.
Regenerating model.
Command:


AutoCAD LT - [Z:\Tests\farm.dxf]
Command: *Cancel*
Command: *Cancel*
Command: <Switching to: Layout1>
Regenerating layout.
Regenerating model.
Command:

azarnasim

AIR CONDITIONING COMPANY

Part 21

AIR CONDITIONING COMPANY

21 Errors

21.1 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.

21.2 Basic errors

These are usually related to the absence of a valid licence to run the program or incorrect operation of the security key drivers, but other options are also possible:

- Attempts to run the software on earlier versions of Windows operating system may fail – the software is only validated for Windows 2000 and XP.
- A working PIPENET system may suddenly stop working if a new operating system is installed or a new Windows service pack is installed. Generally Sunrise Systems will discover these errors in advance and be able to provide suitable fixes, new security key drivers, etc.
- No valid licence – separate licences are available for the three PIPENET modules Transient, Spray/Sprinkler and Standard. Attempting to run a module for which you have no licence will produce an error. To determine which licences you have see the **Help | Key status** menu option . If in doubt, contact Sunrise Systems for details of your current licences.
- Licence has expired – with the issue of PIPENET VISION 1.10 expiry date checking was added to HASP security keys. If an old key is being used, or if the licence for the module you are attempting to run has expired, then a message will be displayed informing you of the situation. To see the expiry dates see the **Help | Key status** menu option, if no expiry dates are shown then you have an old key that needs to be updated. Keys can be updated remotely by Sunrise Systems providing (usually via email) a small encrypted key-update file – see the Updating a key licence section for further details.
- Drivers for security keys not installed correctly – see the installation instructions supplied with the software for information on how to check the status of the drivers and if necessary re-install them. Periodically the supplier of the security

key drivers will have to issue new versions of the drivers when a new operating system or service patch is released. When this occurs Sunrise Systems will make the drivers available on our website or via email.

21.3 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 contains 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.

21.4 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 Standard module the following errors are possible, classified as errors, warnings or information:

Blocked pipe found – warning
A blocked pipe has been found – this is a warning message only but is noted since blocked pipes can have an expected affect on a calculation. If this item is double-clicked in the Check dialog the pipe with the block will be highlighted.

Broken pipe found – warning
A broken pipe has been found – this is a warning message only but is noted since blocked pipes can have an expected affect on a calculation. If this item is double-clicked in the Check dialog the pipe with the break will be highlighted.

Calculator does not permit non-scheduled pipes whilst in the design phase – error
One or more pipes do not belong to a schedule and therefore the design phase cannot be run.

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.

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 is under specified in design phase - error
There are insufficient specifications in the design phase, that is the number of specifications and number of input/output nodes do not meet the specification requirements.

Network must have at least one inlet - warning
The network must have at least one inlet.

Network must have outlets or nozzles – error
The network must have one or more outlets.

No analysis phase pressure specification – error
At least one pressure specification must be provided for the analysis phase – see specifications requirements.

This node may need to declared as an input node - warning
This warning message will appear for each node that has a single pipe that uses the node as an input, but there is no pipe that uses the node as an output. That is, there appears to be flow out of the node but there is no flow into the node.

This node may need to declared as an output node - warning
This warning message will appear for each node that has a single pipe that uses the node as an output, but there is no pipe that uses the node as an input. That is, there appears to be flow into the node but there is no flow out of the node.

No design flow specification – error
If a design phase is being used then there must be one or more design phase flow specifications – see specifications requirements.

No design phase pressure specification – error
If a design phase is being used then there must be one design phase pressure

specification – see specifications requirements

Non-pipe component has unequal input and output elevations - error
If node elevations are being used then the input and output nodes of all non pipe components, for example pumps and valves, must have the same elevation.

One I/O node with no flow spec is required for Design - warning
If a design phase is being used then there must be at least one input/output node with no flow specification – see specifications requirements

Only one design phase pressure specification allowed – error
If a design phase is being used then there must be one and only one design phase pressure specification – see specifications requirements

Pipe type <pipe-type-name> has a lining that is too thick – error
The pipe named pipe type has a lining with a thickness that is greater than or equal to half the smallest used bore.

Reversed flow at inlet- warning
There is a reversed flow at an input node. This will occur if (and taking specifications into account) the total flow into the network is greater than the flow out of the network and the node has been specified as an input node with the value for the flow specification being unset.

Reversed flow at outlet- warning
There is a reversed flow at an output node. This will occur if (and taking specifications into account) the total flow out of the network is greater than the flow into the network, and the node has been specified as an output node with the value for the flow specification being unset.

There must be at least two specifications - error
There must be at least two specifications provided for the analysis phase – see specifications requirements.

Water temperature is greater than 368.15 K (95 degrees C) - warning
A warning is issued if the selected fluid is water and the temperature is greater than 368.15 K.

21.5 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)
- Roughness more than 10% of diameter (ducts)
- Coefficients must be less than zero (filters)

- Valve setting must be given (control valves)
- Pump efficiency must be given (pump)
- Orifice plate - pressure drop must be given (orifice plates)

Note that many of these errors are of the general form *<attribute name> must be given,* as in the last three examples above, indicating that a value for the attribute has not been provided.

Double-clicking on the error in the Check status dialog will highlight the component in the schematic.

21.6 Numerical errors

There is a general category of errors that cannot be detected in the GUI because they can only be checked by the calculator, either in setting up the calculation, or during the calculation itself. For example:

- calculation fails to converge – this may because the calculation cannot be solved as defined, or it may just require a few more iterations than are specified in the calculation options. A calculation can also fail to converge if the network is incorrectly specified in some way.

Also note that when you request a calculation a check is performed first and any errors or warnings noted in the check status dialog.

Part 22

22 Reference Data

22.1 Bibliography

[CRANE] Technical paper No 410M "Flow of Fluids Through Valves, Fittings and pipe"; Crane Co, 1988

[POLING] "The Properties of Gases and Liquids", Poling, Prausnitz and O'Connell, Fifth Edition, McGraw Hill, 2000.

[HPGL2] "The HP-GL/2 and HPRTL Reference Guide", Third edition, Addison Wesley, 1997

[ACAD06] "DXF Reference", March 2006, Autodesk

[IAPWS] Industrial Formulation 1997 for the Thermodynamic Properties of Water and Steam (IAPWS-IF97).

22.2 Properties of water

The physical properties of water are built into PIPENET. A selection of the properties for the temperature range 5 - 90°C are shown below:

Temp °C	**Viscosity**		**Density**		
	Pa s	Cp	lb/ft.hr	kg/m^3	lb/ft^3
5	0.001506	1.506	3.643	999.9	62.42
10	0.001300	1.300	3.145	999.7	62.41
15	0.001135	1.135	2.746	999.1	62.37
20	0.001002	1.002	2.424	998.2	62.32
25	0.000891	0.891	2.156	997.0	62.24
30	0.000797	0.797	1.928	995.6	62.15
35	0.000718	0.718	1.737	994.0	62.05
40	0.000651	0.651	1.575	992.2	61.94
50	0.000544	0.544	1.316	988.1	61.69
60	0.000463	0.463	1.120	983.3	61.38
70	0.000400	0.400	0.968	977.5	61.02
80	0.000351	0.351	0.849	971.8	60.67
90	0.000311	0.311	0.752	965.3	60.2

22.3 Surface Roughness

The following table shows values of surface roughness for typical materials. These values have been obtained from a number of sources and are not endorsed by Sunrise Systems Limited.

Material	Roughness	
	(mm)	(inch)
Asbestos Cement	0.015 – 0.06	$5.91\ 10^{-4}$ to $2.36\ 10^{-3}$
Brass	0.001 to 0.003	$3.94\ 10^{-5}$ to $1.18\ 10^{-4}$
Cement – smoothed	0.3	$1.18\ 10^{-2}$
Concrete - ordinary	0.6 - 1.5	$2.36\ 10^{-2}$ to $5.91\ 10^{-2}$
Concrete – precast	0.06 – 0.6	$2.36\ 10^{-3}$ to $2.36\ 10^{-2}$
Concrete – coarse	0.3 – 5.0	$1.18\ 10^{-2}$ to $1.97\ 10^{-1}$
Copper	0.001 to 0.003	$3.94\ 10^{-5}$ to $1.18\ 10^{-4}$
Glass Reinforced Plastic (GRP)	0.03 – 0.06	$1.18\ 10^{-3}$ to $2.36\ 10^{-3}$
Lead	0.001 to 0.003	$3.94\ 10^{-5}$ to $1.18\ 10^{-4}$
PVC and plastic pipe	0.0015 – 0.007	$5.91\ 10^{-5}$ to $1.18\ 10^{-4}$
Cast iron		
New - unlined	0.25 to 0.8	$9.84\ 10^{-3}$ to $3.15\ 10^{-2}$
Worn	0.8 to 1.5	$3.15\ 10^{-2}$ to $5.91\ 10^{-2}$
Rusty	1.5 to 2.5	$5.91\ 10^{-2}$ to $9.84\ 10^{-2}$
Asphalted	0.12	$4.72\ 10^{-3}$
Steel		
Stainless	0.015	$5.91\ 10^{-4}$
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	$5.91\ 10^{-4}$

Material	Roughness	
	(mm)	(inch)
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

22.4 Hazen-Williams Coefficients

The following table provides guidance and suggested values for C-factors. These values have been obtained from a number of sources and are not endorsed by Sunrise Systems Limited.

Material	C-factor	
	Possible values	Suggested value (NFPA13)
Asbestos Cement	140 - 150	140
Brass	120 - 150	
Black steel (dry systems)	100	100
Black steel (wet systems)	120	120
Cast iron		
New unlined	120 - 130	100
10 years old	105 - 75	

Material	C-factor	
	Possible values	Suggested value (NFPA13)
15 years old	100 – 60	
20 years old	95 – 55	
30 years old	85 – 45	
50 years old	75 - 40	
Bitumen-lined	140	
Cement-lined	140	140
Concrete	85 - 150	140
Copper	120 - 150	150
Fibre glass pipe	150 - 160	
Fire hose (rubber)	135	
Galvanized steel	120	120
Lead	130 - 150	
Polyethylene	150	
PVC and plastic pipe	150	150
Stainless steel	150	150
Steel new and unlined	140 - 150	
Steel, welded and seamless	100	
Vitrified clays	110	

When a range of values is provided the smoother value (higher C-factor) is for new pipes and the lower value for old pipes. When a single value is quoted this is for new pipes. Unfortunately there is no single source for the above data, and the data has been compiled from a number of sources, including:

1. NFPA13, 2007 edition
2. The Engineering Toolbox (www.engineeringtoolbox.com)
3. Binder, R.C. (1973), Fluid Mechanics, Prentice-Hall, Inc. (Englewood Cliffs, NJ).

4. Harry E. Hickey (1980), Hydraulics for Fire Protection, NFPA publication

22.5 Physical Constants

All values are SI.

Atmospheric Pressure	101325 Pascal
Acceleration due to gravity	9.80665 ms^{-2}
0° C	273.15 Kelvin

22.6 Built-in Fittings

Note the abbreviations are those used in the old style .DAT files

BEND	90 deg bend (r/d = 1.5)
ELBOW	90 deg elbow
TEE	Tee (run)
BRA	Tee (branch)
ELB45	45 deg elbow
BALL	Ball valve
BFLY	Butterfly valve
GATE	Gate valve
GLOBE	Globe valve
ANGLE	Y - pattern valve
GLANG	Globe/angle valve
CHECK	Y swing check
GLBSC	Globe swing check
GLBLC	Globe lift check
ANGLC	Y lift check
DISC1	5 deg tilting disc
DISC2	5 deg tilting disc
GLOST	Globe-stop check
ANGST	Angle-stop check
YGLST	Y globe-stop check
YANST	Y angle-stop check
FOOTP	Poppet foot valve
FOOTH	Hinged foot valve
PLUG	Plug valve
COCK2	3 way cock (run)
COCK3	3 way cock (branch)
RETB	Return bend
INP	Projecting entrance

FLUSH	Sharp flush entrance
EXIT	Pipe exit

Losses due to the presence of fittings are calculated in accordance with the Crane manual [CRANE].

22.7 Built-in Fluids

Data for a number of common fluids is built into PIPENET. This enables users to rapidly set up problems involving these fluids. The built-in fluids are as shown below:

Class 1: Water

Class 2: Saturated and super-heated steam
The properties of saturated and super-heated steam are calculated using correlations from the 1967 IFC Formulation for Industrial Use.

Class 3 or 4: built-in fluids
For all these built-in fluids, the only information the user must supply is the fluid temperature:

- Oxygen
- Nitrogen
- Carbon Dioxide
- Air
- Water Vapour
- Methane
- Propane

22.8 Built-in Gases

The following table provides details of the specifications used for each of the seven built in gases:

Code	Name of Fluid	Molecular Weight (kg/kgmol)	Critical Temp (K)	Critical Pressure (bar)	Critical Volume (m^3/kg)	Ratio of Specific Heats
1	Oxygen	32.0	154.8	50.76	0.00233	1.40
2	Nitrogen	28.0	126.2	33.94	0.00322	1.40
3	Carbon Dioxide	44.0	304.2	73.87	0.00214	1.28
4	Air	28.97	132.5	37.66	0.00305	1.40
5	Water Vapour	18.02	647.0	221.19	0.00311	1.31
6	Methane	16.04	190.65	46.41	0.00620	1.28
7	Propane	44.1	369.0	42.56	0.00454	1.1

Index

- . -

- < -

- A -

- B -

- C -

- D -

- E -

- F -

- G -

- H -

- I -

- W -

- Z -