

PUMPSIMTM

3D PUMPING SIMULATION SOFTWARE

PumpsimTM

Version 2.2

User Manual

Copyright © 2016 Chasm Consulting

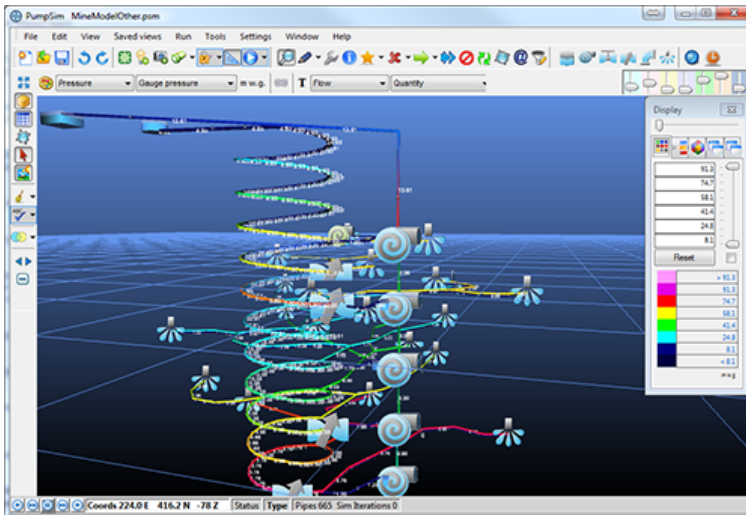
Table of Contents

1. Introduction	4
1.1 What's New	5
1.2 System Requirements	5
1.3 Support & Maintenance	5
1.4 License Manager	7
2. Pumpsim™ User Interface	9
2.1 The Menu Bar	13
2.2 The View Toolbar	40
2.3 The Action Toolbar	45
2.4 The Data Toolbar	59
2.5 The Popup Context Menu (Right-Click)	62
2.6 The Scaling Manager	67
3. Pumpsim Elements	69
3.1 Pipes and Channels	69
3.2 Fittings	80
3.2.1 Tanks/Dams/Recievers	82
3.2.2 Valves	88
3.2.3 Sprays	91
3.2.4 Junctions	92
3.2.5 Pumps/Compressors	95
3.3 Fluids	105
3.4 Scenarios	107
4. Building A Model	111
4.1 Initial Model Construction	111

4.2 Manual Scaled Construction	112
4.3 Manual Schematic Construction	113
4.4 Spreadsheet Text Import	113
4.5 Layers	113
4.6 DXF Graphics Import	115
4.7 Simulation	121
4.8 FAQ	123
4.9 Dynamic Simulation Report	124
4.10 Step by step guide	125
5. Configuration	133
5.1 Presets	133
5.2 Settings	136
6. Appendix	154
6.1 Appendix A - Glossary of Terms	154
6.2 Appendix B - Summary of Data Types	156
6.3 Appendix C - Icon Picture Guide	158
6.4 Appendix D - Display Problems	161
6.5 Appendix E - Simulation Errors	163
6.6 Appendix F - Shortcut Keys	168

1. Introduction

Welcome to Pumpsim™!



Pumpsim™ is designed to simulate pumping and water network systems to design systems, greatly improve efficiency and reduce costs for existing sites and future developments.

Pumpsim™ provides the user the tools to:

- Simulate **flows** and **pressures** in an existing network.
- Perform 'what if' simulations for planned new development.
- Help in short term and long term planning of flow requirements.
- Assist in selection of pumps and valves.
- Calculate installation and running costs.

Pumpsim™ has been designed to make the process of pipe network model analysis as intuitive, informative and easy to use as possible. Pumpsim™ utilises sophisticated 3D graphics, driven by a fully graphical mouse driven interface. Pumpsim™ is compatible with **Microsoft Windows XP, VISTA, WINDOWS 7** and **WINDOWS 10**. The software can also be run on Apple Mac Computers with suitable graphics hardware, running Microsoft Windows under dual boot or emulation.

Pumpsim™ automatically installs as a 32 bit or 64 bit version. The 64 bit version has the ability to utilise additional computer memory, and therefore much larger models can be used. Recommend maximum model sizes are 3,000 pipes for the 32 bit version, and up to 10,000 pipes for 64 bit versions. Increased amounts of DXF reference data can also be set to be displayed in the 64 bit version.

1.1 What's New

[View the Update Log.](#)

1.2 System Requirements

Pumpsim™ relies heavily on 3D graphics hardware to present detailed smooth graphics. The following guide is recommended for computers.

Minimum Configuration

- Intel based (AMD or Intel) processor 1 Ghz +
- Windows XP, Vista or Windows 7 / 8 / 10
- 2GB RAM, Hard Drive Space 100 Mb
- Direct X9 compatible graphics (minimum Intel integrated graphics)
- Two button mouse

Recommended Configuration

- Intel based 32 or 64 bit dual core processor 2 Ghz +
- Windows XP, Vista or Windows 7 or Windows 10, 32 or 64 bit
- 4GB Ram. Hard Drive Space 100Mb +
- Dedicated /Discrete Graphics card from INTEL, ATI or NVIDIA with greater than 128Mb or RAM. INTEL "COREi" processors (generally sold from 2010 onwards) now have reasonable graphical power and perform well with Pumpsim™ without a dedicated graphics card.
- Two button mouse with centre scroll / click wheel
- Pumpsim™ is not officially supported on Apple Mac computers, however it has been confirmed that Pumpsim™ is capable of running on newer Macs under Windows dual boot or Parallels™ virtualisation software.

Ideal Configuration

- Windows 7 / 10 64 bit. 4GB or more RAM, Dedicated 1GB+ NVIDIA or ATI graphics card.

1.3 Support & Maintenance

Chasm Consulting is committed to providing technical support for all Pumpsim™ products. Please contact us so that we can address any technical issues and concerns that you may have.

Remote Assistance

Chasm Consulting can assist with the set-up and training of Pumpsim™ remotely for clients anywhere in the world. Using [TeamViewer](#) screen-sharing software, we can walk through the features of Pumpsim™ and assist in setting up the software to address the specific needs of your organisation. Please contact us to discuss this option further.

On-Site Assistance

Chasm Consulting offers an on-site set-up assistance and training service tailored to the specific needs of your company. Please contact us to discuss this option further.

Maintenance

Annual maintenance fees contribute to the development and improvement of Pumpsim™, and provide customers with the following benefits:

- Software updates, improvements and new features
- Immediate and ongoing access to new versions of the software
- Full technical and license support
- You can check if your maintenance is up to date using the **HELP > ABOUT** menu in Pumpsim™.

If your maintenance has expired, please consider obtaining a quotation for updated maintenance. The maintenance for expired customers will include an upgrade cost to cover past improvements and upgrades to the very latest version, and also a component for future upgrades and support to a date of your choosing.

Maintenance Extension Policy

Maintenance for Pumpsim™ licenses are applied from the previous expiry date. Where lapsed maintenance is renewed, the portion of maintenance to the current date becomes an **“upgrade”** cost (which helps us cover the cost of implementing and licensing all the new features which have been implemented since expiry), and further amounts after this date are for future updates and support. For an extended lapse in maintenance renewal, maintenance upgrade charges are limited to a maximum of 50% of a license cost. While maintenance is entirely voluntary, this effectively means that unless the maintenance lapses for 2.5 years or more, there is no benefit in not renewing maintenance when it becomes due.

This arrangement is consistent with nearly all mining software licenses, due to the small market size and high development costs. Other types of software with larger markets often withhold features and release major new versions every couple of years, however it is common to charge 50% or more to upgrade to these new versions.

We believe a maintenance scheme which allows users to immediately access new features and versions is a more equitable system, however because our maintenance system is also capped at 50% this provides an

alternative infrequent upgrade scheme to those who did not wish to pay maintenance every year.

1.4 License Manager

Pumpsim™ licenses are floating licenses which can only be used on one computer at a time. They can be activated for use on one computer and then if required released and used on another computer (within the terms of the license agreement). License certificates are stored on the Pumpsim™ server and prevent simultaneous installation of the same license on more than one computer.

As part of the digital licensing agreement, the license collects the computer name and user name of the computer that Pumpsim™ is currently activated on. This system protects against unauthorised use of the Pumpsim™ license and informs other users who has currently activated the software license.

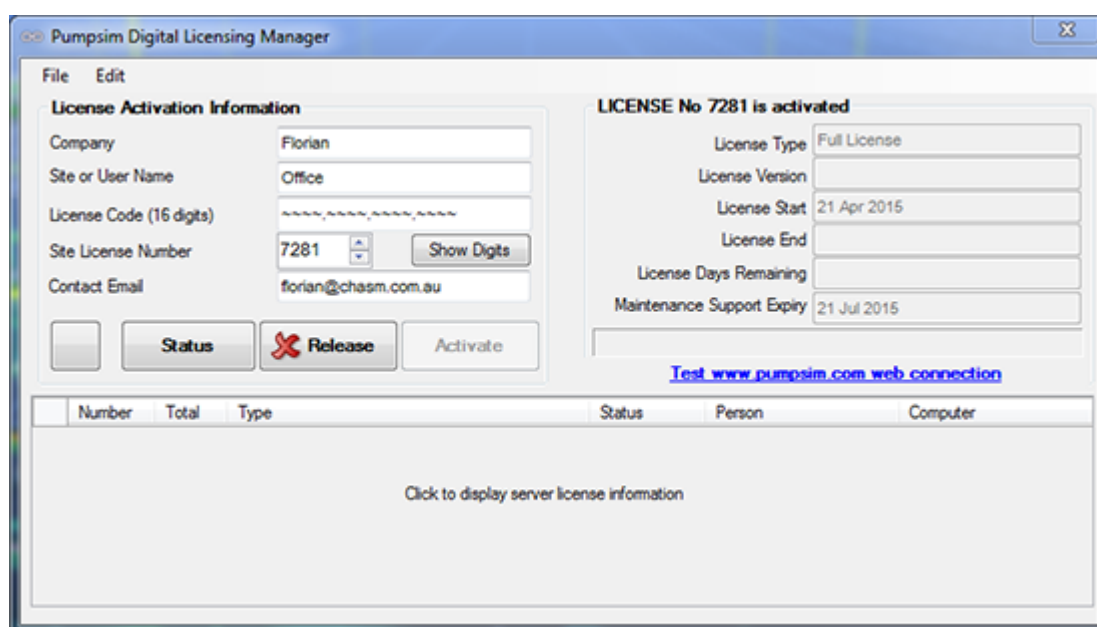


Fig. 1.4.1: The Pumpsim™ Digital Licensing Manager.

Releasing a License

Once a license is activated on a computer, it cannot be moved to another computer unless it is first released from the activated computer. If another computer tries to activate the same license simultaneously, this will be prevented and a warning will display the currently licensed computer and the login user name of the person who has installed it.

An internet connection is required to activate and release license certificates from a computer. If an internet connection is not available or an internet connection is not authorised by your firewall or company administrator, try to arrange a link to the <http://www.pumpsim.com/> website. If this cannot be done, a manual license transfer can be performed via email located under the file menu.

License Troubleshooting

In the event the **License Manager** fails to activate or release the license, note the error message and take the following actions:

- If the error message reports the license is active and in use by another computer, then only that computer can release the license. If this computer is no longer available and license has not been released then contact admin@chasm.com.au for further assistance.
- If the error message reports an internet connection problem, then click the “**TEST http://www.pumpsim.com/ web connection**” link on the **License Manager** form. Ensure that Microsoft Internet Explorer can link to the Pumpsim™ webpage. If this does not work, then the Windows connectivity problem needs to be resolved before further action can be taken in Pumpsim™. Ensure an internet connection is available, and the firewall or proxy server is not blocking the Pumpsim™ website.
- Some companies have proxy settings that inhibit third party programs using the internet. In most cases, Pumpsim™ will automatically adopt the same proxy settings as Microsoft Internet Explorer. If your company blocks third party software from accessing your internet, you will need to enable a proxy exception on your system to access to our licensing website at www.ventsim.com.
- If the license is a **DEMO** or a **STUDENT / EDUCATION** license, and needs to be removed from the computer, click on the **FILE > RESET LICENSE** menu option in the **License Manager**.

2. Pumpsim™ User Interface

Pumpsim™ operates in a full three dimensional (3D) graphical environment. The main Pumpsim™ window contains all the functions you will require to **create**, **edit**, **view** and **simulate** pipes and models.

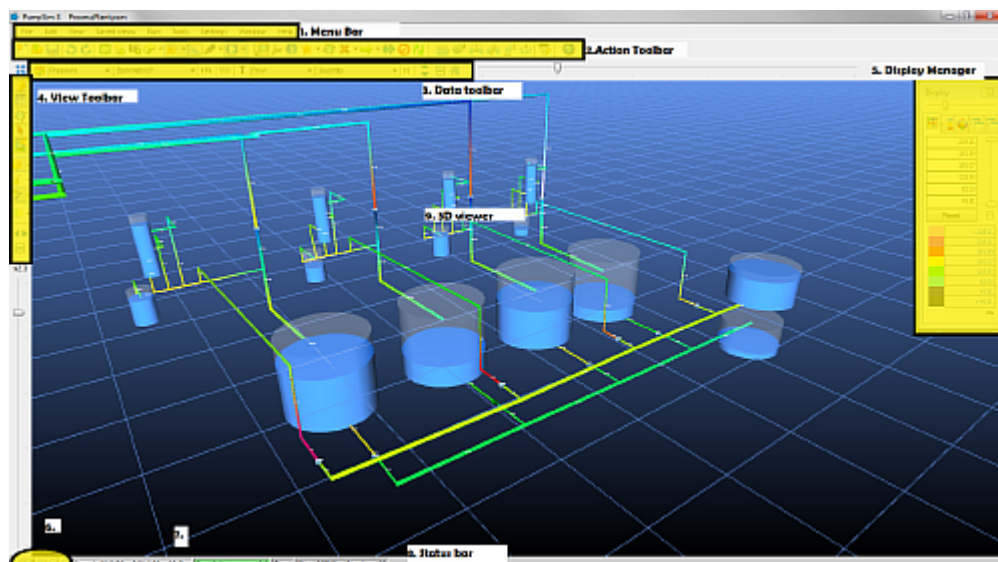


Fig. 2.0.1: A Pumpsim™ window open with the Menu bar, "View" Toolbar, "Action" Toolbar, "Data" Toolbar, Display Manager, Data Position Control and Simulation status highlighted in yellow.

1. **Menu Bar** – Consists of options required to load, save, view and manipulate the model, as well as options to change settings and simulate models.
2. **"View" Toolbar** – Contains options to show or hide various graphics items such as arrows, text, nodes and reference graphics (DXF). This toolbar can be hidden or moved to the left or right of the screen.
3. **"Action" Toolbar** – Consists of many of the construction tools required to build models, as well as options to save or load new model and alter animation speeds for arrows. This toolbar can be hidden, but not moved to other locations.
4. **"Data" Toolbar** – Allows the text and colours of pipes and channels to be changed. Displays or hides the colour legend control. This toolbar can be hidden or moved to the upper or lower part of the screen.
5. **Display Manager** – Contains controls to alter the colour or transparency of pipes and channels on the screen. In addition it can also change the display to show different layers, levels (elevations) or water types on display.
6. **Data Position Control** – Selects the position along a pipe or channel from which data will be displayed as text on screen or as colours. The condition of water will change along a pipe or channel, therefore this option is available to select which position along the segment display data is sourced from. The first and last button displays the conditions of water entering and exiting a segment. The middle button displays an average of data through the segment.
7. Mouse cursor position coordinates in the 3D screen.
8. **Simulation status** – Green = successful simulation, Yellow = successful simulation with warnings, Red = unsuccessful simulation.

The successful utilisation of some of the unique aspects of a 3D perspective view comes from an understanding of how this software functions. Pumpsim™ has a number of guides to assist in viewing and construction in 3D.

Point of Focus

The view in the main 3D Pumpsim™ window is essentially a view seen from a 'camera' floating in space, aimed at a **point of focus**. Your eye is the camera. The point of focus is always in the centre of the screen, at a pre-determined distance away from the camera.

Moving the Point of Focus

You can move closer or further away from the point of focus with the **Mouse Scroll** button, and rotate around this point of focus with the **Mouse Right** button. To move the screen to a different location, the point of focus needs to be moved.

There are several ways to move the point of focus;

- Draw a window with the mouse around the area you wish to focus on. The focus will move to the centre of the window, at the closest distance of an object (pipe, channel or node) in the window.
- Drag (pan) the screen with the **Centre** mouse button. The point of focus will move along the current horizontal plane with the mouse cursor
- Click on an object while in the view mode, or when in any other mode with the **Mouse Middle** button (or a **Right/Left** combination for those without a 3 button mouse). This will automatically put the focus onto the clicked pipeline at the same viewing distance as was previously set. If the object being clicked is a long way away, this has the effect of quickly flying through space towards that object in order to maintain the same view distance as the previous point of focus.
- The **Elevation** of the point of focus can also be changed by selecting the **Shift** key while using the **Mouse Scroll** button. A **transparent grid** (if the grid is turned on) will show the horizontal plane of the point of focus.

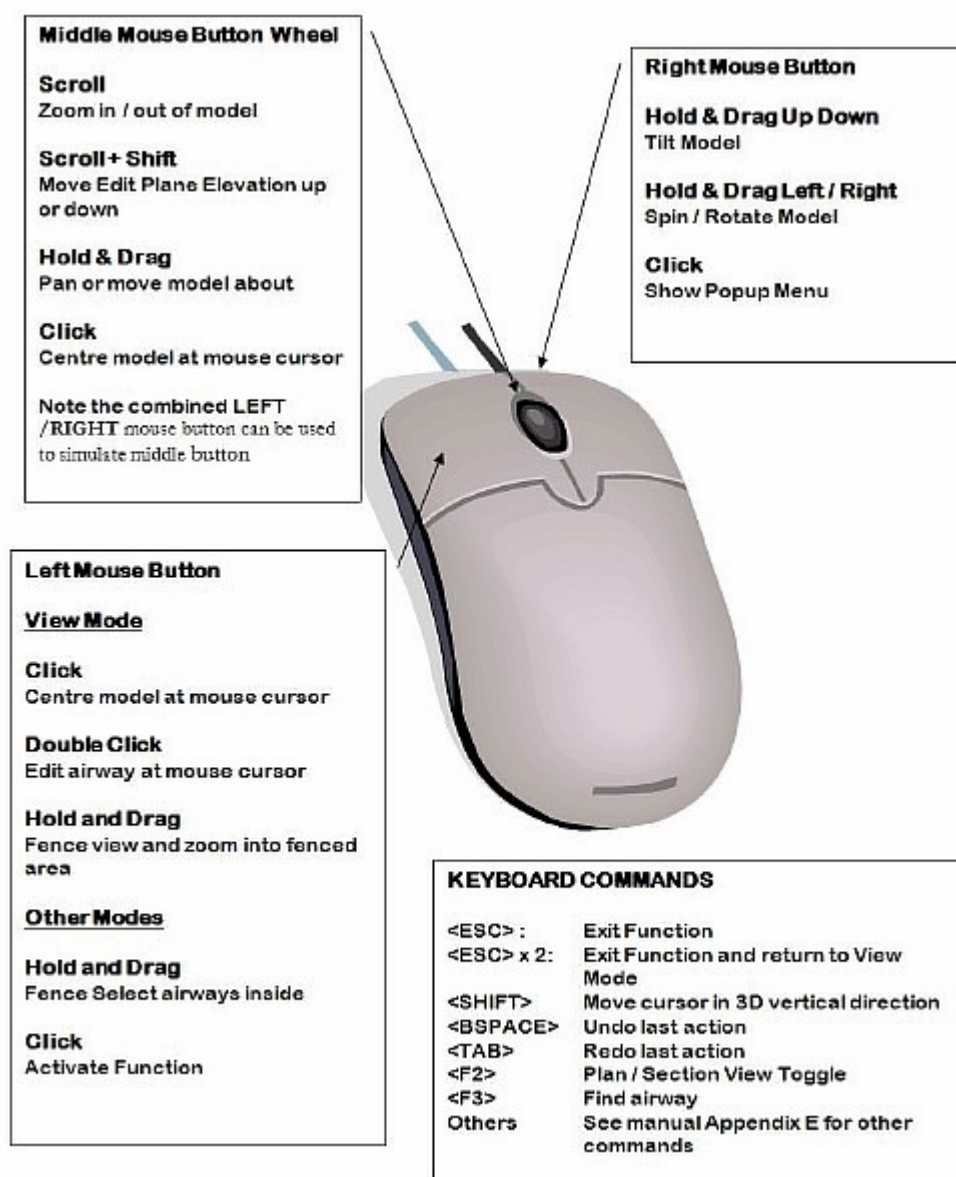


Fig. 2.0.2: Mouse commands.

The Edit Plane

The **Edit Plane** is a horizontal plane at a set elevation. By default, any new objects are initially constructed on the edit plane. The edit plane can be viewed by ensuring the grid function is turned on. The plane will be crossed with grid coordinate lines. If the **Shift** key is held, the edit plane will be made semi-transparent which will indicate where it intersects existing pipes and channels.

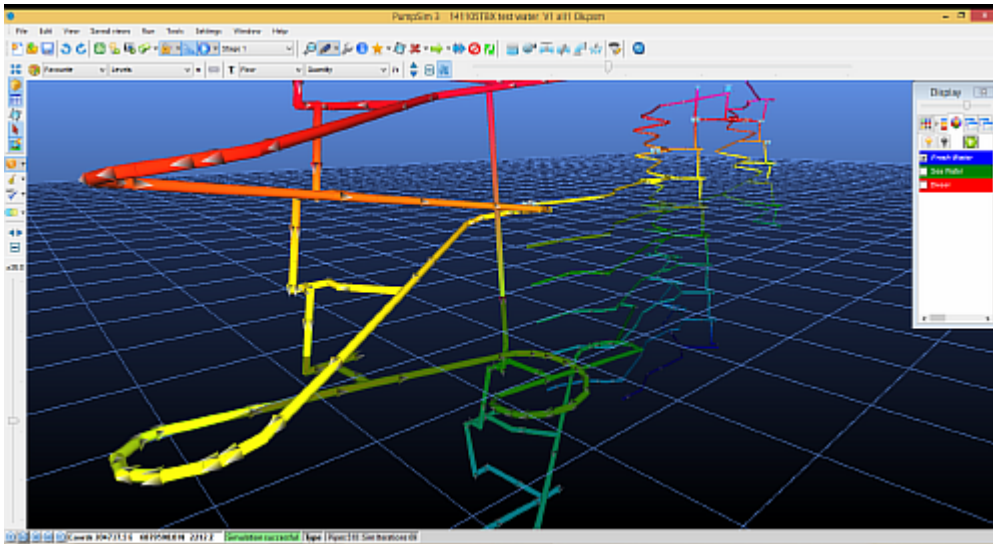


Fig. 2.0.3: The **Edit Plane** is the horizontal plane at the elevation of the user.

To move an edit plane, select a new point of focus by centering on an object, or use the **Shift-Mouse-Scroll** combination to move the edit plane elevation. The coordinates in the status bar at the bottom left corner of the screen will always show the elevation of the edit plane.

Hint: The Edit Plane will also temporarily move automatically to the level of an object being drawn from another object. Additionally, the edit plane and point of focus can be moved manually by the **Set Edit Plane** function in the **View** menu.

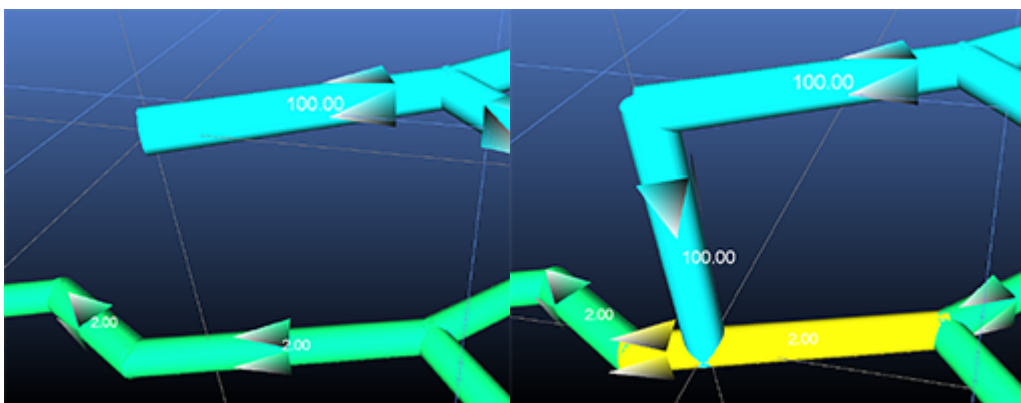


Fig. 2.0.4: Changing the **Edit Plane** to draw pipes.

Drawing in the Third Dimension

Drawing in three dimensions can be a challenge as the view is being displayed on a two dimensional screen monitor. Pumpsim™ alleviates this problem by only drawing on the horizontal plane **except** when the **shift** key is pressed, or the screen is orientated to a vertical cross section orientation.

The True Vertical Guide

3D perspective views do not necessarily show a vertical object as pointing straight up. As objects move further left and right of the point of focus, they 'lean' over away from the centre of the view. To assist the user as to which direction is truly up while creating or editing, a **true vertical line** is displayed while drawing, moving or copying. If the object being drawn aligns with the **true vertical line**, then it is vertical.

The true vertical line can also assist in locating objects directly under or over other objects at different elevations by observing where the line 'intersects' objects above and below the current point being edited.

2.1 The Menu Bar

The Menu Bar accesses a host of functions including simulation options and settings, and viewing options and settings. Many of the main menu bar function are duplicated in the toolbar buttons.

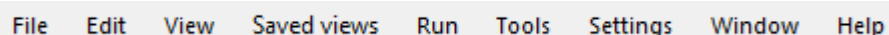


Fig. 2.1.1: The Menu Bar.

"File" Menu

- New/Close -** Erases the model currently in memory. An option is given to save any unsaved changes from the current model. The previously used **Level**, **Layer** and **Pump Database** will be retained for the new model. The **New** option closes the model, and load up the system startup defaults, whereas the **Close** option closes the model, but retains the previous model settings and defaults.
- Open -** Loads a pre-saved model. You can also open Pumpsim™ files by dragging file icons from your Windows folders onto the Pumpsim™ screen.
- Merge -** Joins two models together instead of erasing the currently loaded model. Similar to the Open command, this may be useful for joining separate modelled areas of the same site. Caution should be taken however, as duplicate objects are not immediately checked when the models are merged (duplicate objects will be subsequently be deleted if an attempt is made to simulate the model).
- Save -** Saves changes made to the model. If the Pumpsim™ title bar shows that the model is untitled, the user will be prompted to select a name before the file can be saved.

Save As - Saves the model, but gives the option of saving under a different name. Pumpsim™ can be saved in one of two formats. The default format in the PSM file, which is the standard file format. This format is highly compressed and cannot be read by other programs.

Pumpsim™ files can also be saved as a **Text** format. This format follows the standard TAB separated values format and can be loaded by programs such as Microsoft EXCEL, WORD or ACCESS. The internal contents of the file can be viewed, modified and resaved as a **Text** file. The **Text** file can be reloaded into Pumpsim™ providing the basic structure and the HEADER and FOOTER tabs remain the same.

Master Link

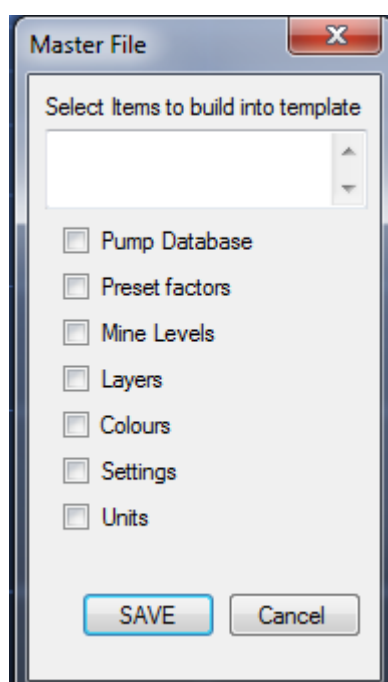


Fig. 2.1.2: The Master File window.

Enables a common settings file (Master File) to be linked and shared with multiple Pumpsim™ files.

Master Files store a user definable selection of shared settings (for example resistance and friction factor preset values or pumps). When a Pumpsim™ file is linked to a Master File, and the file is saved, the settings are also saved to the Master File and will be available to all other Pumpsim™ files which have a link to the Master File. The settings in linked Pumpsim™ files are updated from the Master File when loaded. If the Master File is not available, then a warning will show and the most previously saved settings will be used.

Warning: Using linked Master Files can be dangerous if settings made in one file adversely affect another linked file. For example if a pump is removed or replaced with another pump, and the file saved, then all other linked Pumpsim™ files which used to use this pump may no longer work correctly. If a linked Master File is used, it is normally better to add new settings, not remove or delete existing settings which may be used by other files.

Create New - Creates a new Master File template, which can then be linked to other Pumpsim™ files. When creating the file, it is automatically linked to the current file. Other Pumpsim™ files can be linked to this same Master File by opening the files and using the Link

option below. Save the Master File in an accessible file location. A descriptor can be saved with the file to explain information about what components have been saved.

***Hint:** When creating a new file, there is an option to specify what common settings you would like to make in the master file. For example if only the pump database, and the common resistance, friction and shock factors are to be used, then click on the Pump Database, and Preset options in the dialog box.*

- Link -** Opens a dialog form to search and link to a Master File. If the current Pumpsim™ file has an existing linked Master file, the new Link will replace the data in the Pumpsim™ file. An alternative method to link a file is to simply drag and drop a Master File into the current Pumpsim™ window.
- Unlink -** Breaks a link to the Master file (but does not change the Master File). Any changes to file settings made after the link is removed will no longer update the Master File.
- Update -** Refreshes the current Pumpsim™ file with the data in the Master file. This may be necessary if another file has modified the data in the Master file after the Pumpsim™ file has been opened. It is normally not recommended to have multiple files open accessing the same Master File as only the most recently saved Pumpsim™ file will have updated the master settings. If another file has updated the Master File while the current file is open, then a WARNING will display indicating a possible conflict.

***Caution:** When selecting 'REPLACE' many components such as pumps databases, levels and most presets may not correctly map to the existing model's pumps, levels and presets, particularly if (for example) pumps within the database are in a different order. These may need to be manually corrected by checking and re-editing pump pipes to ensure the correct pumps have been placed.*

Defaults (Pumpsim™ Startup Settings)

The **Defaults** file is stored in the user's personal Windows directory. The file is loaded when Pumpsim™ starts and specifies the settings, behaviour and pumps when the program is first loaded. Each user who logs on to the computer will have a different default file which is initially created when the program is installed.

Pumpsim™ files have a copy of defaults stored within the file which may have been modified after the model file was started. This will override the standard start up defaults when the file is loaded.

Save

Saves the settings currently loaded in memory to the defaults file. These will be loaded automatically next time the program is run.

Restore

Restores the defaults file to the original file created when the Pumpsim™ program was installed.

Refresh

Reloads the original defaults file that is first loaded when Pumpsim™ is initiated.

This may be required if the default settings that are automatically loaded with a Pumpsim™ simulation file are incorrect or out of date, or you wish to overwrite the current file settings.

Caution: This will reset all file parameters such as pumps databases and graphics options. If you wish to only update selected components, use the inherit function to load selected components from an existing file.

Inherit Attributes

Adopts selected attributes from another model. Pumpsim™ files contain many different components such as pump databases, file and simulation settings, level and layer database and many other options. Instead of setting up new parameters for a model, these components can be loaded from an existing file without deleting the pipe data in the current file.

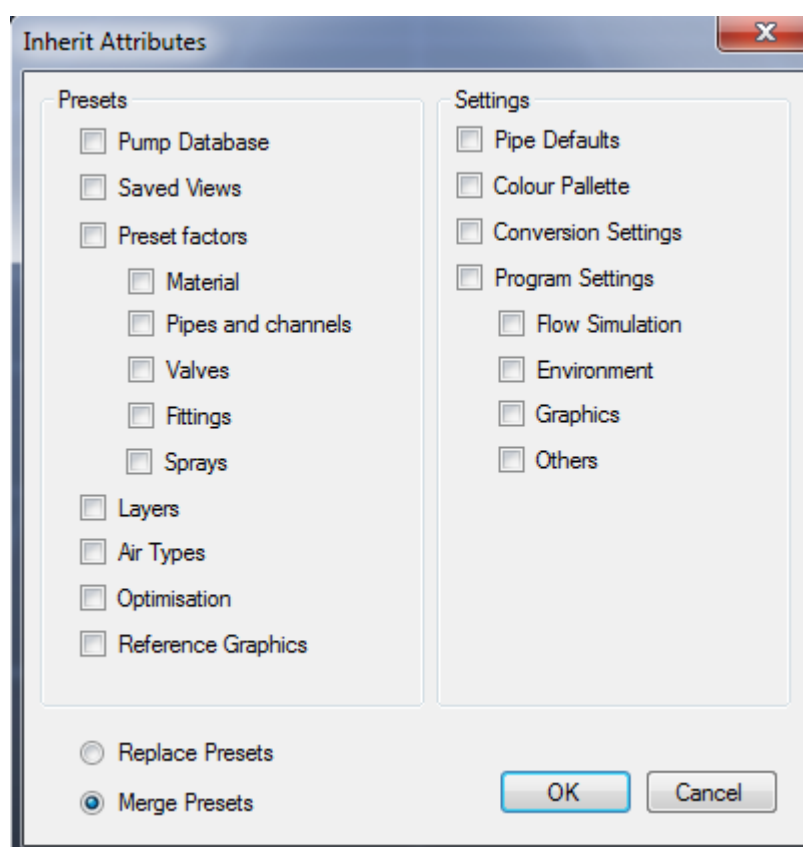


Fig. 2.1.2: The inherit attributes menu. Parameters of a model may be added to a Pumpsim™ model.

Hint: Pumps from other Pumpsim™ files can be used in your existing files by inheriting and 'combining' the pump databases. The simulation file pumps inherited will only be added if no existing pumps exist with the same name. The 'merge' option will ensure all the current model pumps will be preserved, and new pumps made available for selection. The pump list can be edited from the pump database, or from the Presets spreadsheet.

Once a suitable file is selected, an option panel is displayed to allow the user to select various components they wish to inherit from the saved file. Once loaded, these components will become part of the existing file. An option is provided to merge presets with existing presets (for example, the friction factors may be added to a list of currently used friction factors), or completely replace existing presets.

File Tools

A series of tools to assist with securing, comparing or linking pictures to Pumpsim™ files. The tools are described below.

- **Compare** – Compares the current loaded file with a saved file. Any obvious changes to pipes such as moving, deletion, new attributes, presets or sizes will be highlighted on screen and in the error list box.

Hint: This function may be useful where there may be multiple similar versions of the same file, and it is unknown what changes may have been made between versions.

- **Security** – Sets the file security with a password to prevent unauthorised changes or access. In addition, a password protected file cannot be merged with another. To activate security for a file, simply select this option, enter a password and click what security option you wish to apply.
- **View Only** – Allows the model to be loaded and viewed, but not changed in any way or re-saved.
- **Read Only** – Allows the model to be loaded and modified, but not resaved or copied to a new model.
- **Lock File** – Prevents the file from being opened or viewed.

When a security enabled file is loaded, Pumpsim™ will request a password. If the password is not entered or is incorrect, only the restricted security access option will be available.

Caution: Passwords are case sensitive. Ensure you can recall the password. A lost password means the file will not be able to be opened or changed in the future.

Import

Imports external data into Pumpsim™ to build model. Pumpsim™ can import data from TXT (text) files, DXF (drawing exchange format Autocad) files, DWG (Autocad native format), STR (Surpac string files), DM (Datamine Files – String or Wireframe Solid) and VDP (VnetPC™ files).

Export

Provides a utility to export Pumpsim™ centrelines, text and solid graphics to a DXF file for importing into other CAD software. The colours and text exported will be set to the current screen colours and text. Different items can be selected for exporting. They will be placed on different DXF layers so they can be turned on or off independently in the CAD program.

Note that any pipeline attributes will be lost during the export process, and DXF files cannot be re-imported back in to Pumpsim™ as pumping network models.

Manage References

Provides a utility to separately manage any imported graphics reference. Each imported object can be separately coloured, made transparent or hidden from the main view through the form options.

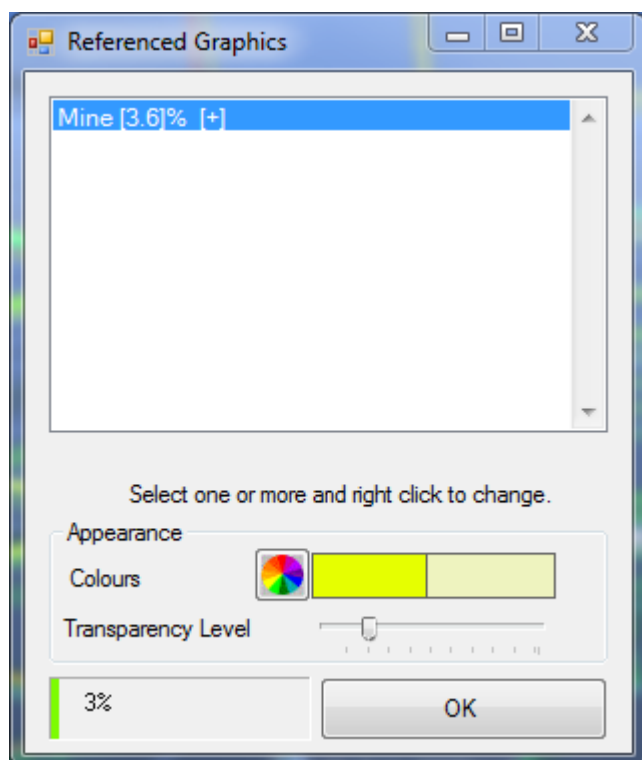


Fig. 2.1.3: The referenced graphics menu. Allows the user to reference different imported elements of a model.

Select one or more graphics objects from the list with the mouse (hold the **SHIFT** key down to select multiple items). A **[+]** against the name indicates it is currently being shown. A **[-]** indicates it is currently hidden.

- **Colour** – Changes the colour of the items. The second colour shows the transparency effect on the colour. Clicking the multi-coloured box will colour the reference graphics object with a full spectrum of colours based on elevation.
- **Transparency** – Changes the transparency of the item.
- **Memory** – Shows the current internal Pumpsim™ memory reserved for storing reference graphics. If this approaches 100%, consider removing some reference graphics objects, or alternatively if the computer has sufficient memory increase the Reference Graphics memory from the **Settings > Settings** menu.

Right click on a reference to open other options:

- **Hide** – Hides the object from the screen view
- **Show** – Shows the object in the screen view
- **Merge** – Joins together two or more selected files from the reference graphic list.
- **Delete** – Removes the reference object from the file

- **Remove Duplicates** – Search all reference graphics for duplicate graphics and removes any duplicates to reduce memory requirements. This function does not remove duplicates on different reference layers.

Title Note

Allows the user to specify a unique file comment which appears in the top title bar. This comment can help identify the date, name and purpose of the model.

File Memo

Allows users to write an extensive descriptor regarding the function or description of the current model. This is saved with the file for future reference.

Print Image/Print Vector

Prints a graphic picture of the model in the View Window to an installed printer.

Only printers with Windows supported graphics capabilities will be capable of printing the model. As Pumpsim™ uses a perspective view, no particular scale is applied to the image. The image is sized to the maximum size of the page and orientation. To reduce colour output, the Pumpsim™ screen colours can be changed under the **Settings > Settings > Graphics > Colours** menu. For detailed engineering accuracy output, it is recommended to export the model to a DXF file for loading into a suitable CAD program for later output to a printer.

Load Demonstration

Pumpsim™ comes pre-loaded with some generic models of typical mines. These models is for demonstration purposes only, and contains a number of simple site examples to view. Note that while these items can be modified, **they cannot be resaved in the same folder.**

Licence Manager

Opens the **License Manager** form.

Exit

This command closes Pumpsim™. Pumpsim™ will prompt if your model file, pump database or defaults have not been saved since last modified.

"Edit" Menu

Undo

Reverses the previous action. **Undo** is a fully functional undo facility that will undo a number of previous

changes made to a model (up to the buffer size of the undo function). Note that while it will not directly undo a simulation, by pressing **UNDO** until your previously model is in place and then re-simulating, this will produce the same results.

Redo

Redo will reverse the result of the **undo** function.

Copy and Paste

- Copy pipes from one Pumpsim™ program to another.
- Copy and Paste objects creates an exact replica of selected existing objects in a model and pastes them into the same location into the existing or new model. The function is primarily designed to copy and paste pipes between Pumpsim™ files or between different **stages** in a pumping network design; for example to update pipes from a model that has been modified.
- To use this function to copy and paste between models, ideally have two copies of Pumpsim™ open with different model files. Select the **Copy** menu item, and click on, or fence the pipes you wish to copy.
- To paste the pipes, move to the new model (or load it up) and select the **Paste TRUE** menu item. The copied pipes will be pasted into the new model at the same coordinates as the original pipes.
- To paste duplicate objects between **Stages**, use the same technique, but simply copy the object in one stage, switch to the desired stage name or number, then **Paste pipes TRUE**. A copy of the pipes will be set in the new stage.
- If pipes become duplicated in the process, Pumpsim™ will delete one of the duplicate copies.
- To paste objects in a different location, use the **Paste pipes LOCAL** menu option. This will paste the pipes around the current local screen location set in the view window, effectively copying the pipes from the original location to a new location.

Clone and Apply Attributes

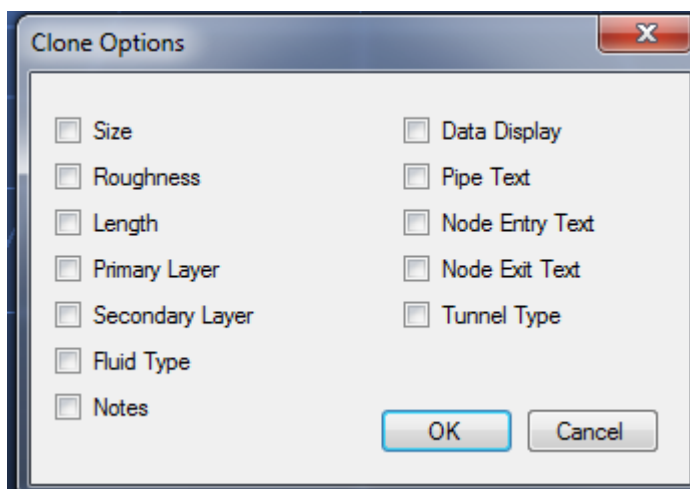


Fig. 2.1.4: The Clone options window.

Attributes are physical pipe parameters such as size, friction factors and shock losses, as well as identifying attributes such as layers. You can decide which attributes to apply to an existing pipe by selecting the *Clone Options* function from the menu or *Select Manager Form* from the toolbar.

To clone attributes, select the **Clone Attribute** menu item, and click on an existing pipe. The pipe properties will be copied to memory and the program will automatically enter the **Apply Attribute** mode.

To apply attributes, make sure you are in the **Apply Attribute** mode if it has not already been selected and either click on an existing pipe, OR fence a selection of pipes to apply the attributes to that group. **Cloned Attributes** can be applied at anytime (even after other editing has been done) and will apply the most recently cloned values.

Example: An existing pipe has been set as an exhaust pipe with custom primary and secondary layers. To copy these attributes on to other pipes,

- Select **Primary, Secondary** and **Fluid Type** options on the **Select Box**
- Select **Edit > Copy Attributes**
- Click the pipe you wish to copy the attributes from.
- Select **Edit > Paste Attributes**
- Click or Fence the pipes you wish to copy the attributes to.

New Pipes

Control which attributes are applied to new pipes.

- **Use Inherited** – New pipes drawn from existing pipes will inherit the settings from the pipes they are attached to (such as size, shape, friction factors etc). pipes which do not originate from another pipe will use the default setting values.
- **Use Defaults** – To force the program to use the Default Values (from the Settings Menu), choose Default. All new pipes drawn (regardless of whether they connect from existing pipes) will use the default settings.
- **Use Cloned** – Forces the program to use the Cloned Settings from a previously cloned pipe. For example if a pipe is to be drawn from a horizontal pipe, cloning a similar pipe and then drawing the new pipe with this function set, will result in a new shaft with the same size, parameters and layer settings as the cloned pipe.

Find / Find Next / Find ALL

Automatically locates specific pipes and moves the screen to selected parts of a model.

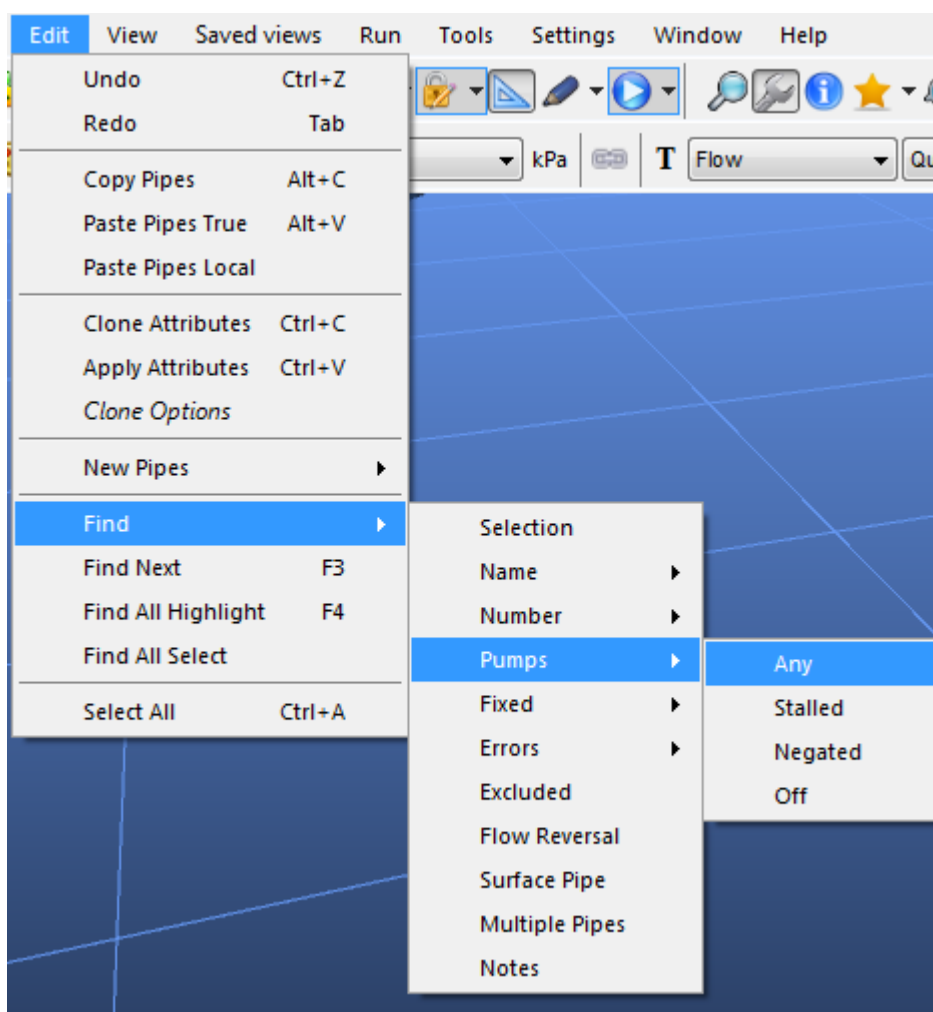


Fig. 2.1.5: The find command, as accessed through the edit tab.

Selecting one of the options will find and highlight the object searched for. The option may be repeated for further objects by pressing the **Find Next** menu function, the **Find** tool bar button or by pressing **<F3>** on the keyboard.

Hint: Pressing **<F3>** will rapidly allow you to search through a model, by repeatedly jumping to the next item. To view more detail about the items being jumped to, leave the **EDIT BOX** open. The **EDIT BOX** will instantly update with detail about every pipe that is jumped to.

Highlight or Select All

Highlights or selects all objects matching the previous find criteria. To find all instances of an object (for example to find all objects with the name 'shaft' in the pipe name, or to find all pumps, click on an initial find type and then click on **Find All**. All items with these parameters will highlight and flash.

For example, if **FIND** was used to locate an object with the name "M52 Decline", **Select All** would select ALL objects with this name in the pipe name field. The selected pipes could then be edited or manipulated as a group.

"View" Menu

Fit All

Fits all the display data into the view window.

CAUTION: *If display data is from two regions a long way apart (for example data may have been imported into Pumpsim™ from a different coordinate system). **Fit All** may not be able to accommodate the range of data attempted to be shown, or the camera may be too far away to view the data effectively. Ensure all data is in the same coordinate region before it is loaded or merged.*

Show All

Resets all hidden levels, layers and data and displays the entire model. Occasionally a model may have hidden or semi-transparent pipes set from a previous action. **Show All** will quickly bring the entire model back to full view. **Show All** will force Pumpsim™ to show all elevations of data in your model, even if they are outside of the ranges specified in the Level Database.

Reset Display

Resets the graphics display adaptor and re-establishes the graphics on the screen. Some types of hardware graphics display adaptors may occasionally corrupt or fail to show the screen graphics, particularly if the computer has been brought out of hibernation, sleep or screen saver. This option in most cases should recover the graphics.

Quick View

Quick Views can quickly be saved and recalled sequentially outside of the normal **Saved View** system. **Quick Views** are not added to the **Save View** menu selection. The views can be quickly recalled at any time using the arrow keys. The function primarily assists with navigation of large models, where different areas can be quickly returned to for viewing simulation results.

- **Save Quick View** – Creates a temporary view position for a model. The view is added sequentially to previous stored **Quick Views**. The sequence of stored quick view can be recalled or returned to using the **LEFT** and **RIGHT** arrow keys.
- **Clear Current** – Clears and removed the currently recalled quick view. The view is removed from the sequence of other stored **Quick Views**, and can no longer be recalled.
- **Clear ALL** – Clears all **Quick Views** from memory.
- **Previous Quick View** – Recall the previous **Quick View** from the sequence of saved **Quick Views**.
- **Next Quick View** – Recalls the next **Quick View** in the sequence of saved **Quick Views**.

Copy to Clipboard

Copies the screen to the Windows clipboard for direct pasting into an external documents (such as a PowerPoint presentation). Static Views are primarily for referencing areas before and after simulation.

Copy to Clipboard (all)

Similar to above, but copies any overlying windows (such as legends and graphs).

Copy HI-RES

Similar to above, but copies an **ultra-high resolution picture** to the clipboard. This picture has more detail than the screen graphics, but may fail to work on slower or older computers. Hi-Res pictures will show more clearly in large format printing or reports, but are of less use in screen or projector presentations where the screen cannot show the higher resolutions.

Snapshot

Creates a picture copy of the current window in a new form window. Static Views are primarily for referencing areas before and after simulation. It may be useful in showing a sequence of changing pumping simulation results by storing previous results for comparison with new results.

The form window defaults to 25% of the size of the normal view window, but can be resized or maximised at any time to show full detail. The static view forms can be renamed for reference or saved to a picture JPG file for future use in other software.

Static views are simply picture copies of the original model graphics and remain the same regardless what happens to a model after the static view is created. They cannot be edited, moved or rotated. The number of static views created is only limited by the memory capacity of the computer.

Hint: Both the EDIT box and the pump DATABASE form can also be reproduced as a STATIC view. This is useful to provide a quick comparison between pipes or model results before and after simulation changes. To use this function, simply RIGHT click the EDIT box or pump form, and select Static View.

Set Edit Centre

Sets the edit grid and point of focus to a specified coordinate or elevation. New pipes will be drawn on the specified elevation.

Show All Elevations

Shows all elevation ranges for the model. Turns on all level elevations views, and includes pipes and references outside of the defined elevation ranges.

Show All Layers

Shows all layers (primary and secondary) in the model. Turns on all layers so all pipe graphics will be visible.

Hide Zero Flow

Hides pipes with no flow. This function is useful for hiding disused or non-flowing parts of a site so they do not clutter the display. Pipes with no flow (or flow below the level set as zero flow value in the **Settings** menu) will be shown as transparent or hidden, depending on the transparency settings described above, and the transparency amount set in the **Display Manager**.

Hide Excluded

Hides pipes which have been set as **EXCLUDED** from the model. This function is useful for hiding parts of a site which are not required to be simulated, or are not set to be part of the current simulation model. The exclude options for pipes are available from the **EDIT** box.

***Hint:** An old sealed off development, or a future development yet to be mined can be excluded from the model to speed up simulation and display of models. Excluded pipe can be hidden to simplify the display, but can be shown and converted back to normal pipes at any time if required.*

"Saved Views" Menu

Save New

Saves the current view and stores the save name in menu for future recall. Save view will save all attributes in a view including RL levels, layers and display options. Saved views can be pulled back into any current window by selecting the saved name from the pull down view list.

Delete View

Deletes the saved view stored in the **CURRENT** pull down menu list (and hence removes this name from the list).

Saved Views

Four view orientations are set as defaults:

- **PLAN**
- **EW SECTION**
- **NS SECTION**

- **ISO**

These standard views will display the model at various orientations, although the model can still be orientated to the same orientations by using the **RIGHT** mouse button. These views cannot be deleted or changed.

Caution: *The Perspective view will distort some aspect of the views. For example, the plan view will show the pipes at the EDIT grid level to plan, but pipes above and below this elevation will appear larger and smaller respectively.*

Any further saved views will be placed below these items. **Saved Views** save the position, orientation, colour scheme, data types and attributes of the screen at the time at which they were saved.

Hint: *Saved views are not only useful for recalling the location and orientation of a model. Because they recall selected levels, layers, data types and colours, they can be a quick way to establish a template to edit and view different data aspects of your model. For example, you may have an FLOW view, custom set to highlight a certain range of flows in different colours, while a 'HEAT' view may be saved to highlight a range of temperatures.*

"Run" Menu

Flows

Undertakes a steady state incompressible flow simulation of the model.

Summary

Displays a summary of the current model collectively, or grouped under different tabs. The data may be copied to the clipboard, for pasting into another package such as Microsoft Word, or an email.

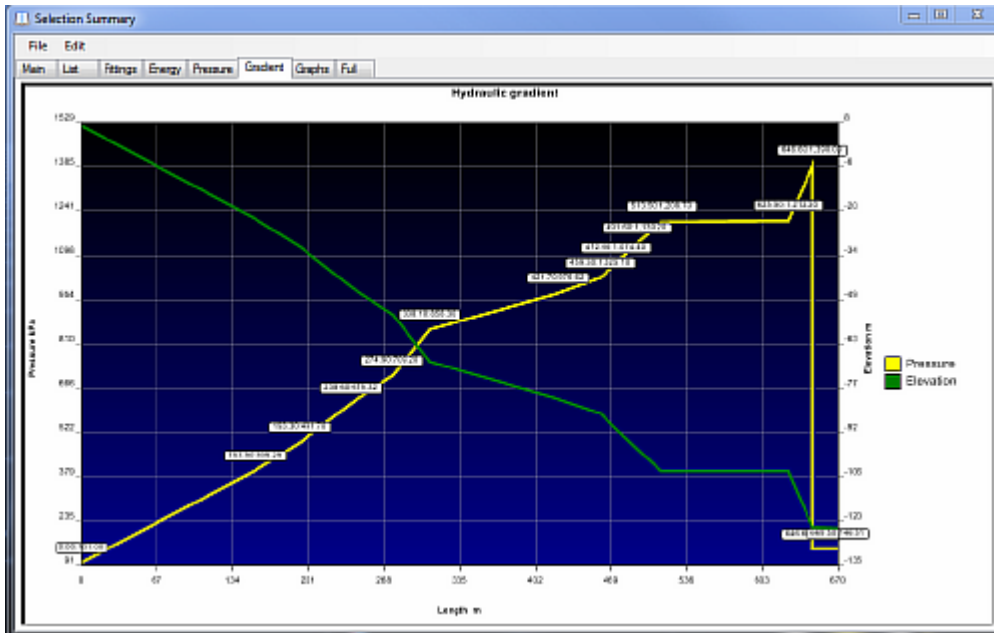


Fig. 2.1.6: The Gradient tab of a Selection Summary window.

- **Main** – General information about the model
- **List** – List of all elements in the model.
- **Cost** – Calculate the cost of installation and power in the selected part.
- **Fittings** – Information about pumps, valves and other fittings
- **Energy** – Energy summary
- **Pressure** – Pressure summary
- **Gradient** – Only available if a line is selected. Shows the hydraulic gradient. The higher point will be on the left.
- **Graphs** – Energy loss graphics

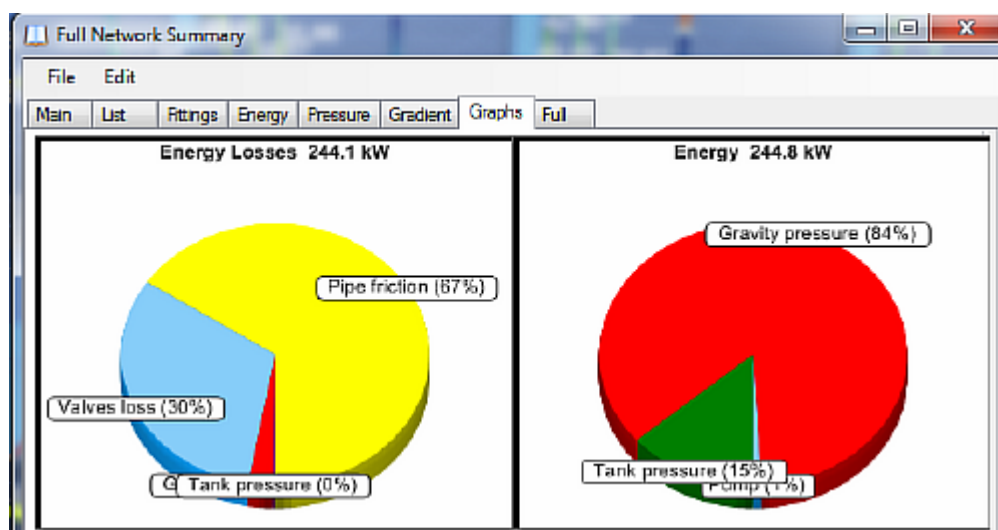


Fig. 2.1.7: The graphs tab demonstrating energy loss graphics of a network summary.

"Tools" Menu

A selection of tools and settings to check, modify and fine tune the model.

Pumps

Displays a windows form that allows editing, adding and deleting of all pumps in the model pump database. Up to one thousand (1000) pumps and the associated pump curves may be entered into the pump database. A display for each pump curve and data will be presented when a pump is selected from the display list.

For further information on entering and using pumps, see the pump section in this manual.

Levels

- A list of elevation ranges on which pipe data can be individually viewed.
- This form allows editing or creating of a list of levels (elevations) between which pipe data is located. The level list can contain up to 1000 levels on which your model will be created.
- Elevation data can be edited in any order, as well as added at a later date - Pumpsim™ will sort the data from highest to lowest when next displayed.
- To select which levels are displayed, use the **Data Display** manager to select what levels to display. Alternatively, levels can also be independently set by using the mouse context pop-up menu (right mouse button on the screen) to **Select Levels**.
- To view ALL levels at any time, simply use the **Right Click** pop-up menu to select **All Levels**. Levels can be sorted by clicking the column header for the desired item.

Spreadsheet

- Displays a **READ-ONLY** list of currently viewed or selected pipes in a spreadsheet form.
- The spreadsheet of pipe data displayed can be copied and pasted to other applications such as Microsoft Excel or Word.
- Up to 100 different types of data can be displayed on the spreadsheet. To display different data types, use the **SELECT > DATA OPTIONS** menu to select the data you wish to display.
- Data can also be removed from the spreadsheet, by selecting **EDIT > REMOVE** or using the right mouse button.
- Data columns can be reordered by selecting the column title and dragging to a new location. In addition columns can be resized and resorted by using the appropriate menu commands.

***Hint:** While the spreadsheet function is not designed for editing data, extensive data characteristics for each pipe can be displayed, and copied to external programs such as Microsoft Excel™. Pumpsim™ provides an Index number and a Unique number reference for each pipe. The Index number is referenced internally and during simulations by Pumpsim™, but may change for each pipe as pipes are added or removed. The Unique number will not change, and should be used as a permanent reference number to each pipe if other identifiers such as names are not used.*

Filtering Tools

Filtering Tools consist of a selection of pipe utilities to help filter and correct complex, disconnected or overlapping raw model data. The tools can be run simultaneously as a group, or individually.

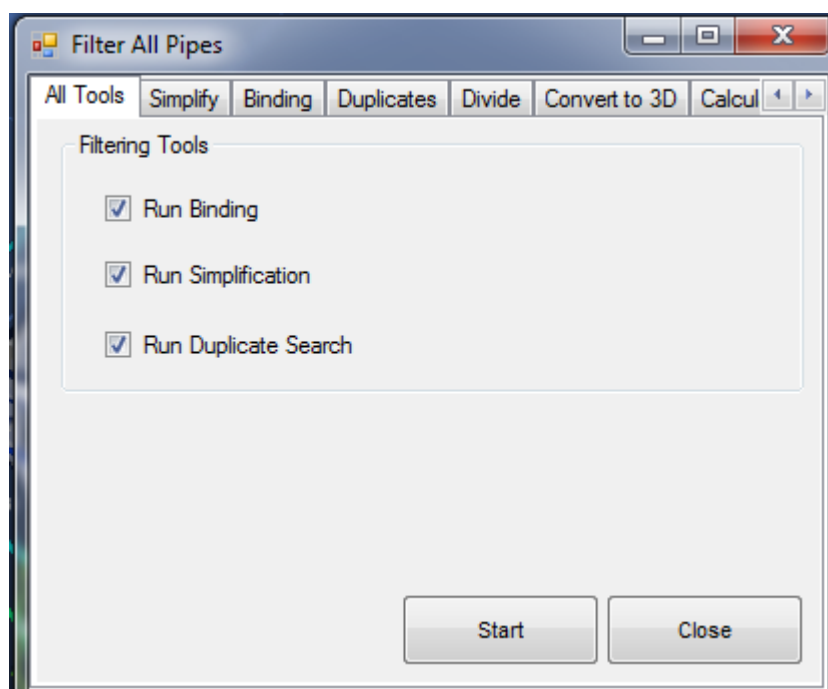


Fig. 2.1.8: The Filter window, used to filter and correct raw model data of pipes.

The **"All Tools"** tab in the form, allows simultaneous filtering using the various selected tools. Alternatively, each tool can be selected and run individually from the other tabs.

Simplify

Simplify allows the user to reduce the number of pipes in a model to a more efficient number without affecting the overall model analysis.

It is particularly useful where a model has been imported via a DXF file and contains a large number of very detailed but unnecessary small connected pipes.

Simplify will search a model for sequences of pipes that may be reduced to single straight pipes. In doing so, much of the overhead and effort required to set parameters to every pipe can be reduced. Note that the simplify function will only combine pipes that have a single entry and exit. Pipes at junctions and split objects will remain untouched.

- **Maximum Joined Angle** – The **Maximum Joined Angle** option specifies the maximum angle between two pipes being considered for merging. For example, if the original model contains a number of pipes that form a curve and **Maximum Joined Angle** is set to 20°, **Simplify** will keep on merging pipes until the pipes being considered have directions changed by 20° or more.
- **Maximum Joined Length** – The **Maximum Joined Length** option restricts simplification to pipes less than the specified length. For example if set to 30m, only pipes with lengths of less than 30m will be considered for merging and simplifying.
- **Use combined lengths** – Ensure the new length of a combined pipe is fixed to remain exactly the same as the original combined lengths even if the original pipes formed a curve. If not set, the new lengths will be recalculated as the true length of the new straight line. In most cases the difference is minor, and it is recommended to leave this option unchecked.
- **Ensure no resistance presets filtered** – Ensures that pipes with a preset resistance (such as a block pipe) are not simplified and merged with other pipes.
- **Ensure pipes dimensions are the same** – Check to see if pipes that are considered for merging and simplifying are only joined if they are exactly the same size.
- **Ensure wall types are the same** – Similar to dimensions, only pipes with similar wall (friction factor) types are joined.
- **Selected pipes only** – Only simplify selected pipes and ignore all others.

Note: In general, the higher the values of the **Maximum Joined Angle** and **Maximum Joined Length** settings, the more aggressively Pumpsim™ will simplify and combine pipes.

Hint: Short irregular pipe segments are best simplified with high joined angle settings (+45 degrees) or smaller joined length settings (<20m). Long smooth pipe segments are best simplified with low joined angle settings (<20 degrees) and high joined length settings (>50m). Note that pipes can be individually selected for simplification.

Binding

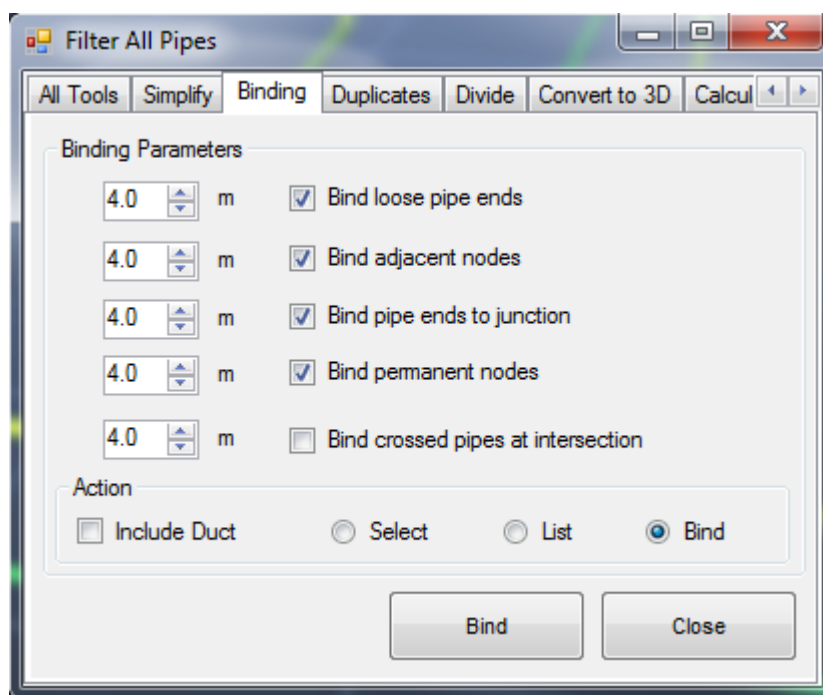


Fig. 2.1.9: The Binding tab of the Filter window.

Binding connects disconnected pipes ends or intersections. Many imported DXF files do not have correctly connected centre lines or ends. This function will search for unconnected ends or crossed pipes and join them together if they are close enough. Pipes must always be correctly joined to allow flow.

Warning: These functions are intended for preliminary clean-up of new models, not for existing balanced models. It can permanently change the characteristics of your model and may delete some preset items or values. Ensure only the required pipes are first SELECTED before binding or simplification.

- **Bind loose pipe ends** – Defines the distance to join pipe ends in a model which are close together but do not join. Imported DXF file commonly have lines which don't exactly join due to editing or drawing methods used in the original file. Pipes which do not exactly join in Pumpsim™ will not flow, and will result in a No Exit / Entry warning. The search radius can be entered in the preset box. For example a search radius of 4 will join any loose end (pipe end without another connection) to any pipes within 4m of the end.
- **Bind adjacent nodes** – Defines the distance to join adjacent connected nodes in a model which are close together but do not join. This function is a little like simplify, in that small pipes between the joined nodes will be removed.

Warning: Selecting this option with too greater distance can seriously distort the model. Use with caution or not at all.

- **Bind pipe ends to Junction** – Defines the distance to join pipe ends to other pipe mid sections. It effectively splits and joins into the new pipe. Imported DXF files commonly have lines which cross the path of pipe ends, but do not have a joining node. This function will create a joining node along the line.
- **Bind Permanent Node** – Defines the distance for joining fitting. It effectively splits and joins into the

new pipe. This function will create a joining node along the line.

- **Bind Crossed pipes at Intersection** – Defines the distance for joining pipes that intersect or come close to intersecting, but do not have a joining node. It effectively splits and joins both pipes into new pipes. Imported DXF file commonly have lines which cross the path of another pipe, but do not have a joining node.
- **Duplicates** – Searches for pipes which have duplicates in the same or similar position. Duplicate pipes can cause problems with simulation due to poorly defined or hidden flow paths that the user cannot see.

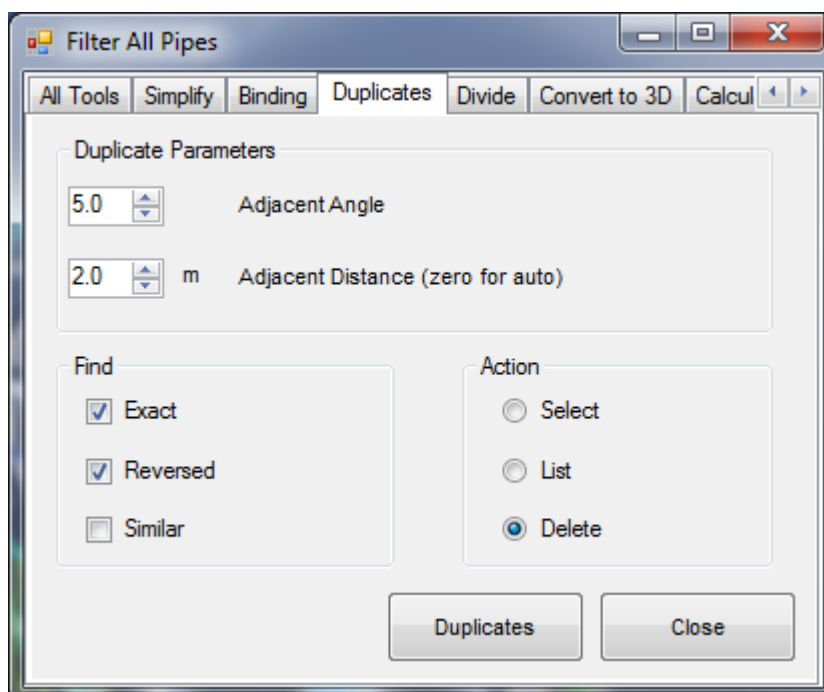


Fig. 2.1.10: The Duplicates tab of the Filter window.

- **Adjacent Angle** is the maximum angle difference between adjacent pipes that may be considered as a duplicate. Pipes next to each other with greater angles between them will not be considered as a duplicate. The smaller the angle in degrees, the more 'parallel' and pipe must be to be considered a duplicate.
- **Adjacent Distance** is the maximum distance two adjacent pipes can be apart over which they will no longer be considered as duplicates.
- **Exact** selects only an exact matching pipe as a duplicate
- **Reversed** selects pipe the same, but facing opposite directions.
- **Similar** selects pipes that are similar but not identical. The adjacent angle and distance will be used as a criterion for selection.
- **Action** defines what action will be taken when the duplicate pipes are found.

Convert To 3D

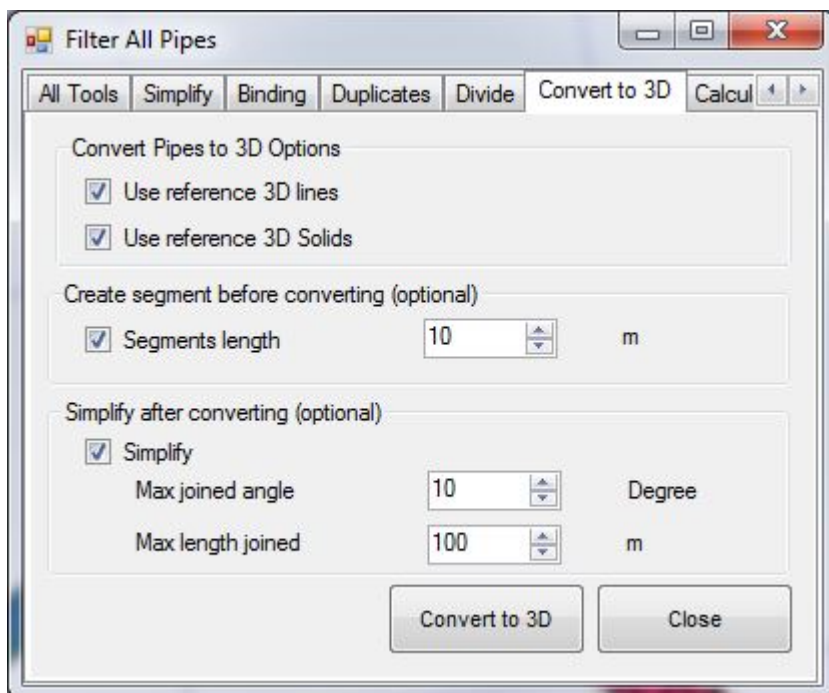


Fig. 2.1.11: The Convert to 3D tab of the Filter window.

Moves the model pipes on to a 3D surface or contour plan, effectively converting the model from a 2D plan to a 3D model. Note that if only parts of a model need to be converted, they should first be selected with the Select tool.

- **Use Reference 3D Lines** – Uses lines (such as contours) loaded in from a DXF as a reference to convert the model to 3D.
- **Use Reference 3D Solids** – Uses solids (such a terrain polygons) loaded in as a referenced DXF object, to convert the pipes to 3D,
- **Create segment** – Break the pipes in small section in order to obtain a better adjustment.
- **Simplify** – Join the pipe together after adjustment in order to decrease the number of pipe.

Hint: To convert a multi-seam or level model to 3D using this function, individually group-select the pipes required, and ensure only the relevant 3D DXF object for that level is turned on before conversion.

Utilities

The **Utilities** sub menu provides many functions to assist with the manipulation of pipe data, particularly raw data which may have been imported from a DXF file or similar.

Swap Axis

Swaps the **Easting** and **Northing** (X and Y) coordinates of all pipes in the model. This may be useful where the coordinate system in Pumpsim™ does not match the directions used on the **Site Plan Grid**. Where the direction of the coordinates needs to be reversed, use the **Settings > COORDINATES** settings to change the direction of the axis coordinates.

Rotate Coordinates

A utility which rotates all (if none are selected) or just the selected pipes by a number of degree around the point of focus. The point of focus is always at the centre of the screen. It can be set by middle clicking the mouse button on a point, or by entering the centre coordinates in the **View > Set Edit Centre** option.

Reverse Coordinates

Changes the sign of pipe coordinates from positive to negative or visa-versa. Not that this is similar to mirroring coordinates around a zero centre.

Scale Coordinates

A utility which scales all pipe or reference coordinates by an input factor. This effectively reduces or increases the size of the model without changing the pipe size. A pipe length will not be changed if it has been fixed.

Warning: *This function should only be used to scale schematic pumping model with predefined fixed pipe lengths. Scaling a model without fixed length will change the pipe length and resistance, making it invalid for true scale model simulation.*

Scale Pipe Size

A utility which scales all pipe dimensions by an input factor. This effectively reduces or increases the size of the pipes in the model.

Hint: *This function will recalculate pipes lengths and hence change any previously fixed lengths. If there are any doubts, use the EDIT function to individually fix/unfix lengths of only the pipes that need to be changed.*

Pipes

Auto Name

Automatically places a code number on every pipe without a current pipe name. A letter may be specified to head the number if desired (e.g. B157).

Resequence Index/Unique Numbers

Sequences the order of pipe numbers in a Pumpsim™ model.

- **Index numbers** are dynamic numbers representing pipes stored internally in Pumpsim™. The numbers may change as pipes are deleted or added. Pumpsim™ uses index numbers to refer to pipes during simulation and when identifying problems. All index numbers are sequential and the highest index number will be the sum of all pipes. Resequencing index numbers rennumbers all pipes from higher elevation to lowest elevation, easting to northing.
- **Unique numbers** are static numbers which do not change when pipes are deleted or added. Numbers are sequenced incrementally from the last highest unique number. As such, as pipes are added and deleted, there may be large gaps of numbers between pipes. Unique numbers are a useful reference to identify the same pipe as the model is developed. Resequencing the unique numbers rennumbers all pipes from number 1, sequentially top to bottom to the last pipe.

Hint: Resequencing numbers is not a requirement in Pumpsim™, but may simplify tracking pipes, as the resequence function tends to group pipes in similar locations with similar numbers. This is particular handy in the SPREADSHEET function which initially lists pipes in sequential index order.

Reset Network

Removes all flow and temperatures from a model. The simulation processes is restarted with fresh data. Pumpsim™ utilises flows, densities and temperatures from previous simulation to help it to recalculate new simulations faster and more accurately. If data has been corrupted during a bad simulation (which may have produced errors, excessive flows, heat or densities), this may hinder subsequent simulations from finding an acceptable solution, or may simply produce further errors.

Resetting the model will remove all calculation flows, pressures, densities and heat. It will NOT remove any values fixed by the user, such as fixed flows, pressures or heat sources.

"Settings" Menu

Presets

A list of preset factors and characteristics used for placing in pipes.

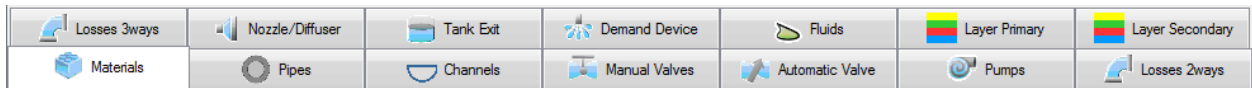


Fig. 2.1.10: The Duplicates tab of the Filter window.

Presets include:

- Materials
- Pipes
- Channels
- Layers
- Fluid Types
- Pumps (limited to header information only)
- Valves
- Fittings
- And more

Values entered into the preset spreadsheet will be made available for applying to pipes in the **EDIT** box. Updating existing values in the preset spreadsheet will automatically change all pipes with that preset on the next simulation.

[See the **PRESETS** section for further information](#)

Warning: Be careful when removing existing settings from the preset spreadsheet. Settings removed which already belong to an existing pipe, will force the pipe to 'set' the values permanently.

Units

Sets the units used for Pumpsim™.

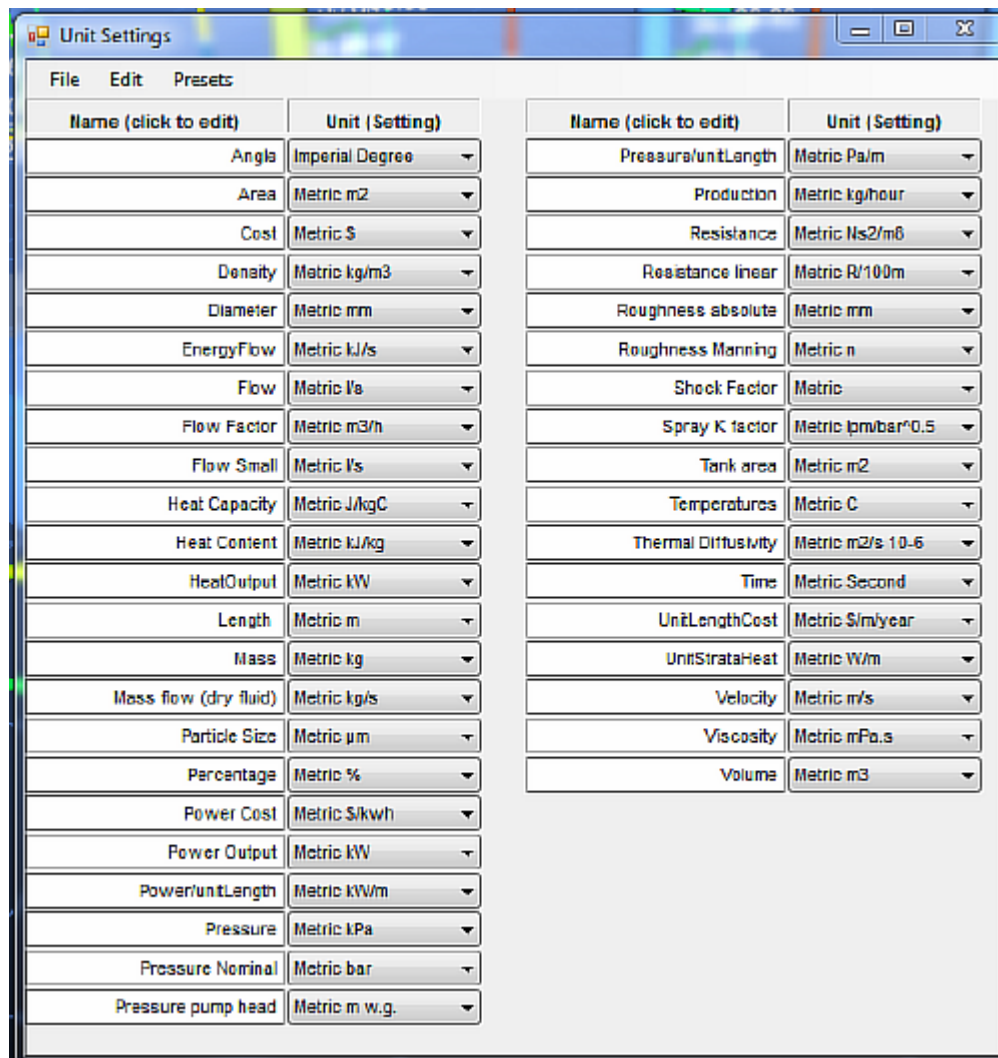


Fig. 2.1.12: The Unit Settings window.

To add or modify units, click on the name of the data (flow for example).

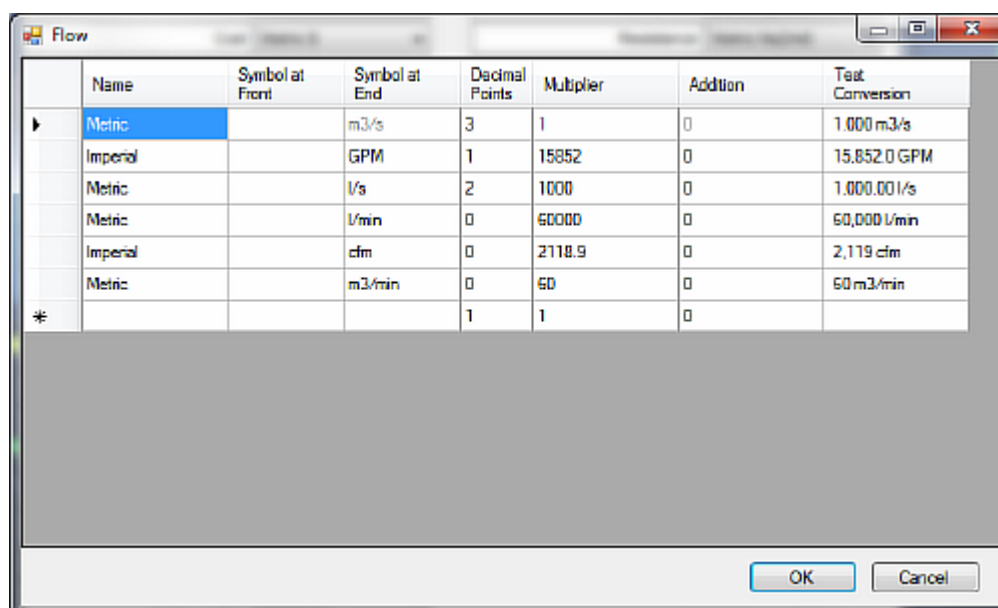


Fig. 2.1.13: The Flow units, accessed through the Unit Settings window.

Settings List

An extensive list of settings used by Pumpsim™ that control the behaviour, simulation parameters and visual appearance of the program.

[For further information on settings, go to the **Settings** sections.](#)

"Windows" Menu

Fit All

Fits all display data on the screen.

Zoom Out

Increases the distance away from the window point of focus. This may be of use if the mouse used does not have a scroll wheel.

Tile

Enables windows to automatically arrange to fill the master window form. Note that only docked (windows within the master window) will be affected.

- **Vertical** – Tiles all open windows in a vertical direction with equal spacing
- **Horizontal** – Tiles all open windows in a horizontal direction with equal spacing

- **Tiled** – Tiles all open windows in square or tiles to fill the master window display.

Auto Arrange

Enables windows to automatically dock and arrange the window position within the master window. Windows that are dragged into the master window will be automatically arranged when this function is activated.

2.2 The View Toolbar

The **VIEW** toolbar adjusts the display of items shown on the screen.

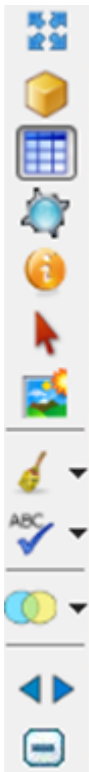


Fig. 2.2.1: The **VIEW** toolbar.

Fit All

Fits available graphics centrally on the screen. If graphics are spread over a huge area, then Pumpsim™ may elect to only show the pipes in the most prominent area.



Solid/Wireframe

Switches the model between wireframe and solid display mode. The wireframe mode is sometimes useful for fine editing of models, particularly when pipes are large or very close together.



Grid

Displays or hides the main grid. Note that in a section (vertical) view, the grid automatically changes to show elevations.



Nodes

Displays or hides nodes (junctions) of pipes. This option can also show or hide individual types of icons by clicking the sub option drop down arrow to the right of the button.



Arrows

Display or hides arrows showing flows.



References

Displays or hides reference graphics (for example DXF graphics or data imported into Pumpsim™).



Limit Data

Provides options for limiting the amount of data displayed in a model. For complex models, displaying data for every single pipe segment can lead to a cluttered display and is often unnecessary, particularly if the data shown is largely the same (for example flows). However, for other items this large amount of text data can still be useful.

Limit Off:

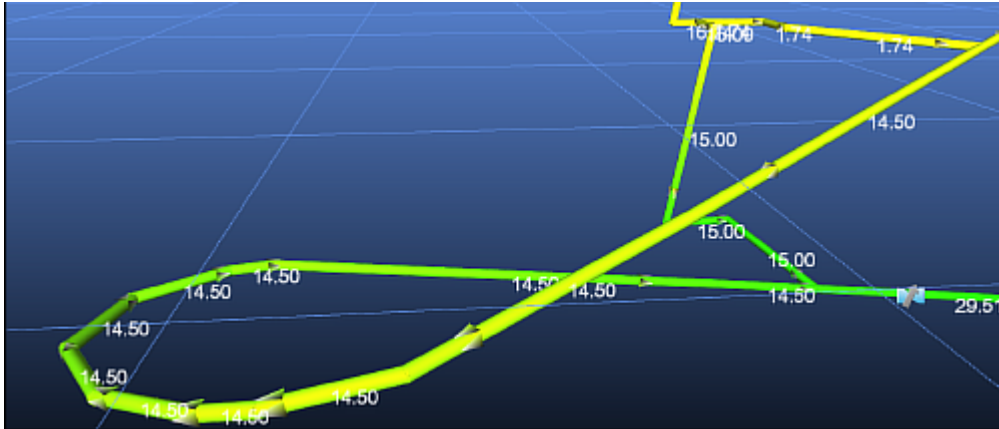


Fig. 2.2.2: A section of a pipe shaft model with the Limit command off, thus displaying all data text.

Limit On:

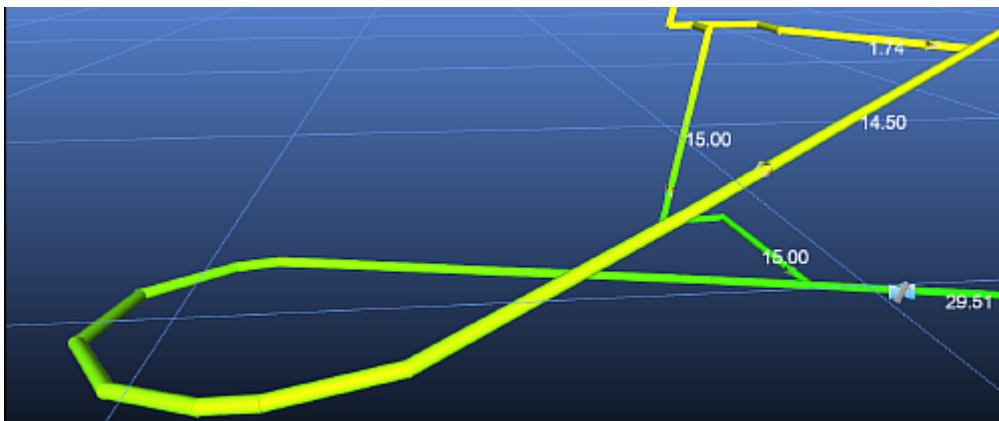


Fig. 2.2.3: The same section of pipe shaft as Fig 2.2.2, with the Limit command on. The model has less data text, with not all segments display data.

Hint: For data which may vary from one end of a pipe to another (such as pressure), Pumpsim™ will always show the ENTRY data on the pipe unless otherwise specified.

Limit All

Only displays text, nodes and/or arrows on pipes which specifically have 'Show Data' option set in the **EDIT** box. For example, a loop in a decline consisting of 10 pipe segments will normally show data for every segment. To only show one text data for all the segments, ensure only one pipe is set to 'Show Data', and the activated the LIMITED > text display. To force a pipe segment to show data, use the **EDIT** box and click '**Always Show Data**'.

Limit Off

Shows all pipe data, regardless of whether the 'Show Data' option is set in the EDIT box.



Text

Controls how much text information is displayed on the screen.

All On

Displays text information for data values, node names, pipe names and error messages.

All Off

Hides all text information.

Data, Pipes, Nodes, Grid, Log Values

Text data, pipe names or node names can be individually turned on or off by selecting them from the view menu.



Transparency

By default, all data which is non-active (for example levels or layers which are turned off, data which is outside of the **Display Manager**, or zero flows if set) are shown as a transparency, with the transparency amount set in the **Display Manager**.

To specifically hide some items (for example layers which are not visible), but still show transparencies for other items (for example **Zero Flows** or **Coloured Data**), data types can be individually set to show transparency or be hidden.

- **Data** – coloured data which is outside of the selection range in the **Display Manager**, and is selected to be hidden can be made transparent or removed from the display.
- **Levels** – level elevations outside of the selected ranges can be made transparent or hidden.
- **Layers (primary and secondary)** – layers that are not selected can be made transparent or hidden.
- **Zero Flows** – pipes carrying no flow can be made transparent or hidden.



Move Toolbar

Moves the toolbar to the left or right of the screen window.



Hide Toolbar

Hides the toolbar from the window. To show the toolbar again, **RIGHT CLICK** the mouse anywhere on the screen and select '**View Toolbar**'.

2.3 The Action Toolbar

The toolbar allows quick access to commonly used Pumpsim™ functions.

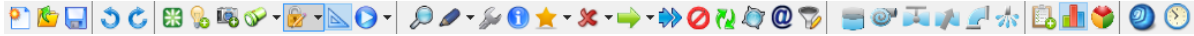


Fig. 2.3.1: The quick-access toolbar.

File and Utility Functions



Fig. 2.3.2: The first segment of the toolbar, the file and utility functions.

New File

Creates a new file. The current file will be cleared.

Open File

Open the file dialog to load a new file.

Save File

Saves the existing file. If no name has yet been set, a Save File dialog will be displayed.

Reset Display

Reset the graphics card if the display becomes corrupted.

Undo

Reverses the last action performed in Pumpsim™.

Show All

Automatically turns on all hidden pipes to ensure all pipes in the model can be seen.

Snapshot

Takes a static image of the screen in a form which may be saved, copied or remain on the screen for further comparison with other windows. Snapshots are static only and do not change.

Find

Searches for specific items within a site model. Pressing the 'searchlight' icon directly, repeats the previous find action. Pressing the submenu arrow next to the icon, activates the find submenu as shown below.

Some examples include:

- **Name** – Find the name of a pipe name or node. Entering any part of the name will find pipes containing the name part.
- **Number** – Find a pipe with the specified index or unique number
- **Pumps** – Find pumps, pumps that are turned off, or pumps that are performing outside of their rated pump curves.
- **Fixed** – Find fixed flows or pressures.
- **Errors** – Find model errors defined during the previous simulation.

Lock

The lock function controls how the Pumpsim™ cursor attaches to other pipes or graphics objects. When editing, this is often desirable to ensure pipes attach correctly to other pipes or reference graphics while drawing or editing, however it may become problematic particularly in graphically crowded environments.

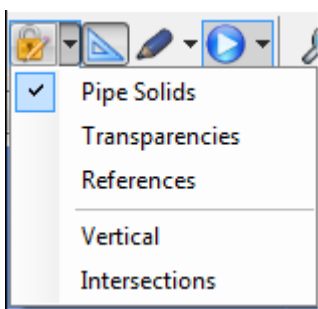


Fig 2.3.3: Lockable parts of the model, available from the lock icon on the toolbar.

Individual graphical locks can be turned off using the function below. Note that **ALL LOCKS CAN BE TURNED OFF** temporarily by pressing the **CTRL** key while editing (moving or drawing) pipes.

***Hint:** You may wish to carefully move pipe ends or joins, however the move keeps connecting the ends to other nearby (or far away) pipes in the same view direction. To disable this behaviour and allow fine control over moves, simply hold the CTRL key down while moving to prevent the pipes locking on to other objects.*

Pipe Solids

Locks on to the closest pipe when adding, moving or copying new pipes. De-selecting this option will allow pipes and nodes to be finely moved without locking on to a nearby node or pipe, or to travel through each other without connecting.

***Hint:** The lock target can be de-selected when arranging pipes which are closer together. The lock function when selected will normally lock on to the closest available pipe, ensuring a positive join between the pipe.*

***Warning:** If the lock target function is turned off, pipes may not join into each other precisely (even if they appear close or overlap), resulting in no exit or entry errors. The 'lock target' function should always be selected when you intend to join into other pipes or nodes. You can check whether pipes truly join into each other by turn on the NODE display.*

Transparencies

Allows transparent (non-active) pipes to be locked on to, selected or edited. Transparent pipes are normally treated as invisible by the program and by default cannot be selected or changed. Enabling this function will allow the pipes to be selected or edited the same as any other pipe.

References

Locks the mouse cursor onto a referenced graphic surface or wireframe. Pumpsim™ will preferentially search for pipes when moving the mouse cursor to new locations. If no pipe is found, if this option is enabled the program will search for a reference graphic (imported graphics) to lock on to.

***Hint:** This function is very useful for tracing or moving pipes on to imported referenced graphic locations. By using the referenced graphics as a guide, pipes can be accurately placed within a model. To force the program to use the referenced graphics only, and not lock on to other pipes, turn the Lock Target options OFF.*

Vertical

Forces new pipes, or moved or copied pipes to be drawn exactly vertically when the vertical movement function is chosen (with the SHIFT key or the RIGHT MOUSE button) while drawing. When this function is not selected, pipes can be moved both vertically and parallel to the plane of the viewing screen while being drawn.

***Hint:** Pipes drawn or moved with the Lock Vertical function activated can still be angled. Simply release the SHIFT key or RIGHT mouse button while drawing and moving when you are at the desired elevation and the pipe end can continue to be moved horizontally at the new elevation.*

Intersections

Checks whether new pipes intersect existing pipes, and inserts a node if they do. The function works by searching surrounding pipes for intersections or crossing paths that are close. Turn this function OFF if you require a pipe to cross another without joining (for example an overcast pipe) or pipes are required to be drawn very close together without being joined.



Perspective View

Sets the viewing system to **orthogonal** or **perspective**. Orthogonal removes the effect of perspective, making objects in the distance appear the same size as close objects. This mode may be of assistance when viewing sections or plan views, where the perspective appearance would otherwise distort the true scale of objects at different distances. The mode has the disadvantage of a more cluttered and complex display.

Perspective view is more suited to general editing and viewing, where objects in the distance are obscured by perspective scale, creating a less cluttered display.

Perspective On

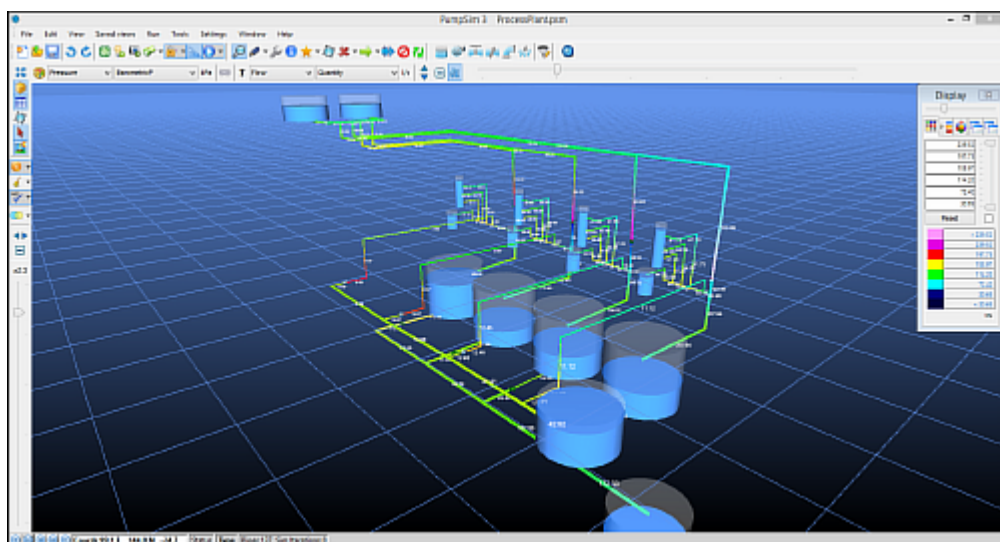


Fig 2.3.4: A pipe system depicted with the perspective on, making parts of the model more distant and others closer depending on the perspective of the user.

Perspective Off

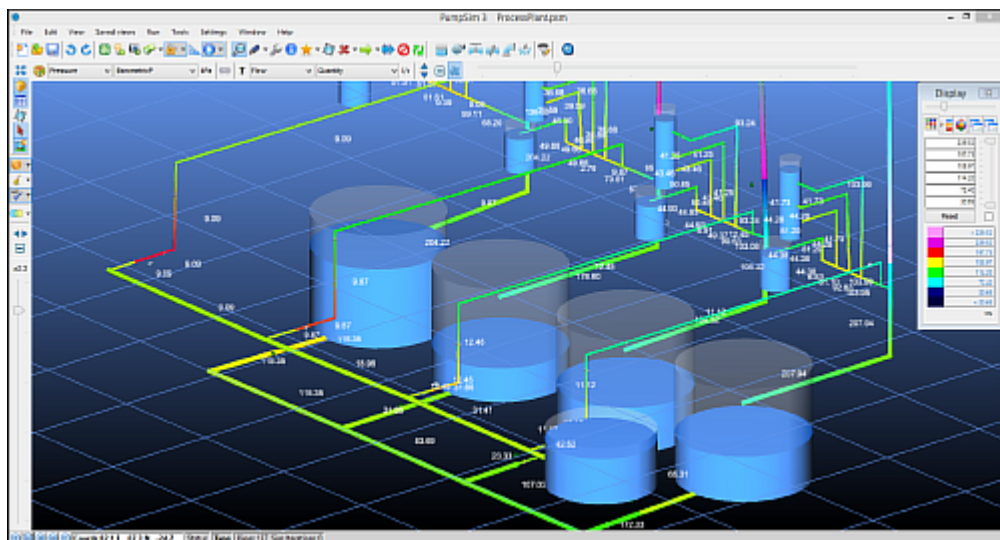


Fig. 2.3.5: A pipe system depicted with the perspective off (orthogonal view), making all parts of the model appear the same distance from the user's point of view.

***Hint:** When performing true PLAN or SECTION views, ensure that ORTHOGONAL mode is chosen to prevent perspective errors in these views. In plan or section view, pipes at a different plan or section depth will be shown at a different scale in perspective view. You can quickly change between views by pressing the 'P' key on the keyboard.*

Flow Animation

Providing the model is designed true to scale, the progress of flow can be tracked at the same speed as expected flow in the underground site when set to **x1** speed.

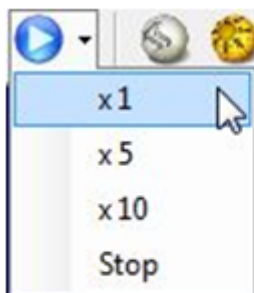


Figure 7-3 Animation Speed Control

Fig. 2.3.6: Flow animation speed control.

The animate toolbar button has a number of sub menu items:

- **x1** – **Animate flows in true real time.** Arrows travel at the same scale speed as calculated velocity.
- **x5** – **Arrows travel at five (5) times** the scale speed of calculated velocity.

- **x10** – **Arrows travel at ten (10) times** the scale speed of calculated velocity.
- **STOP** – **Stops flow animation.** Flow animation in large model can consume significant processing power and slow model response and rotation. It is recommended flow animation be turned off when not needed on larger models.

Hint: Clicking the main flow animation icon will cycle the flow speeds through all available ranges.

Drawing Functions



Fig. 2.3.7 The drawing function of the toolbar.



View

Places the program in **view mode**.

- **Left Mouse Drag** – **Draw a window to zoom into part of a model.** The front most pipe in the view window will define the point of focus for the zoomed in area. To zoom into an area behind other pipes, ensure the zoom window does not contain any portion of the front pipes.
- **Left Mouse Single Click** – **Centre the view on the pipe.** Centres the view on a pipe so the screen can be rotated around the point clicked.
- **Left Mouse Double Click** – **Edit the pipe.** Left clicking the mouse on top of a pipe will show the Edit Box for that pipe.
- **Middle Mouse Drag** – **Press and hold to pan** the screen horizontal around the current **Edit Plane**. **Click to centre the Edit Plane** and rotation point about a specific pipe or point on the Edit Plane.
- **Right Mouse Click** – **IN ALL MODES**, right mouse click will rotate the model graphics about the point of focus. Hold and move the mouse vertically to tilt the model. Hold and move the mouse horizontally to spin the model.



Add

Places the program into draw (add) mode to allow creation of new pipes or measure between pipes. The function has several sub-functions available by clicking the small arrow at the right.

In normal (free draw) mode, pipes that are constructed from other pipes will adopt the attributes of the joined pipe (such as size and wall friction factors types).

Pipes that are NOT constructed starting from other pipes will adopt the default set in the **Settings** menu.

- **Left Mouse Click** – **Edit the pipe.** Left clicking the mouse on top of a pipe will show the **Edit Box** for that pipe.

- **Left Mouse Drag** – **Constructs a new pipe** from where the mouse is initially pressed, to the point where the mouse button is released. To manually control the coordinates of the pipe being constructed, select the sub-menu functions of the add button as shown below.

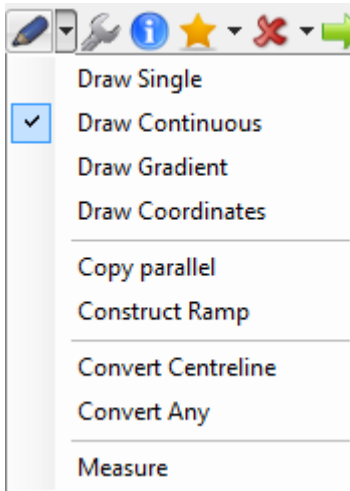


Fig. 2.3.8: The draw mode menu.

- **Draw Single** – Allows the mouse to draw both ends of a single pipe. pipes can be drawn vertically by hold the **SHIFT** key down.
- **Draw Continuous** – Allows the mouse to continuously draw joined pipes until either the **ESCAPE** button is pressed, or the pipe joins into another pipe. Pipes can be drawn vertically by hold the **SHIFT** key down.
- **Draw Gradient** – Allows the mouse to continuously draw joined pipes at a predefined gradient until either the **ESCAPE** button is pressed, or the pipe joins into another pipe. The defined gradient is request when activating the function and can be entered as a percentage (eg 10%), degrees (eg 12.5) or a ratio of horizontal to vertical distance (eg 1:10).
- **Draw Coordinates** – Shows the coordinate entry box after the pipe is drawn to allow manual entry of pipe end coordinates.
- **Copy Parallel** – A tool for quickly building another pipe section parallel to the selected section. Pipes need to be selected before clicking.
- **Construct Ramp** – A tool for quickly building spiral ramps. After selecting the tool, draw the initial direction of the ramp, and the ramp builder toolbox will show, allowing gradients, ramp bends, straights and heights to be established. A ramp can be previewed to try different factors before clicking **BUILD** to commit the ramp design to the model.

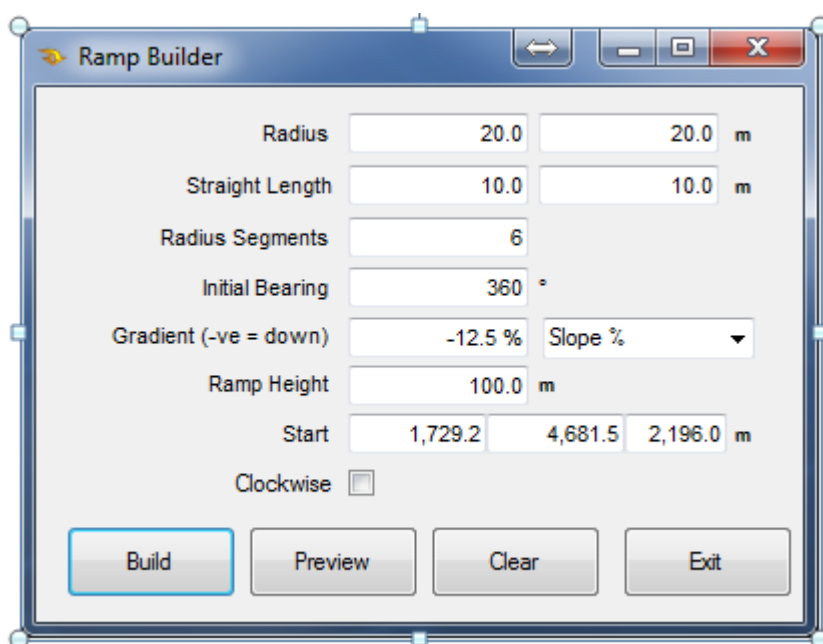


Fig. 2.3.9: The ramp builder window.

- **Convert Centreline** – Allows the selective direct conversion of DXF reference lines to pipes. If imported DXF centrelines are shown as a reference layer on screen (instead of automatically being converted to pipes on import), then this function can be used to select or fence the reference lines that are to be converted to pipes.

Note that each reference line can only be selected once, as the program will not duplicate multiple selected lines.

- **Convert Any** – If centre lines are not available, this is an experimental feature which agglomerates groups of reference graphics to try and establish a pipe path which could be constructed. Examples may include survey string data showing wall, floors and backs, or 3D solids of actual site development or shafts. Note that the results from this function are very approximate and will likely require significant manipulation with the MOVE and DELETE tool function after being done to create a workable model.
- **Measure** – Allows the mouse to measure distances between two points. Click on the start point of where the measurement will start, and drag the mouse to the end point. The measurement will display in the bottom left Status Bar area of the screen.



Edit

Place the program into **Edit** mode.

- **Left Mouse Click – Edit pipe or node.** Left clicking the mouse on top of an object will show the Edit Box for that object.

Note that if several object are selected, only the pipes and channel edit box will show.

- **Left Mouse Drag** – **Selects the pipes within the fence box being drawn.** Selected pipes can then be edited by clicking on any of the selected pipes.

Hint: Selecting multiple pipes permits attributes for a large number of pipes to be changed simultaneously. This can greatly speed up creating a model.



Info

Allows the user to see detailed information about a specific element or a group of elements.

- **Group selected** – Run the summary for this pipe group.
- **Single object** – Show the info box.



Select

Selects a group of pipes. Selected pipes are considered as a group by a number of other Pumpsim™ functions including **Edit**, **Delete**, **Move**, **Copy** and **Filter**. These options will apply changes to all selected pipes simultaneously.

- **Left Mouse Click** – Selects or de-selects the pipe under the current mouse cursor.
- **Left Mouse Drag** – Fences a number of pipes for multiple selection.
- **<Escape>** – De-selects all currently selected pipes
- **<SHIFT>** – De-selects fenced pipes
- **<CTRL>** – Individually selects or de-selects pipes, ignoring the group setting or other group selection options.

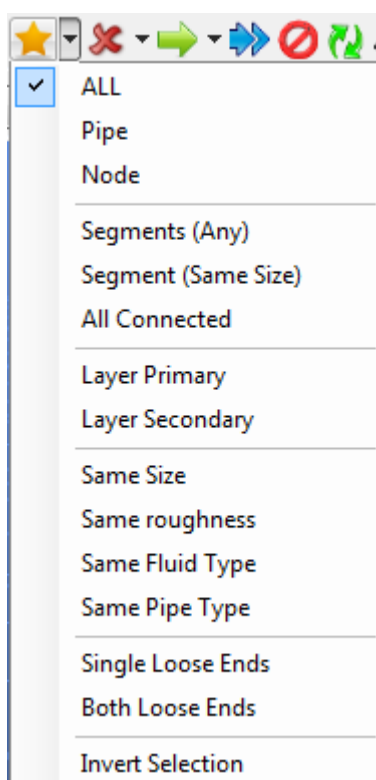


Fig. 2.3.10: The select menu allows the user to select a certain group of model parts.

Multi-Select Options

Permits groups of pipes to be automatically selected, based on either an initial pipe selection, or subsequent selections with the mouse. The selection mode will remain set, until it is reverted back to the single pipe select option, or the <escape> key is pressed twice.

For example, to select all pipes of the same size, either select a single initial pipe with the select button, and then select the multi-select option **“Same Size”**. OR select the **“Same Size”** option first, and then click on a pipe. All pipes of the same size as the initial selection will be selected. Once selected, the pipes can be edited, deleted, moved etc.

Delete

Deletes a pipe or pipes from a model.

- **Left Mouse Click – Delete the object.** Left clicking the mouse on top of an object will delete the object beneath the mouse cursor. If multiple objects have been selected, it will delete all selected objects.
- **Left Mouse Drag – Selects the objects within the fence box being drawn.** Selected objects can then be deleted by clicking on any of the selected objects.

Deletion Options

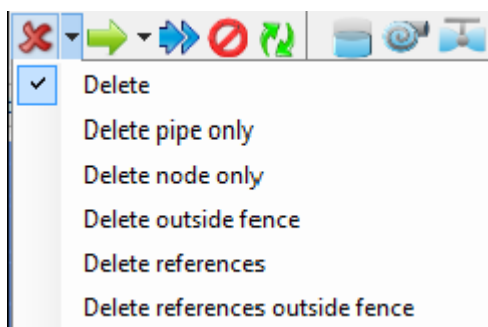


Fig. 2.3.11: The delete menu.

- **Delete pipe only** – will delete only the pipe but not fittings.
- **Delete node only** – will delete will delete fitting but not pipes.
- **Delete outside fence** – will delete all objects not inside the fenced area. This can be useful to remove errant objects which may be located a long way from the main model.
- **Delete references** – removes DXF graphics within the fence area.
- **Delete references outside fence** – remove DXF graphics not within the fenced area.



Move

Moves the selected objects, pipe ends or icons to a new location by clicking and dragging the **LEFT MOUSE BUTTON**. Unless the **SHIFT** key is held, the move will always be done at the same horizontal elevation. A 'true vertical' line is displayed showing where the move point is in relation to other pipes.

- To move a pipe end (and all other attached pipe ends), click on or close to the end node of a pipe.
- To move a pipe end away from other pipes, click slightly back from the pipe end node. The pipe should 'break' away from other connected pipes.
- To move a entire pipe (both ends), first select the pipe with the select button, or fence the pipe to be moved, and then drag the selected pipe(s) to the new location.

Multiple objects can be moved by first selecting the objects and then dragging any one of the selected objects to the new location. Moving a fitting will move all the attached pipes.

- **Left Mouse Click** – Opens the coordinate entry dialog box to allow the move to be specified as a coordinate or offset from the current position.
- **Left Mouse Drag + Shift** – Drags the pipe or node to a new location in the vertical plane, adjacent to the viewing screen. The Edit Plane is moved with the dragged pipe to indicate where the elevation is in relation to other pipes.

Copy

Copies the selected objects or icons to a different location. The original pipes or icons remain in place.

- **Left Mouse Click** – Opens the coordinate entry dialog box to allow the copy to be specified as a coordinate or offset from the current position.
- **Left Mouse Drag** – Drags the copied pipes or icons to a new location. If the ICON is being copied, it will show on pipes under the mouse cursor. When moving pipes, a 'true vertical' line is displayed showing where the move point is in relation to other pipes. To copy an icon (such as a pump, heat source or resistance) along a pipe, simply grab the icon with the left mouse button and drag the icon to another pipe.
- **Left Mouse Drag + Shift** – Drags the copied pipes to a new location in the vertical plane, adjacent to the viewing screen. The **Edit Plane** is moved with the dragged pipe(s) to indicate where the elevation is in relation to other pipes.

Block

Blocks or unblocks a pipe with the highest resistance available (as defined in the **Settings** menu). This will restrict nearly all flow through the pipe.

Warning: Ensure that pumps or fixed flows are not present in pipes leading to or from blocked pipes. This will create a simulation error, as flow will be unable to travel through the blocked pipe without unreasonable pressure or heat build-up.

Reverse

Reverses the direction of the selected node. The direction of the pipe, and the flow within the pipe is reversed. Clicking a node will reverse the attached pipes.

Insert Node

Inserts a node or junction within an existing pipe. The pipe is split into two pipes as a result. This node can then be moved or joined into by other pipes.

Hint: Drawing a pipe into the middle of another pipe will automatically create a new node or junction into the joined pipe.



Filter

Applies a filter to selected pipes to simplify or bind together pipes. See the **TOOLS > FILTER** section for more detail, or more control is required on how the filtering is applied.



Tank

See [Tanks](#)



Pump

See [Pumps](#)



Manual Valve

See [Valves](#)



Automatic Valve

See [Valves](#)



Junction

See [Junctions](#)



Spray

See [Sprays](#)

Scenarios option



See [Scenario](#)



Display a diagram after a dynamic simulation. See [Simulation](#)



Place a monitor to record data during dynamic simulation.

Simulation Functions



Steady State Simulation

Performs a steady state simulation on the model data. For more information, see [Simulation](#).

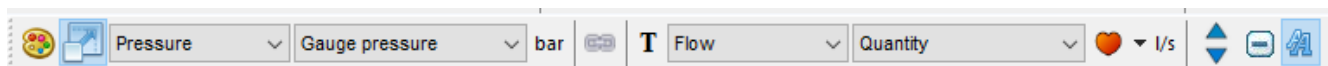


Dynamic Simulation

Performs an flow simulation on the model data. For more information, see [Simulation](#).

2.4 The Data Toolbar

The **DATA TOOLBAR** controls the colours and text displayed on the screen.



Display Manager

Shows the **Display Manager** dialog box, which floats on top of the Pumpsim™ windows and can be kept open at all times if desired.

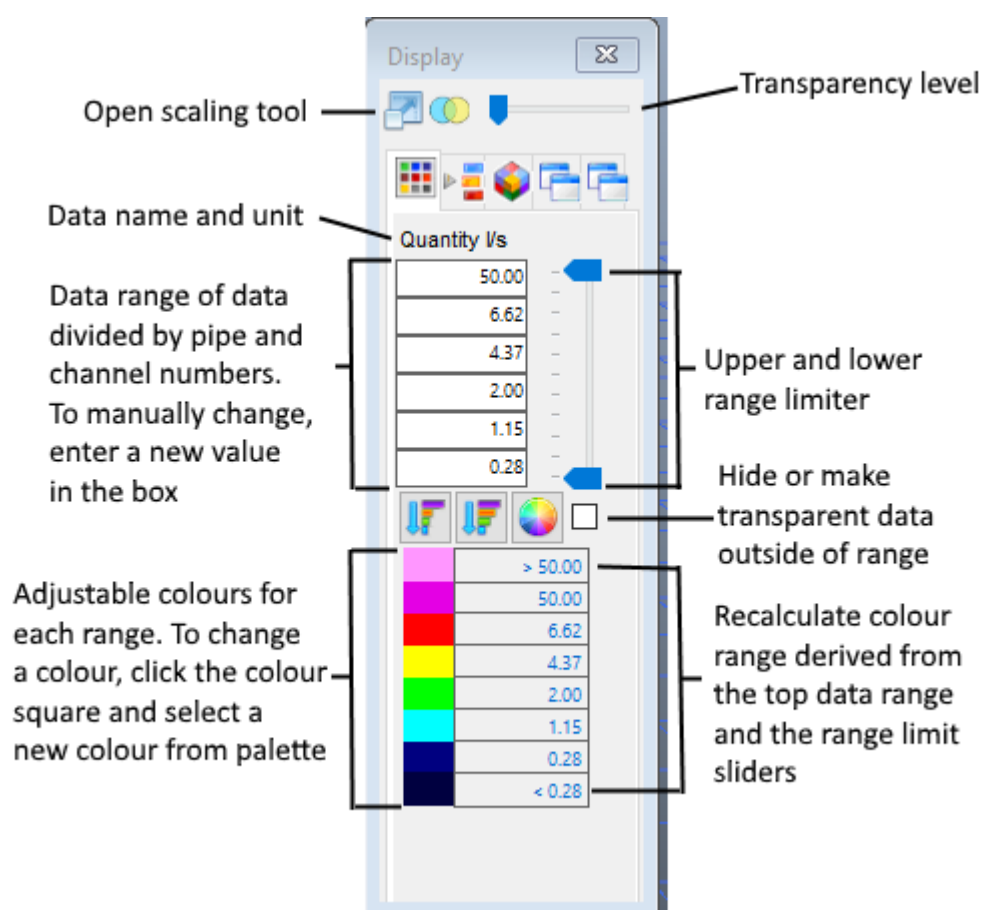


Fig. 2.4.1: The display manager window, with explanations of its components.

- The **Display Manager** control provides options as to what graphics details and colours to display. The display manager contains five (5) tabs which change the colour display between data types.
- The **Display Manager** controls many of Pumpsim's™ advanced display functions. At the core of the

manager is the ability to display any type of flow data as a colour range, including the ability to hide or apply transparent to data outside of a set range.



Ranged Data

The data which creates the colour legend is automatically 'ranged' to fit the model data values. It may occasionally require resetting if new or changed values are present in a model (after a simulation for example), or the values do not divide the colours as you would like. The data can be re-ranged by either manual typing in new values, or allowing Pumpsim™ to automatically choose new values by selecting the **RESET** button.

OPTIONS (Right Click Mouse Button)

- **Reverse Spectrum** – Reverses the colour range direction
- **Restore Colours** – Restores the colours to the default Pumpsim™ palette.
- **Auto Range** – Restores the range to the full spectrum of available pipe data values.

***Hint:** The Display Manager will initially default to automatically range and set colours for the displayed data. If the range values are manually adjusted, or the colours are changed, the Display Manager will no longer automatically range and adjust colours for the data displayed. To restore or recalculate the range and colours, simply click the 'Auto Range' or 'Restore Colours' buttons.*

***Hint:** Pumpsim™ will store a custom colour and data value range for each type of data type. This setting is saved with the Pumpsim™ file.*



Levels

Selects which levels (elevation ranges) are displayed on screen. To select or de-select all levels, choose the check box adjacent to the level name. To change a level name, right mouse click on the existing level name and type in the new name.



Fluid Types

Selects and colours the **Fluid Types** to be shown on screen. **Fluid Types** are designed to allow the user to colour pipes according to water quality or purpose (for example fresh or exhaust water). **Fluid Types** behave in much the same way as primary or secondary layers. To change a fluid type name, right mouse click on the existing fluid type name and type in the new name.



Primary / Secondary Layers

Selects which primary or secondary layers are displayed on screen. Layers are a way to show selected pipes (such as a decline system or a shaft system) on screen while hiding other pipes. Primary and

Secondary layers can be set to individual pipes in the **Edit Box**.

To change a Layer name, right mouse click on the name and type in the new name. Names can also be changed directly from the pipe **Edit Box**, or in the **Preset Box** available from the **Settings** menu.

Data Category Drop-Down

Limits the data types shown in the adjacent pull down menu, to the category type specified.

- **Flow** – Shows data types such as water flow volume, velocity, mass flow and density
- **Pressure** – Shows pressure related data such as relative to surface, barometric and friction loss
- **Attributes** – Shows pipe resistances, friction factors and shock loss numbers
- **Descriptors** – Shows pipe specific identifying traits, such as index or unique numbers and primary and secondary layers numbers

Data Type Drop-Down

Displays screen data related to the **Data Category** specified in the left hand pull down menu. Each data category will have a unique selection of data types that are made available when the category is changed.

Warning: Some data types (such as pressures) may not be correctly set until after a successful simulation. While Pumpsim™ saves previously simulated values in a file, these values may not be valid if further changes and modifications have been made to the model.



Dynamic Text Size

Dynamically rescale the text on the screen so that it can be easily adjusted for clarity or presentation. The "AA" button will toggle between zoom scaled text (which grows larger or smaller the closer to the object graphics) or constant size text (which remains the same size regardless of zoom level).

2.5 The Popup Context Menu (Right-Click)

The **popup** menu, activated at any time by a **right mouse click**, provides a useful collection of commonly used functions. While most of the functions are also available from the toolbar or top menu, the **popup** feature can save considerable time as the mouse cursor is not required to return to the top of the page when a new function is selected.

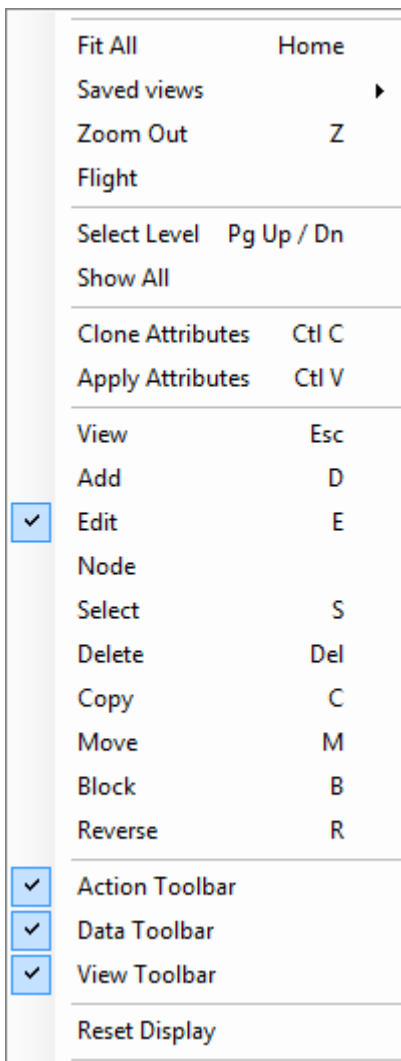


Fig. 2.5.1: The popup menu the appears when the right mouse button is clicked.

Fit All

Scales the view window so all model graphics fit on the screen.

Saved Views

Point of view can be saved and used. See The Menu Bar - > "Saved Views" Menu

Zoom Out

Increases the distance away from the point of focus by a factor of two.

Flight

Enters a flight mode, which allows the user to freely 'fly' around the model in three dimensions. To control the flight system, simply move the mouse similar to an aircraft flight stick, using the scroll wheel to move forwards and backwards.

Mouse Controls

- **Mouse Up** – Pitch down
- **Mouse Down** – Pitch up
- **Mouse Left and Right** – Yaw (turn) left or right
- **Mouse Scroll Button** – Control Speed Forward and Reverse
- **Escape Key** – Exit flight mode

Select Level

Selects one or more levels or elevations for separate display on screen.

This can isolate a specific elevation range of pipes for display, which can clarify display on screen in complex models. The function works independently of the levels specified in the level (elevation) database. Any elevation or range of elevations can be selected at any time.

The elevation range is selected by **LEFT** mouse clicking one or more pipes on the screen, and **RIGHT** mouse clicking to commit and activate the selection. As pipes are clicked, the elevation range selected will turn **YELLOW**. To select a larger area or range of elevations, draw a fence (by holding the left mouse button and dragging the box) around the pipes within the desired elevation.

All pipes outside the elevation range will be hidden or made transparent. The level of transparency can be adjusted from the **Colour Palette Manager**.

Hint: Pipes outside of the normal range of the LEVEL (elevation) database will be displayed as invisible or transparent. The LEVEL database should have a least one level which encompasses all elevations of pipe data in your model.

Level Before:

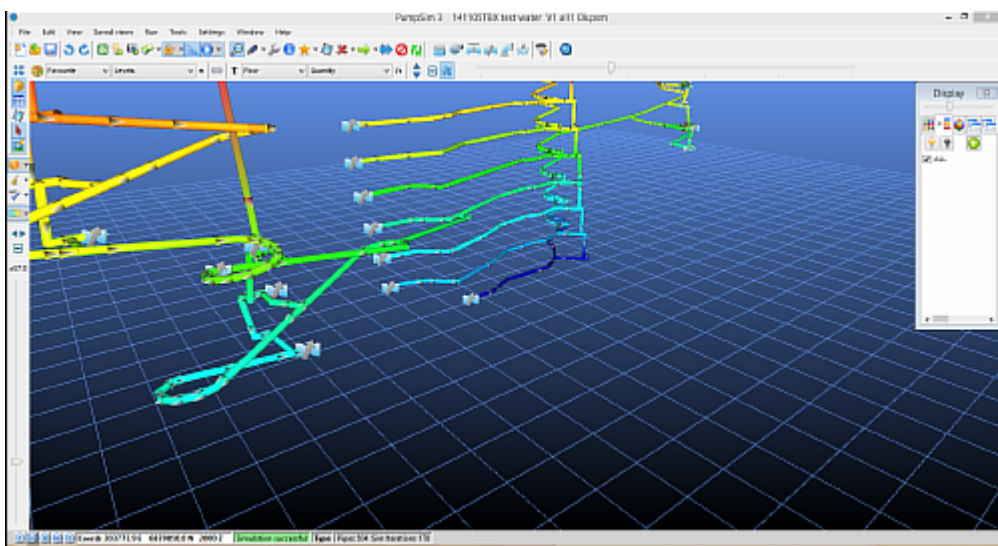


Fig. 2.5.2: A Pumpsim™ model without levels selected for highlighting.

Level After:

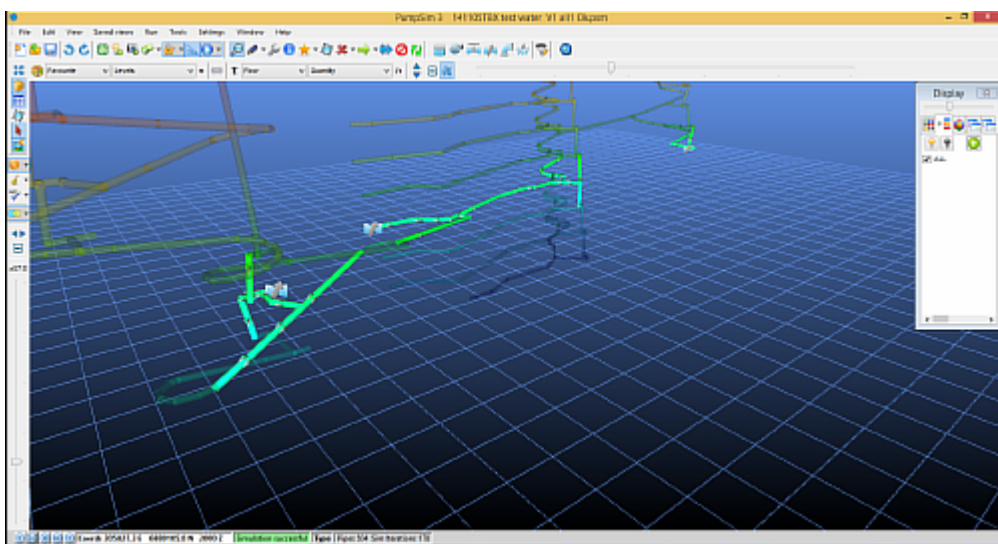


Fig. 2.5.3: The same model as figure 2.5.2 with levels selected. Note how one pipe level remains opaque while the other levels remain transparent.

Hint: Pumpsim™ initially defaults to displaying a 10 metre window range around the selected levels. Pipes outside of this range will not be displayed, or will be made transparent. The range can be changed from the **Settings** menu.

Show All

Display all elevation ranges. This function will reset any levels selected from the **"Select Level"** function.

Clone Attributes

Store the attributes of the pipe or channel.

Apply Attributes

Apply the cloned attributes.

View

Enter the view mode. See [The "Action" Toolbar > View](#) for more information.

Add

Enter the pipe construction mode. Pipes can be drawn by dragging the mouse or edited by clicking the mouse. See [The "Action" Toolbar > Add](#) for more information.

Edit

Enter the pipe edit mode. Selected pipes or fences can be edited in the **Edit** box. See [The "Action" Toolbar > Edit](#) for more information.

Select

Enter the selection mode. Selected pipes can be deleted, edited, moved or copied. See [The "Action" Toolbar > Select](#) for more information.

Delete

Enter the pipe delete mode. Clicked pipes or fences are deleted from the model. See [The "Action" Toolbar > Delete](#) for more information.

Copy

Enter the pipe move copy. Selected pipes can be copied via coordinates or dragged with the mouse. See [The "Action" Toolbar > Copy](#) for more information.

Move

Enter the pipe move mode. Selected pipes can be moved via coordinates or dragged with the mouse. See [The "Action" Toolbar > Move](#) for more information.

Block

Enter the pipe block mode. Selected pipes can be blocked or unblocked to prevent or allow flow. See [The "Action" Toolbar > Block](#) for more information.

Reverse

Enter the reverse pipe mode. Clicked pipes are reversed along with any fixed flows, pressures or pumps. See [The "Action" Toolbar > Reverse](#) for further information.

Action, Data and View Toolbar

The toolbars at the sides of the screen can be toggled off or on as required. Use this function to select the visibility of each toolbar.

Reset Display

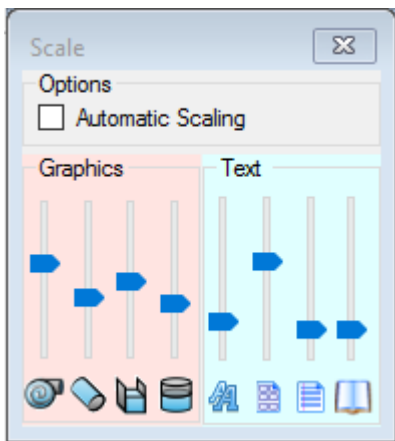
Reconnects the Windows operating system to the graphics hardware device. The software link may occasionally become disconnected or corrupted, resulting in a blank display. Reconnect the DirectX interface will reset the graphics software to hardware link and re-establish the display.

2.6 The Scaling Manager



Scaling Manager

The scaling manager allows the user to increase or decrease the view of items on the screen. For example, if you are designing a precise part of the model like a pump station, decreasing the visual size of the pipes and fittings will allow you to be precise. The scaling manager can be found to the left of the data toolbar as well as in the display manager. The scaling manager does not change the actual size or properties of the items, only the way they appear on the screen. Clicking the icon under the slider will reset the scaling value.



- **Automatic Scaling** – Automatically scale the item in addition to the standard scaling, Automatic scaling is determined by the distance of each item to the point of focus.
- **Graphic Scaling**
 - **Fittings** – Scale the fittings (pump, valves, junctions, demand devices)
 - **Pipes**
 - **Channel**
 - **Tank**
- **Text**
 - **Data Text Size** – Size of the data text
 - **Text Amount** – Changes the amount of data show on the screen
 - **Data Box**
 - **Message Box**

3. Pumpsim Elements

3.1 Pipes and Channels

Drawing Pipes and Channels

To simplify construction of pipes and channels, all **drawing**, **moving** and **copying** of pipes and channels initially defaults to the horizontal **Edit Plane**, regardless of the orientation of the view screen. To assist the user further, pipes and channels being drawn or moved will automatically 'click' and join to objects under the mouse cursor, even if at different elevations or distances away.

*Hint: To quickly change the view from horizontal plan view to vertical section view, press the **F2** button*

If pipes and channels are drawn in isolation (not connected to others), they will use the default settings specified in the **Settings** form. If objects are drawn connected from other objects, they will **inherit** the settings from the objects from which they are drawn. Objects can be drawn from another object's end (node) or from any other point along an object's. Pumpsim™ will create a new node (or junction) if none already exists.

Hint: Pumpsim™ can detect whether an object being constructed crosses the paths of other objects. For example, if a long pipe is drawn from one point to another, over which it crosses the paths of existing pipes, Pumpsim™ will join this object into the crossed pipes with new nodes. This will only happen in the drawing mode. If objects are imported (from a DXF for example), crossed junctions will not automatically be detected.

To draw in the third dimension (up or down in elevation) where there is not objects above or below to click to, first draw the object to the desired horizontal (plan) location, then press the **Shift** key. The **Edit Plane** will turn semi-transparent, and further movements of the mouse will occur in a **Vertical Plane** parallel to the computer screen.

To draw a straight vertical pipe start drawing and press the alt key.

The **Edit Plane** will follow the vertical movement, assisting in showing where the cursor is in relation to other object elevations. In addition, the numbers in the lower left status bar will show the elevation and coordinates of the point.

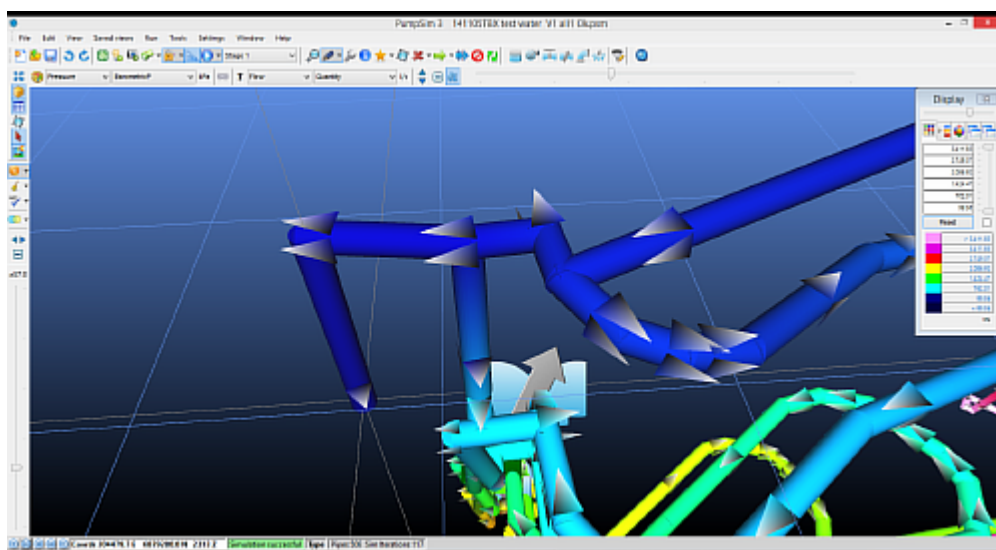


Fig. 3.1.1: A pipe that has been drawn in a vertical plane.

Manual Coordinate Entry

Pipes and channels can be manually added, moved or copied by selecting a manual coordinate entry system. To activate this system while in draw mode, click on an object end or somewhere in empty space. To activate this system every time an object is drawn (to allow manual adjustment or entry of object ends) select the pull down arrow on the **ADD** button and choose the **Coordinates** sub-option from the button.

To activate the system, when moving or copying, simply click an existing pipe or channel end while in the **Move** or **Copy** mode. A coordinate window will show and the coordinates (or offsets) of the object can be entered.

The coordinate entry allow the ends of objects to be manually adjusted, or by using a vector polar coordinate or a physical offset of the easting's, northing's and elevation. End coordinate values are adjusted in real time as offsets or polar coordinates are adjusted. To apply the change, simply click OK when completed.

Moving Pipes

Depending on where the object is clicked with the mouse in **Move** mode, either end, or the entire object may be moved. If the ends (nodes) of the object are connected to other objects, they will be stretched to accommodate. An object can be 'broken' away from a node by selecting it a small distance back from the node, and 'dragging' it away with the mouse. Multiple objects can be copied or moved by **Selecting** the objects (with the **Select** button, or by drawing a **Fence** around the objects while in **Move** or **Copy** mode), and then dragging the selected objects or clicking one with the mouse coordinates. If a fitting is clicked and is not selected, the linked pipes will move with it.

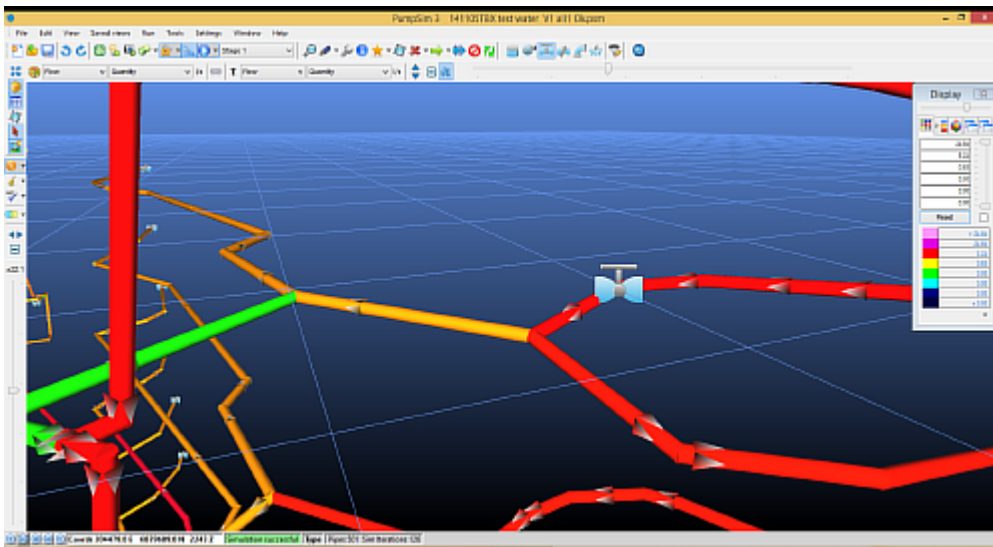
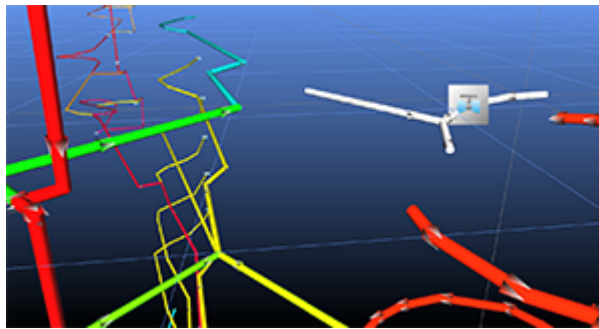
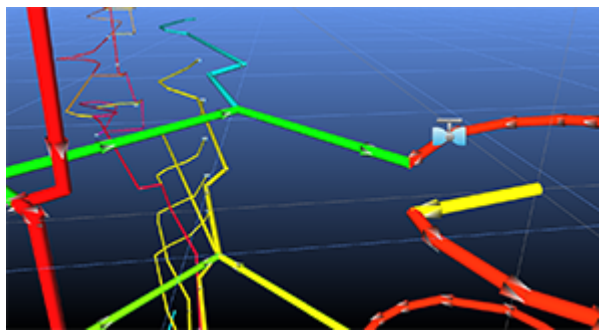


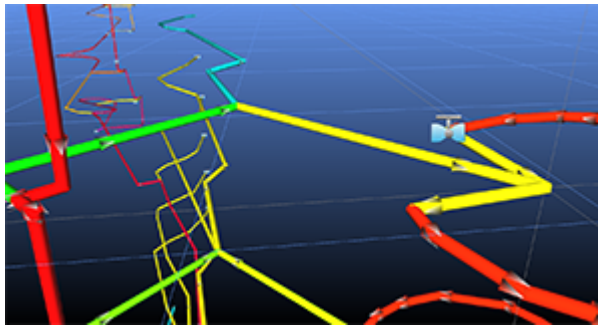
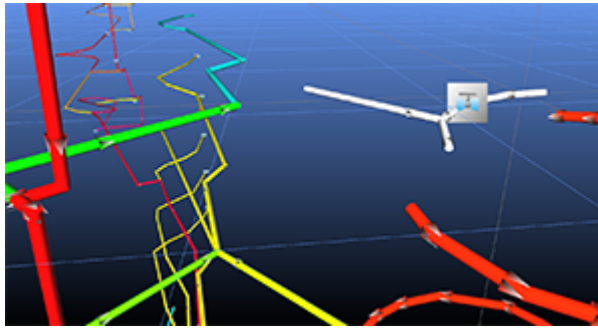
Fig. 3.1.2: A model before pipes have been moved.



Pipes and node are selected and then moved simultaneously.



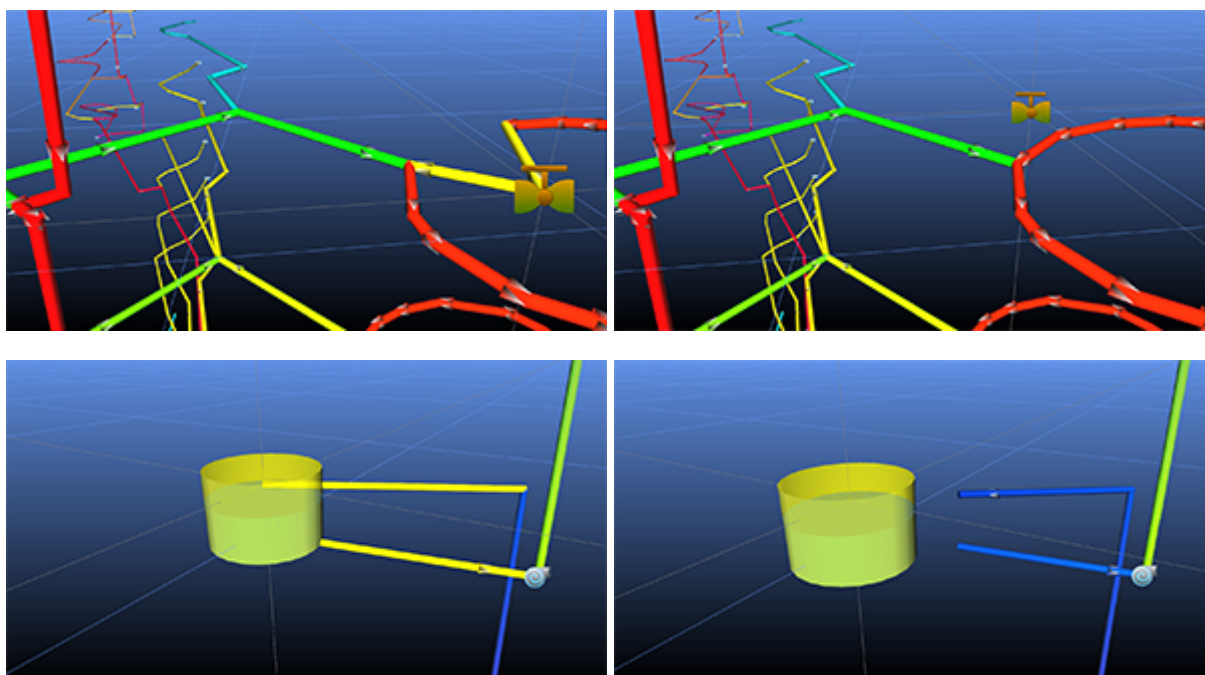
Pipe is 'broken away' from node and pulled away with mouse.



Pipe junction is moved with all attached pipes.

Moving Nodes

If the mouse is on the node only, it will move only the node. If at least one pipe is selected, the pipes will move to stay connected. Pressing the control key allow the tank to move away form pipe.



Junction is moved with all attached pipes

Fitting is moved away without the pipes

Copying Pipes

Objects can be copied, much in the same way as the **Move** function. A pipe can be 'grabbed' with the mouse and 'dropped' in a new location, or the pipe can be clicked and manually copied with new coordinates. In addition, a group of pipes can be selected with the **Select** button, or by drawing a fence around the pipes, and then clicking or dragging one of the selected pipes.

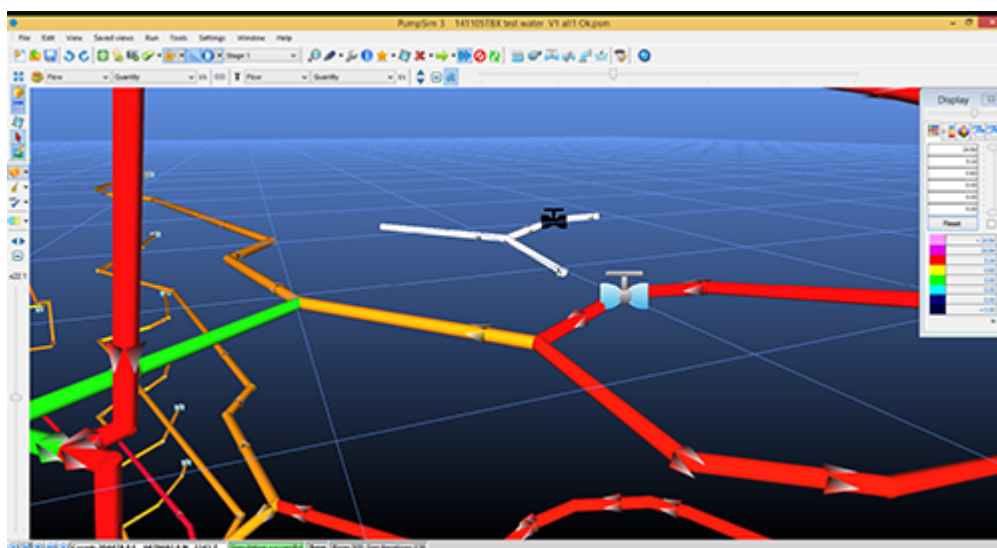


Fig. 3.1.3: A copied pipe.

Channels

Pumpsim™ allows the user to simulate channels in the network. Channels are created with an open top, hence there is no pressure built. The flow entering a channel will be the flow exiting.

Rules for building channels

1. *Entry and exit*

Channels are simulated in sections; water will enter the channel section from a **pipe**, a **fixed flow pump** or an **overflowing tank** and exit by a **drain**, a **tank** or an **open end**. If water is flowing directly from a tank and the channel is at the tank water level, the tank will overflow into the channel. If the water level is higher than the channel connection, the flow will be calculated based on the flow through an orifice with the same size as the channel.

Caution: *The flow in the channel cannot be fixed at the exit. There should not be any fitting at the end or in the channel.*

2. *Direction*

The direction of the flow in a channel is based on the elevation. Pumpsim™ will make all channels flow down, except for a **channel considered flat**. To make the drawing easier, channels with a small slope will be considered flat. The minimal slope can be changed in the **"Settings" panel**. The water depth in a channel will be calculated using the minimal slope.

Caution: *Even though Pumpsim™ can handle flat channels, it is advised to have at least one channel with the right slope per segment.*

Shapes

The shape of a channel will influence the water depth, hence the velocity of the water. Pumpsim™ allows four different channel shapes:

- **Triangular**
- **Rectangular**
- **Trapezoidal**
- **Semi circular** (as based on a circle, the width cannot be greater than twice the depth)

These shapes are defined by their width at the top and their depth (and bottom width for trapezoidal channel shapes).

Drain hole

Pumpsim™ make possible simple simulation of drain. If a pipe is set as a Drain Entry, the water capacity will be limited by how much water goes into the drain and by the maximum capacity of the drain (flow considering than the drain is full and at atmospheric pressure on the drain hole). Two type of flow can develop:

- The flow entering the drain is lower than the maximum capacity. The pipe will then not be full, the flow is assume constant along the pipe and the pipe have no pressure.
- The flow entering the drain is higher than the maximum capacity. The drain will behave like a normal full pipe. A warning will show as there will be an overflow at the entry of the pipe.

Caution: Only one pipe or channel can be connected to a drain hole. If several sources exist, simply add a small pipe or drain connecting on one side the several connections and the drain on the other.

Editing Pipes And Channel

EDIT - 1 Pipes, 162.5 m

Name Index 503 Unique 72'

Entry	<input type="text"/>	1,869.5	1,610.7	245.1
Exit	<input type="text"/>	1,928.9	1,715.6	354.0

Notes

Data Box

Type **Optimise Pipe Size(s)**

Material POLY

Class PN12

Type 100mm PN12

Parallel 1

☒ Pipe ☐ Channel

Options

☐ Open ☐ Close

☐ Exclude ☐ Drain Entry

☐ Show Data

☐ Group

☐ Fix Direction

☐ Fix Length 162.5 m

☐ Gradient 90.3 %

☐ Fix Height 108.9 m

Specifications

100	mm	ID
110	mm	OD
0.0	%	Obstruction
0.00152	mm	Roughness
12.50	bar	PN
0.031	m2	Area
0.00	C/km	Leakage

None Leakage

Tank Connection

Height m

☐ Control Inflow

Layers

Fluid Dirty water

Primary Dewatering (L

Secondary 0

Apply Ok Cancel

Fig. 3.1.2: The window for editing pipes.

Pipe Names and Coordinates

Sets the pipe and node names, pipe coordinates. To set new names or coordinates, simply type the information into the relevant text boxes. Names are optional and are not required for the simulation. To avoid mistake, changing the coordinate of a node connected to a tank is disabled. The height of the connection can be change in the Tank Connection section.

Pipe Size

Sets the physical size and shape characteristics of the pipe.

- **Material** – Filter the different pipe type by material.
- **Class** – Filter the pipe type by the pipe class.
- **Type** – Allows a preset type of pipe to be specified, which will automatically set the pipe profile, dimensions and friction factors. Pressing the TYPE 'T' button will open the PRESET form where pipe preset types can be entered or modified. If a pipe type is modified, all pipes set to that type will be automatically modified during subsequent simulations.
- **Shape** – Sets the profile of the pipe or channel.
- **ID (Inside Diameter) , for pipe** – The size displayed on screen is based on the ID.
- **OD (Outside Diameter), for pipe**
- **Width, for channel**
- **Depth, for channel**
- **Water depth, for channel** – **Calculated after simulation**, knowing the flow.
- **SDR** – Standard dimension ratio.
- **Roughness** – Pipe roughness. Used to calculate the **Darcy Friction** factor. Manning roughness of the channel.
- **PN** – Pressure nominal. Maximum pressure before the pipe breaks, a warning will show if, after simulation, the pressure is higher.
- **Leakage** – Amount of leakage per length. The leakage is specified by how much flow goes out of the pipe for 1bar of pressure per length of pipe.
- **Area** – **The calculated area**, derived from the ID and shape for pipes and from the width, water depth and shape for channels.
- **Obstruction** – Assumes a **continuous area obstruction** along the pipe (for example a duct, services or compartment) and removes this value from the available area of the pipe. Note that this should not generally be used for short fixed objects (such as a vehicle); in this case the orifice function may provide a better simulation result.
- **Length** – **The calculated length of the pipe**. The calculated length is automatically derived from the pipe coordinates. To change the length, click on the **Check Box**, and type in the new length. This may be required if the model is drawn as schematic if a pipe is moved to allow easier viewing, but the true original pipe length still needs to be simulated in the model.

Pipe Options

Controls various attributes relating to the pipe behaviour and identification within a model.

- **Open** – Connects the pipe to the surface, allowing it to freely exhaust and intake water from the surface atmosphere. The pipe end which is not connected to any other pipe is assumed to be the end connected to the surface. If both pipe ends are connected, this item is ignored by the simulation. The exit to the surface is assumed to be at the elevation of the pipe end. Barometric pressures at this point

will be adjusted for any difference in height between the pipe elevation and the defined surface elevation in the settings.

- **Close End** – Allows Pumpsim™ to assume the pipe is a 'dead end' without connecting to other pipes or to the surface. Examples of this may include a dead end, blind developing heading or undeveloped heading. The simulation process will assume this path is blocked, and allow no flow along the pipe. If this settings is not used, a simulation will report a 'no entry' or 'no exit' error.
- **Group** – Groups any selected pipes into a single selectable group that can be selected with a single **Select** click. Individual pipes can still be individually edited by clicking on them directly with the **EDIT** button instead of using **SELECT** first, or by holding the **<CTRL>** key down while selecting. Grouped pipes are a convenient way to collect pipe systems together (for example shafts or decline sections), so they are easier to select and edit simultaneously. It has no effect on simulation.
- **Exclude** – Excludes the pipe from the simulation process. Any errors or problems with the pipes are ignored during simulation and no flow is allowed through the excluded pipes. Any non-excluded pipe which joins with an excluded pipe will be assumed to be blocked. This function is handy for excluding sections of a model design which have yet to be mined, or perhaps removing old sealed or filled sections of a site which no longer flow. Excluding pipes will speed up simulation for the remaining pipes.
- **Fix Direction** – Prevents Pumpsim™ from changing the pipe's flow direction during simulation. An error is displayed after simulation if an attempt to reverse is made. This function is useful to ensure critical flow locations underground are not arbitrarily changed without first warning the user.
- **Show Data** – Always shows text data for this pipe when the **LIMIT** view option is set.
- **Gradient** – Specifies the gradient of the pipe as a percentage. For example a gradient of 10% results in a pipe increasing 1m in height for every 10m in horizontal length. If a group of selected pipes needs to be change simultaneously, use the **TOOL > APPLY GRADIENT** option in the **EDIT** form menu. Note that the first selected pipe clicked with the **EDIT** button will be the pipe from which the gradient is applied.
- **Orifice** – Places a sudden restriction in the pipe, with an open hole equivalent to the specified area. The resulting shock loss and resistance of the orifice will be added to the resistance of the pipe. Orifice settings are normally used to simulate the effect of regulators, holes in wall, partly open doors or other sudden restrictions which may occur in underground headings. It can also be used to simulate the effect of large equipment blocking a pipe by entering the 'remaining' area around the equipment as the 'orifice' size.
- **Fix Height** – Height difference between entry and exit of the pipe. If checked, the simulation will use the value entered instead of being calculated from the position.
- **Fluid Type** – Sets the type of water the pipe is carrying, for example fresh, dirty, slurry etc. The fluid type will change the physical parameter of the fluid in the pipe, such as viscosity or density. The fluid type can be shown as a separate colour on screen by selecting **water type** under the **favourites menu** on the **Display Manager**.
- **Primary Layer** – Sets the primary layer type of the pipe. Once set, primary layers can be independently displayed using the **Display Manager**. The name of the primary layer can be changed by clicking the **Primary Layer** button, and entering a new name or colour in the **Preset spreadsheet**.
- **Secondary Layer** – Sets the secondary layer type of the pipe. Once set, secondary layers can be independently displayed using the **Display Manager**. The name of the secondary layer can be changed by clicking the **Secondary Layer** button, and entering a new name or colour in the **Preset spreadsheet**.

Tank Option

- **Height** – Distance between the pipe entry and the bottom of the tank. Changing this value will adjust the pipe position
- **Connection shape** – Shape of the connection between the tank and the pipe. It adds a resistance coefficient depending on the shape. More shapes can added in the presets.

3.2 Fittings

Fittings are used to simulate elements other than pipes in the network. The available fittings are:

- **Tanks**
- **Valves (Manual and Automatic)**
- **Sprays**
- **Junctions**
- **Pumps**


Changes made to the objects in the box are highlighted in **BLUE**. The changes are applied to the pipe when;

- **Apply is pressed** – the **Edit Box** form will apply changes and remain open
- **OK is pressed** – the **Edit Box** form will apply changes and close

If another pipe is selected, the changes will be applied, and the next pipe data will be displayed.

Multiple pipes can be selected and simultaneously edited from the **Edit Box**. Note that not all edit functions are available for multiple editing and some functions may be hidden. To utilise multiple pipe editing, simply select the required pipes before the **Edit Box** is opened. Only attributes that are changed (and therefore highlighted in **BLUE**) are applied to the selected pipe(s).



Presets can be access by clicking . If the preset is changed, all the objects to which this preset is applied will change.

Adding Fittings

As for pipes and channels, fittings will be added onto the horizontal **Edit Plane**. If a pipe is clicked, the pipe will be separated in two parts and the fitting added at the junction. In order to facilitate the addition of fitting, click-and-drag will select the possible positions in pipes for this specific fitting:

- At every open pipe for : sprays and tanks.
- At every junction between two pipes for : pumps, valves and junctions.

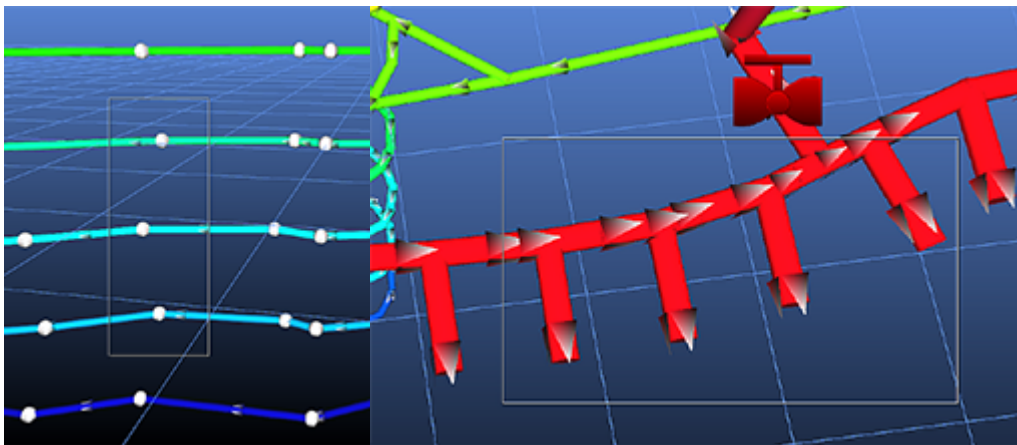


Fig. 3.2.1: Added fittings.

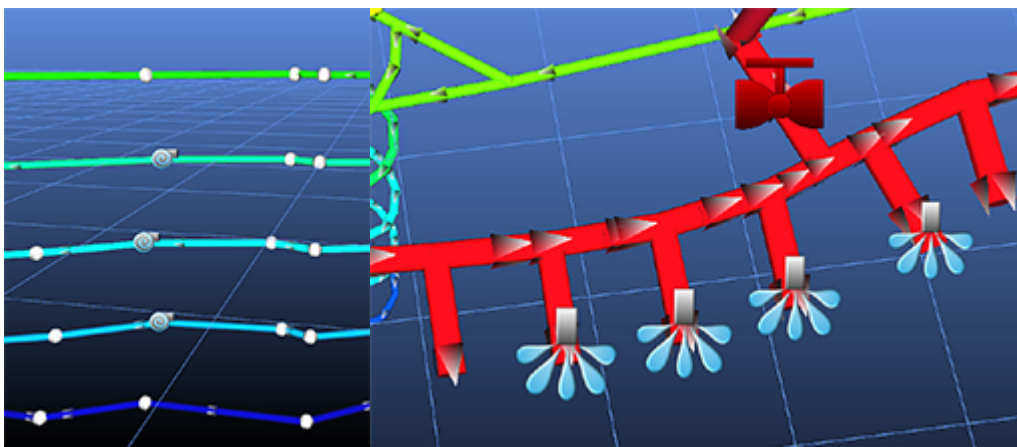


Fig. 3.2.2: Added fittings.

3.2.1 Tanks/Dams/Recievers

Containers are defined by their dimensions, shapes and the level of water and sediment. The pressure loss due to the connection with a tank can be adjusted in the **Pipe Edit Box**.

Pumpsim™ has 4 types of fluid container:

- **Tanks** – Tanks are liquid containers in a simple shape: round or rectangular. The height of the pipe connection will influence the pressure at the inlet of the pipe.
- **User Defined** – User defined containers are a complexly shaped body of water such as an aquifer. They are defined in tools and then the tank tools add a connecting point to the network.
- **Receiver** – Receivers are for air. It is common to have them after a compressor to store the compressed air before being used. The height of the pipe connection will not matter for this type of container.
- **Dam** – Dam are shapes more complex than tanks, for example a dam with a certain slope.

Tank

There are three methods for a pipe to connect to a tank:

- **Below the water level** – The water can flow in and out of the tank. Pressure head will be added to the pipe, except for a dam.
- **Above the water level** – The water can only flow into the tank.
- **At the water level** – The pipe will act like a drain and the flow will be the sum of the flow entering minus the flow leaving the tank by another pipe. If more water leaves than enters the tank or if no water enters the tank, then the flow in the drain will be 0. The flow in the drain cannot be higher than the capacity of the pipe.

Channels can only flow into a tank and be connected above the water level, otherwise a warning will say that the channel is overflowing. Water cannot flow from a tank straight into a channel; at least one pipe is needed between the channel and the tank.

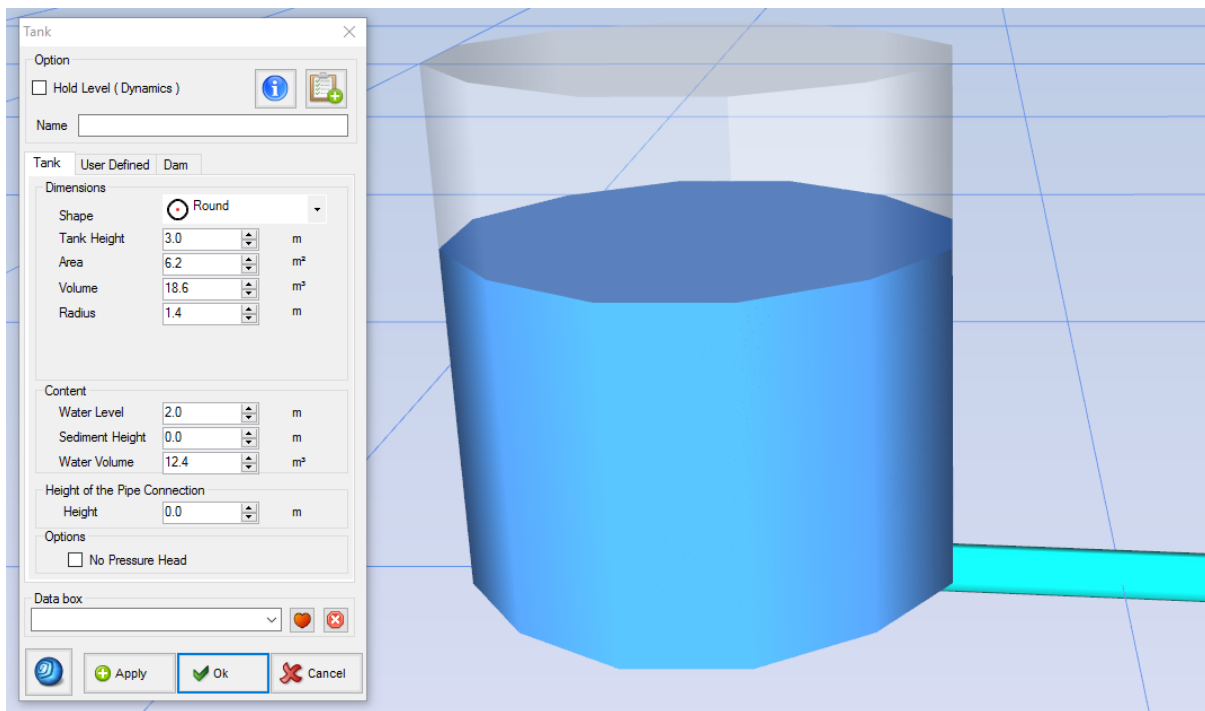


Fig. 3.2.1.1: A tank with a pipe flowing into it below the waterline, with the tank editing window open.

- **Options**
 - **Hold Water Level** – The water level of this tank will remain the same during a dynamic simulation.
 - **Name** – Name of the tank.
- **Dimensions**
 - **Shape** – Shape of the tank.
 - **Tank Height** – Total height of the tank.
 - **Area** – Tank area. Linked to the radius, changing the area will change the dimensions of the tank according to the shape.
 - **Radius / width / width²** – Tank dimensions. Linked to the area, changing the dimensions will change the area of the tank according to the shape.
- **Content**
 - **Water Level** – Height of the water level in the tank. The pressure head will be calculated using this value and the height of the pipe connection with the tank.
 - **Sediment Height** – Height of sediment, measured from the bottom of the tank.
 - **Height** – Distance between the pipe connection and the bottom of the tank. Appears only when there is just one pipe connected to the tank. If changed, the tank position will adjust.

Dam

Dams behave in the same way as tanks, except that they can be more complex in shape, like the following shape. A common dam would be an underground sump with a slope.

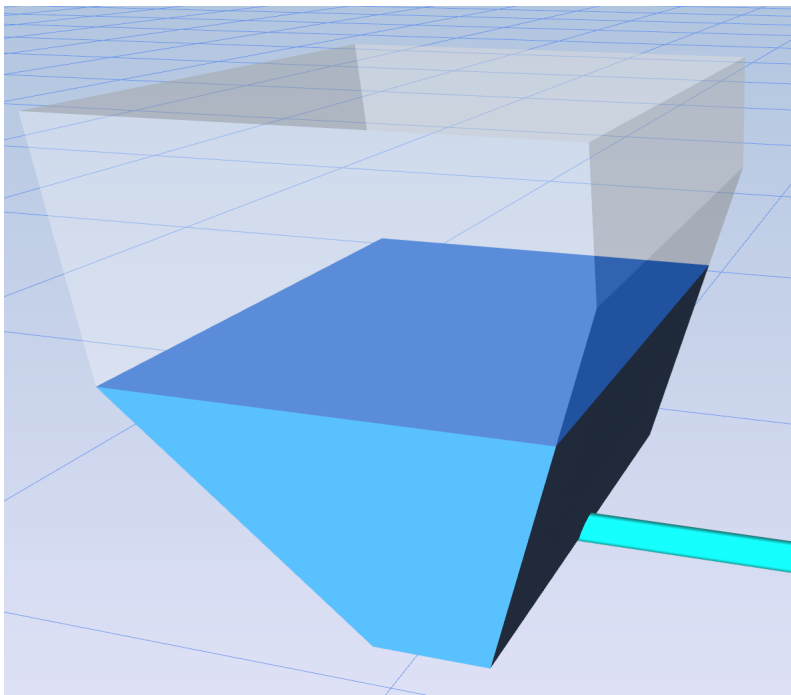


Fig. 3.2.1.2: Rectangular shape dam with slopes on both sides.

To create a dam, add a container and select the Dam option. To edit the shape, click on the picture representing the cross view of your dam. Design of the dam is done by adjusting the height and width on each side.

Tank

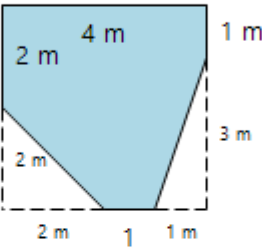
Option

☐ Hold Level (Dynamics)

Name

Tank User Defined Dam

Dimension



Content

Water Level 2.0 m

Sediment Height 0.0 m

Water Volume 0.0 m

Height of the Pipe Connection

Height 0.0 m

Options

Angle 0.0 Degree

Data box

Apply Ok Cancel

Fig. 3.2.1.3: Edit box for the above dam

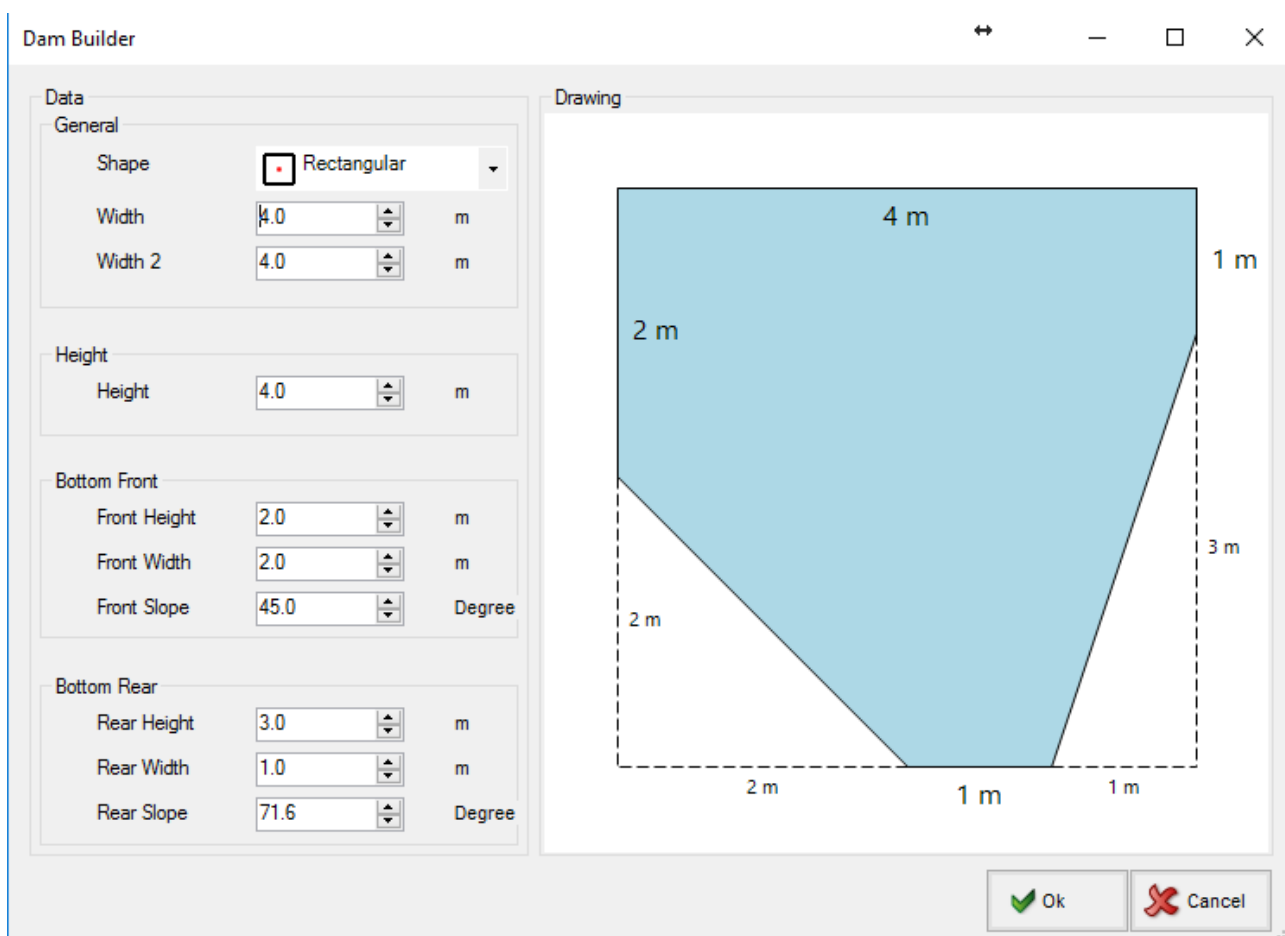


Fig. 3.2.1.4: Dam builder

Hint: Every pipe connected to a dam connects to its centreline, which is the vertical line in the center of the tallest part of the dam.

User Defined

A user defined container is a large body of water from which several different points connect. They can be used when the shape of the object is complex (pit) or unknown (aquifer).

To use them, first enter a volume vs pressure curve for a specific elevation. Also enter a water volume which will change during the simulation.

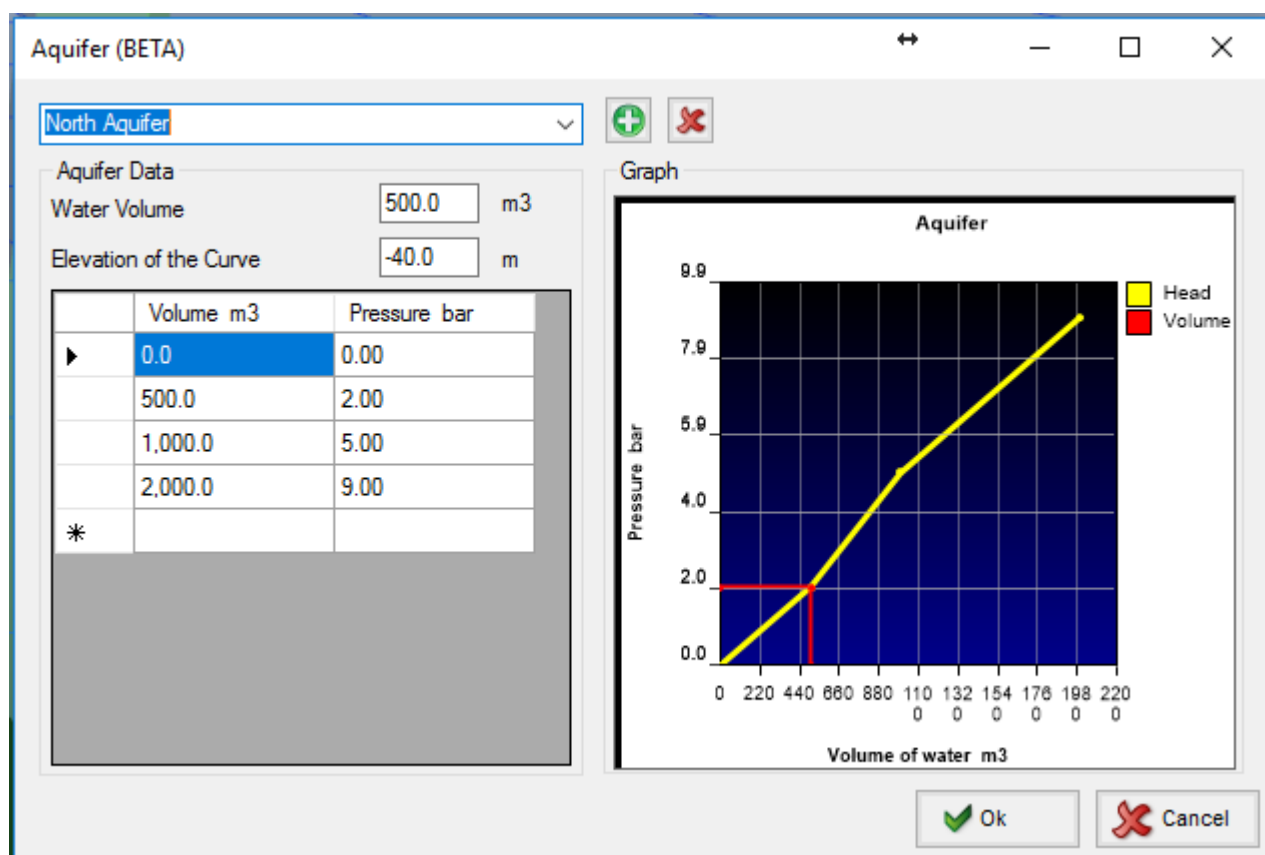


Fig. 3.2.1.2: Creating an aquifer.

Then add the user defined node where water from comes out of the aquifer. The resistance of the aquifer as the water flows down will need to be entered to each point, it is calculated knowing the size of the opening, the pressure at this point when the opening is closed and the flow when the opening is fully open.

Receiver

Air receivers are containers with a similar shape to a tank but the main difference is that the height of the connection does not matter, as it contains a pressurised fluid. The amount of air contained can be entered as the volume of free air or as a pressure. The dark green color in the graphics represents the percentage of pressure in the tank versus the maximum pressure that the receiver can contain.

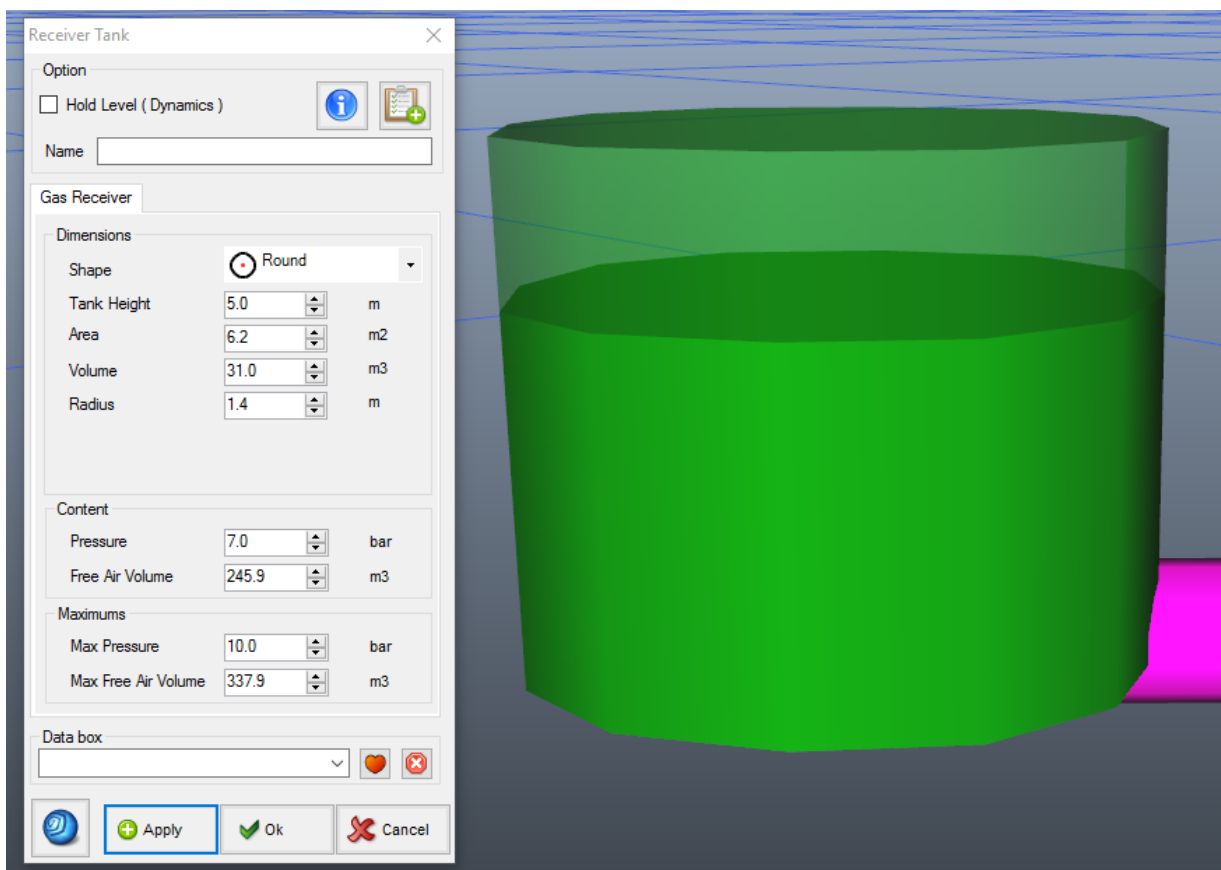


Fig. 3.2.1.2: Receiver and its edit form.

3.2.2 Valves



Manual Valves

Manual valve refers to valves that the user can open or close to increase or decrease the flow across the valve.

They can be only be added from the presets. The more important element is the flow coefficient. It represents the amount of water passing through the fully open valve with a difference of pressure of 1bar (SI) or 1psi (Imperial). The equation type links the opening percentage to the flow coefficient at the opening. It depends on the type of valve. If in doubt, using the linear equation should give an accurate result.

1 valves

Selection

Butterfly 200 mm

Filter: 200 mm

Name: 1300 fresh water inlet

Opening

% Open: 100.00 %

Flow Coefficient: 1280.0000 m3/h

Information

Equation	Ball	
Fully Open Flow Coefficient	1,280.000	m3/h
Size	200	mm
Maximum Pressure	0.00	bar
Cost	55	\$

Valve Notes

Data box

Apply Ok Cancel

Fig. 3.2.2.1: The manual valve addition edit box.

- **Selection**
 - **Valve name** – Valve name, including the size and type. More valves can be added in the presets.
 - **Size filter** – Will only display valves with the selected size.
 - **Name**
- **Opening**
 - **% Open** – Percentage of valve opening.
 - **Flow Coefficient** – The flow coefficient at this specific opening percentage, depending on the type of valve.
- **Information**
 - **Equation** – The type of valve, it will influence the flow coefficient at opening.

- **Fully Open Flow Coefficient** – Depends on the valve and can be found in the valve manual. It is the quantity of flow passing through the valve with a pressure difference of 1bar (SI) or 1psi (Imperial).
- **Size** – Size of the entrance of the valve.



Automatic Valves

Automatic valves refer to valves that will automatically open or close themselves to achieve a set value.

Fig. 3.2.2.2: The automatic valve addition edit box.

- **Selection** – Valve name, including the size and type. More valves can be added in the presets.
- **Data**
 - **Reduced Pressure** – The valve outlet barometric pressure is set. If the set pressure is too low, the valve may fully close. If the set pressure is higher than the pressure with the valve fully open, the valve will be fully open. If the flow tries to go backward, the valve will close.
 - **Sustained Pressure** – The valve inlet barometric pressure is set. If the pressure with the valve fully open is greater than the set pressure the valve will stay open. If the valve pressure with the valve fully closed is smaller than the set pressure, the valve will close. If the flow tries to go

backward, the valve will close.

- **Ratio Reduction** – Sets the difference of barometric pressure between entry and exit of the valve. If the set pressure is too low, it is possible that the valve will have to close. If the flow tries go backwards, the valve will close.
- **Fix Flow** – Fix the flow across the valve. If the set flow is higher than the flow with the valve fully open, the valve will be fully open.
- **No Return** – The valve will close if the flow tries to reverse.
- **Info**
 - **Size** – Diameter of the valve entry.
 - **Cost** – Price of one valve.
 - **Maximum Pressure** – Maximum pressure which the valve can support before breaking.
 - **Notes**

3.2.3 Sprays

Sprays are simulated using a discharge coefficient. In the presets a calculator can approximate this factor from a table flow/pressure.

Spray can be placed at the end of a pipe or between pipes.

1 Demand Devices

Status: **ON**

Name: Level 1200 fire spray

Selection: Flat nozzle 125 mm

Data:

- ☐ Set flow: 0.00 l/s
- ☒ Discharge coefficient: 20.00 lpm/bar^{0.5}
- Size: 125 mm
- Minimum Pressure: 2.00 bar
- Maximum Pressure: 40.00 bar
- Cost: 450 \$

Data box:

Buttons: [Help] [Apply] [Ok] [Cancel]

Fig. 3.2.3.1: The spray addition edit box.

- **Name**
- **Selection** – Spray name, including the size. More sprays can be added in the preset .
- **Data**
 - **Set flow** – Fix the demand flow.
 - **Discharge coefficient** – Spray discharge coefficient. Can be calculated in the preset.
 - **Size** – Characteristic size of the spray.
 - **Minimum pressure** – Minimum pressure for which water will pass through. No water will pass for lower pressure.
 - **Maximum pressure** – Maximum pressure which the device can sustain.
 - **Cost** – Price of an expense.

3.2.4 Junctions

"Two Way" Junctions

"Two Way" Junctions can be used to simulate bends, elbows or any other objects with fixed ***K factor*** (or equivalent length). ***R/D*** represents the radius of the curve divided by the diameter of the curve. The ratio and angle are not necessary for the simulation. Junctions can be selected from the presets or entered as a custom value.

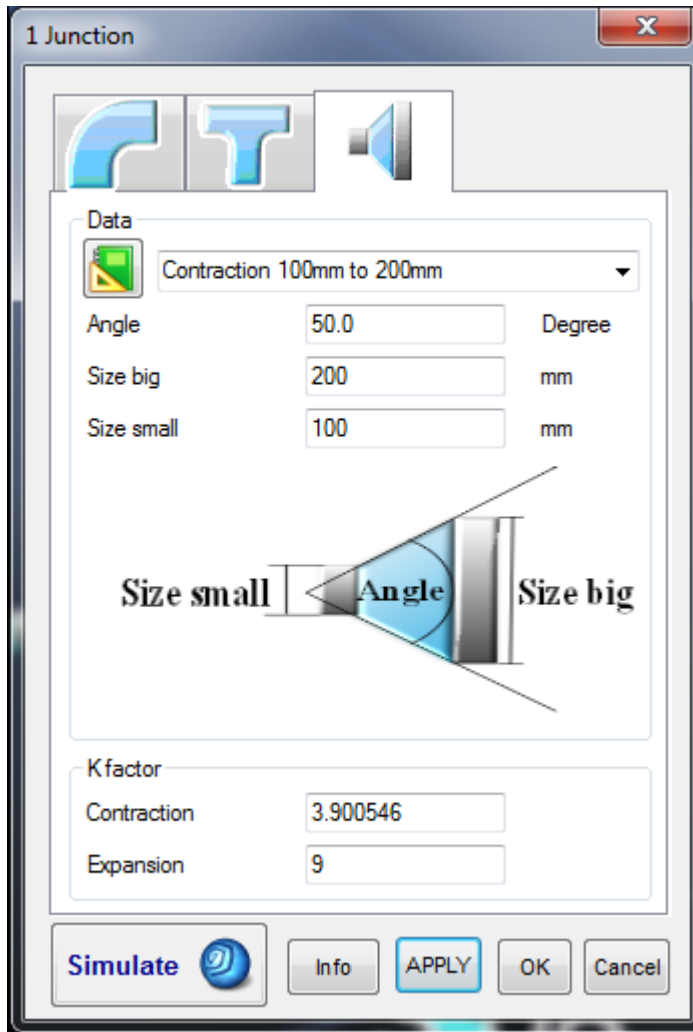


Fig. 3.2.4.1: The two-way junction addition box.

"Three and Four Way" Junctions

"Three and Four Way" Junctions are simulated from two values, one for the flow through the junction and one for the flow of the branch. For "Tee" and "Wye" pieces, Pumpsim™ uses the drawing to determine if it is a flow-through or branch flow. Junctions can be selected from the preset or entered as a custom value.

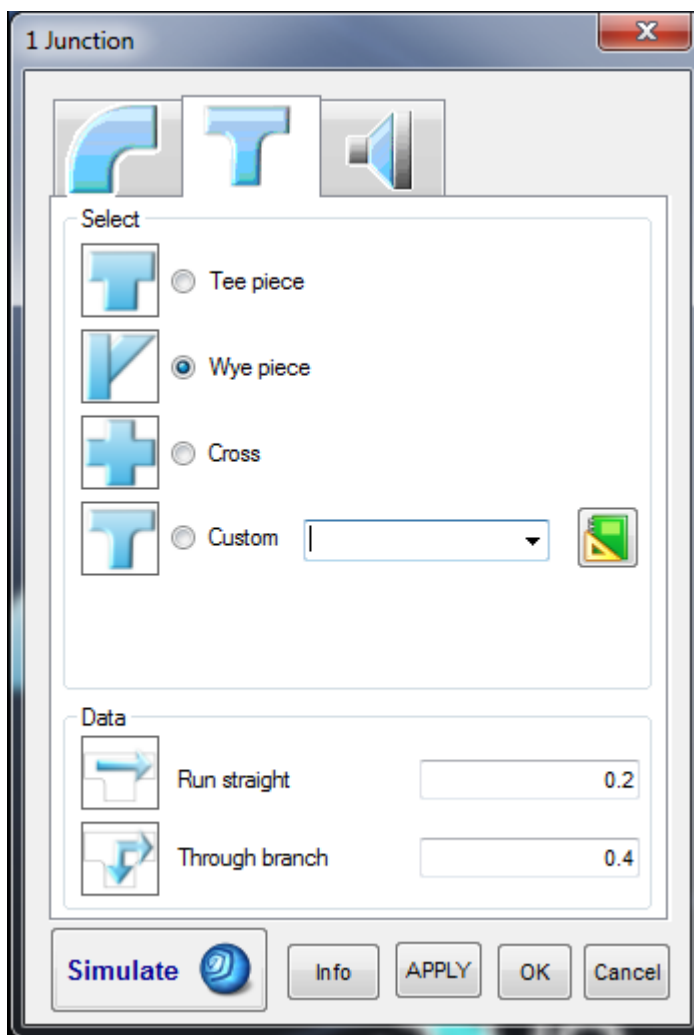


Fig. 3.2.4.2: The three and four-way junction addition box.

Diameter change

The resistance of the junction is automatically calculated from the angle and the two diameters. Pumpsim™ compares the two pipe diameters to determine if it is a contraction or expansion. The ***K factor*** for contraction and expansion is automatically calculated from the two diameters and the angle. They can be created as custom values.

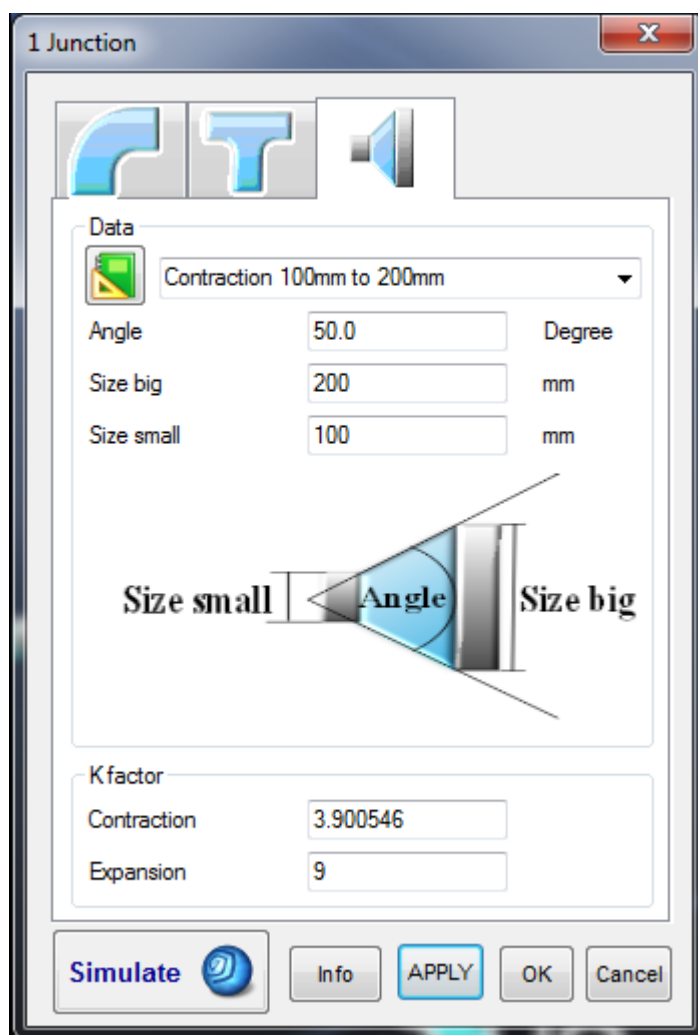


Fig. 3.2.4.3: The K factor (whether contraction or expansion) is calculated from the values inputted into the junction creation box.

3.2.5 Pumps/Compressors

A pump can be added to simulate flow in a model. **Power, efficiency** and the **pump duty point** are calculated and provided in the **EDIT** box.



Edit Box

Displays a form allowing pump, compressor and fixed flows and pressures to be applied to a pipe. If a pump is present, a pump curve graph and duty will be displayed after simulation showing pump performance in a simulated model pipe. If no curve is used for this pump, a blank window will be displayed.

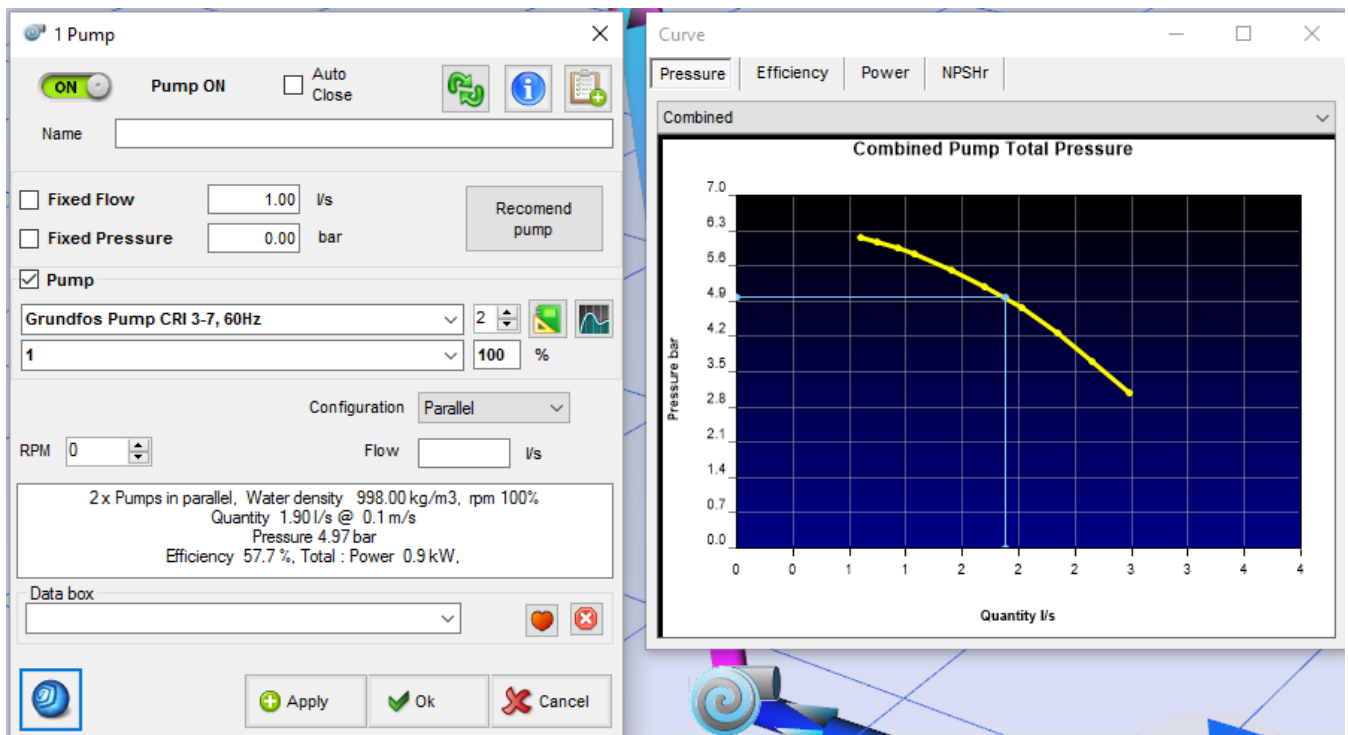


Fig. 3.2.5.1: On the left is a pipe edit box for various applications to a pipe. On the right is a graph depicting pump performance of a pipe that has been simulated.

Fixed Flow

Forces the simulation to produce a flow equivalent to the value entered in the box. **Fixed flows** can either be used to simulate the effect of a pump or a valve, or for forcing flow into parts of a model to reproduce observed flows. A fixed flow cannot be set when a pump is present, and a pump will be removed if the fixed flow is checked. To remove the influence of the fixed flow, simply uncheck the fixed flow check box. When a fixed flow is simulated, the program will calculate a required pressure and power to produce the fixed flow. Where the fixed flow is less than what would normally be simulated, the fixed flow acts as a 'resistance' or negative pressure influence, and restricts flow. In both cases, the results of fixing a flow will be displayed in the **INFO** box.

A fixed flow will force Pumpsim™ to calculate a pressure required to induce flow to the set amount. This pressure can be substantial if the flow needs to be pushed through a high resistance. Conversely, the pressure may actually be negative, if the fixed flow forces the flow to be lower than would otherwise simulate, effectively resulting in the fixed flow acting as a higher resistance.

Hint: If the fix flow is working with compressed air the flow will be free air volume flow.

In all cases, the results of a fixed flow can be observed from the **Edit Box** information function, which describes the pressure, power and resistance required to produce the flow. **This can be directly used to estimate a pump duty point** which would be required to produce the same results. This fixed pressure can be equivalent to **Pump Static Pressure** or **Pump Total Pressure**, depending on the simulation type and location of the pump.

*In general, unless estimating the requirements for a pump, the use of fixed flows to reproduce observable flow is **generally discouraged** as it may adversely affect other parts of a model and does not provide a realistic behaviour for changes in model systems. In many cases it can mask real problems with the model construction.*

Recommend Pump

If a fixed flow is present and has been simulated. The program will examine the pressure and flow required to achieve the flow, and then search the pumps in the program database for a suitable matching and the RPM to achieve that flow. The program will consider up to 4 pumps as being a viable match, and display them in descending match order. It will also display the number of RPM necessary to achieve this flow as well as the efficiency of the pump at that point. To swap the fixed flow for a recommended pump, select the desired pump from the list, the RPM will also be automatically adjusted.

Estimating Pump Pressure Requirements

Fixed Flows are often used to help estimate required pump pressures. The pressure given for fixed flows will be an estimate of the **Collar Total Pressure** – ie the total pressure of the flow in the pipe directly beneath the pump. For a pump with an equal diameter to the pipe, this equates to the **Pump Head requirement**, however caution should be used for this interpretation because differing pump diameters will alter pump head requirements. A pump manufacturer will normally be able to utilise the **collar total pressure** value to an equivalent pump head. The **Pump Head requirement** will be a function of the discharge size and velocity of the pump, as well as any resistance and shock losses in the pump and structure between inlet and outlet. Therefore the pump manufacturer will need to take this into account when selecting a suitable pump. Also note that the simulated pressures provided will be influenced by the simulated pump or Pipe Discharge diameter (evasé) so ensure this is noted and specified when quoting the duty point summary.

It is important to note that fixed flows contribute to model power consumption, in much the same way as pumps. The fixed power within a model is summarised from the **Run Summary** menu item.

When calculating power for a fixed flow, the default pump/fix efficiency from the **Settings** menu is used.

More information on fixed flow and pressure is available from the **Edit Box** functions.

Pump Density

Finally, when specifying a pump duty point based on a fixed flow, it is important that the density the fixed flow was simulated in is noted. Pump performance varies substantially in different densities and a pump manufacturers will need to know the required density so the pump curve can be adjusted for the local condition accordingly. In the simulation, the density is set by the fluid type in the pipe.

Fixed Pressure

Forces the pipe to produce a constant positive pressure equivalent to the entered value in the box. A fixed pressure cannot be set when a pump is present, and a pump will be removed if **Fix Pressure** is checked. To remove the influence of a fixed pressure, simply uncheck the **Fix Pressure** check box. When a fixed pressure is simulated, the program will calculate the required waterflow and power to produce the fixed

pressure. The results of fixing pressure will be displayed in the **INFO** box.

As with fixed flows, fixed pressures consume power, which again is summarised in the **Edit Box** information summary, or from the **Model Summary** menu item.

When calculating power and heat for a fixed pressure, the default pump/fix efficiency from the **Settings** menu is used.

Pump Name, Number and Configuration

Selects a pump. The upper pull down menu allows a pump to be selected from the pump database. Pumps are sorted in the order established in the Preset form. Adjacent to the pump name is the number of pumps to be included in the model, and below this is the configuration of the pumps (parallel or series). Pumps installed in series will increase available pump installation pressure according to the number of pumps available, while pumps installed in parallel will increase available flow.



Pump Database

Open the [pump database screen](#) to view or adjust pump data. Note that any pump changes will not be reflected in the **EDIT** box until after a simulation. Also note that modifying the pump database will change ALL pumps of the same name in other locations in the model. To create a custom change only for the pump at one location, you will need to create a duplicate of the pump in the database, and only modify the duplicate.

***Hint:** Pump operating density is an important specification for designing pump types and installations. Higher water densities will increase the available operating pressure curve (and power draw) for the pump, while lower water densities will have the reverse effect. This can be an important consideration when selecting or designing a pump for a particular area in a site. Note that for Pumpsim™, water densities are assumed fixed at all locations in a site.*

Pump Curve

Pump Duty Point (BLUE lines on Pump Curve Graph)

The pump duty point defines the pressure and flow that a pump operates at, and is shown by blue lines on the pump curve. Note that either an **individual pump duty** can be selected, or if multiple pumps are present at the installation in parallel or series configuration, the entire **combined installation duty** can be displayed.

Due to local water density adjustments, the pump curve may have been adjusted to the one defined in the pump database.

The efficiency returned reflects the efficiency curve entered with the pump. The efficiency curve is the shaft efficiency of the pump. It does not include electric motor or drive inefficiencies, which are considered separately in the Settings menu under Motor Efficiency.

Pumps Stalled, Low Pressure or Negated

If pumps are forced by other model conditions to run beyond the limits of the pump curve, a warning showing

stalled, low pressure or **negated** will appear during model simulation.

Stalled pumps occur when the pump pressure build-up is greater than the capacity of the pump curve to accommodate. Pumpsim™ reduces the flow quantity to a point where the pump pressure is no more than the maximum pressure specified in the pump curve.

Low Pressure pumps occur when a pump pressure falls below the lowest pressure point of the pump curve, but is still operating above a zero pressure. This may occur when a pump is run with too little resistance or the pump is a high pressure type, being used in a low pressure application. The pump pressure curve in this case may not extend down to the point the pump is operating at. This is undesirable as the program must make assumptions as to what duty point the pump is actually running at. In addition the pump is unlikely to be running efficiently at this duty point. To prevent this warning, simply extend the pump curve to a lower pressure point.

Negated pumps occur when the pump offers no useful pressure to the system, and may even retard flow that would otherwise flow more freely without the pump. This may be a result of other model factors or pumps forcing or drawing water through the pump.

When Pumpsim™ encounters a low pressure or negated pump situation, it applies a resistance for flow through the pump above the highest curve quantity. This mimics real life performance of pumps and effectively restricts flow moving through a pump above its limit. The induced resistance results in an additional pressure drop and in negated situations, the pump may apply a nominal negative or 'resisting' pressure, effectively acting as a 'brake' or orifice for the flow.

Any of these situations are not desirable, as pumps are not designed to run beyond their pump curves. Without a pump curve Pumpsim™ is forced to approximate pump performance and power consumption. While stalled, low pressure and negated warnings can be ignored, every effort should be made to reduce or eliminate these occurrences within a model.

Pump Pressure Curve

The pump pressure curve associated with the pump (or pumps if **combined** is selected). Note that the pump curve may have been adjusted by the simulation from the curve entered in the pump database to reflect the change in water density at the pump location. The pump operating duty point is shown at the intersection of the horizontal and vertical lines.

Pump Efficiency Curve

The efficiency curve and the flow volume intersection point define the efficiency the pump is operating at. The efficiency value is used to calculate the estimated pump shaft power. If pump shaft power is entered in the database directly, a pump power curve will be shown and the program will derive shaft power from this curve instead.

Pump Power Curve

If a power curve has been entered for the pump, the power curve and the flow volume intersection point will define the shaft or absorbed pump blade power the pump(s) are operating at. Pumpsim™ will preferentially use this curve to calculate power if present. If the power curve has not been entered for the pump, the efficiency curve will be used to calculate power. If this is not present, the default efficiency in the settings

menu will be used.

Pump NPSHr Curve

To operate without cavitation, the pump requires a minimum pressure at the entrance. If the pressure is lower than the NPSHr, Pumpsim™ will show a warning. Cavitation could make the result of the simulation less precise. The result of a model with cavitation should not be used.

Pump Database

The pump database is accessible from the **TOOLS > PUMPS** menu. The pump database allows editing, adding and deleting of all pumps in the model pump database. Up to one thousand (1000) pumps and the associated pump curves may be entered into the pump database. A display for each pump curve and data will be presented when a pump is selected from the display list.

The **COPY** and **PASTE** functions can be used to copy data to or from another program (for example a spreadsheet).

Conventional Pump

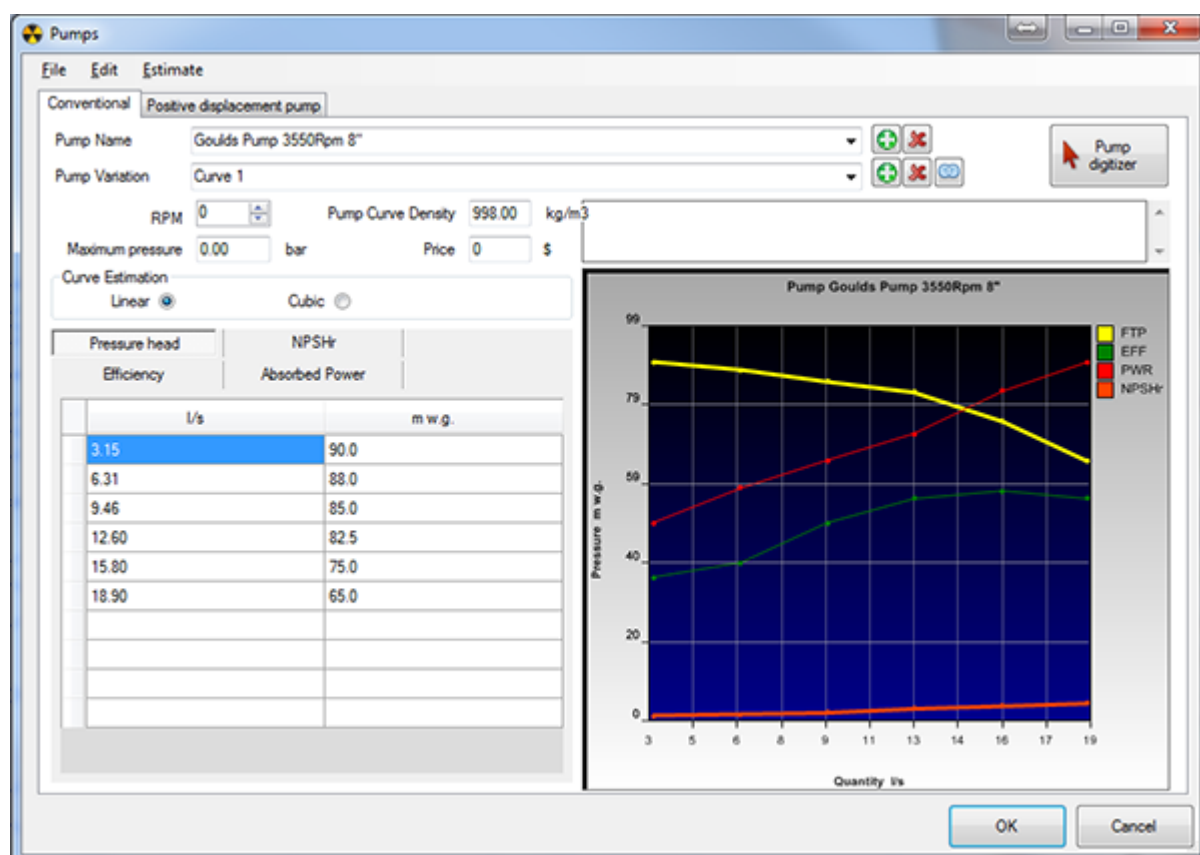


Fig. 3.2.5.2: The pump database. Pump information may be seen from this database; data as well as a pump curve graph is displayed.

Caution should be taken when deleting or modifying a pump, as any model using a deleted pump number will not simulate correctly.

The pump name is entered or chosen at the top from the drop-down menu. To enter a new pump, select **File > New**

At a minimum, pump curve points for quantity and pump head must be entered for each pump. Other curve information such as **efficiency**, **power** and **NPSHr** can also be entered.

%RPM

Adjusts the pump rotation speed from the default setting defined by the pump curve in the pump database.

Note that this is a theoretical pressure and flow adjustment which may not exactly meet true pump performance at different speeds.

Hint: While the simulated speed adjustment to a standard pump curve should be reasonably accurate, if an exact pump pressure and flow curve is required for pump performance at different revolutions, it is suggested a separate manufacturer-guaranteed pump curve be entered for the adjusted variation.

Pump Curve Density

Optional parameter, which will assume a default value if not specified. Most manufacturers supply pump curves at a standard density, however different densities can be entered for the pump curve if required.

Pump speed

Pump speed is a manufacturers reference to the normal speed of the pump operation for the specified pump curve. The number is not directly used during simulation and will not influence the simulation outcome. The percentage pump speed can however be adjusted in the EDIT box.

Curve Estimation

The method used to estimate curve data between specified pump duty points. The **cubic spline** method estimates a curved data path between pump points. If only a few points of data are available, this may produce a better estimate of pump duty, however the method may be limited by sudden changes in curve data point direction. Ensure sufficient points are available to produce a smooth non-reversing curve.

The **linear method** predicts a straight path between points. This method is slightly faster during simulation, and if the maximum number of pump points (10) are entered, it should provide sufficient accuracy in most cases.

***HINT:** It is important that pressure on the curve is not permitted to bend over in a U shape (or upside down U), otherwise the simulation may oscillate between two pressure points. Stall regions of pumps should be omitted for this reason.*

Comments

A **comment box** is available to include more information about the setup or configuration of the pump. It is not used for simulation.

Point Table

The **Pump Point Table** will allow direct entry of pump curve data. Pump curves will be constructed as data is entered. Points can be submitted non-sequentially, and will be automatically rearranged when the pump is re-loaded or saved.

As a minimum, Pumpsim™ requires flow and pump head. Other information such as efficiency or power can be calculated by Pumpsim™ using default settings, however it is recommended to enter one of these values if available to enable more accurate calculations.

To calculate pump power within a model, Pumpsim™ needs either a pump efficiency curve or a pump power curve. If neither of these curves is available, the default pump efficiency will be used from the **Settings** menu. If both efficiency and power curves are entered, Pumpsim™ will preferentially use the pump power curve to calculate absorbed pump power.

Positive Displacement Pump

Positive displacement pumps are designed to work at high pressure with little influence on the flow. The flow changes linearly depending on the RPM.

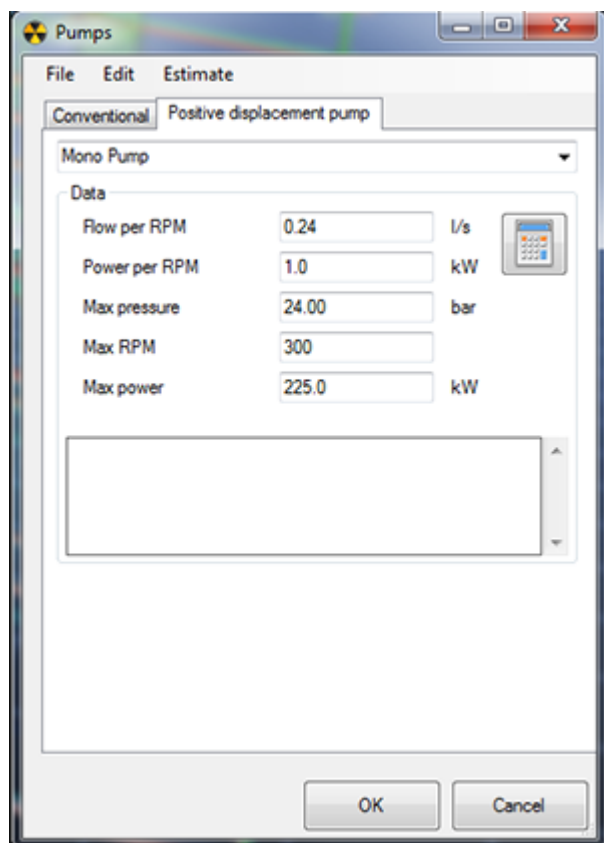


Fig. 3.2.5.3: Data for a positive displacement pump.

- **Flow per RPM** – The amount of flow generated per RPM. The calculator can be used to calculate it.
- **Power per RPM** – The amount of power generated per RPM. The calculator can be used to calculate it.
- **Max pressure** – Maximum pressure at which the pump will stop working properly. A warning will show if the pressure is exceeded.
- **Max RPM** – Maximum number of RPM.
- **Max power** – Maximum power at which the pump will stop working properly. A warning will show if the power is exceeded.

Pump Database Menu

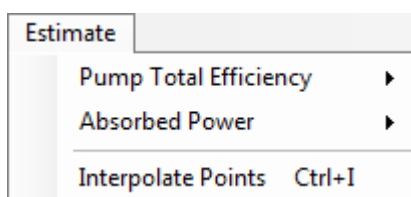


Fig. 3.2.5.4: The estimate tools menu.

Estimate Tools Menu

The tools menu contains a number of functions to assist in estimating pump **Total Efficiency** and pump **Absorbed Power** if either one is unavailable.

To estimate absorbed power, Pumpsim™ will calculate theoretical power using total head and flow. Note that none of these estimation methods take into account shock losses, resistance and compression factors which will slightly affect calculated pressure and powers. It therefore should be used as a guide only, and is no substitute for an accurate pump manufactures curve if available.

Estimate Pump Curve (Interpolate Points)

Quickly estimates 10 evenly spaced points of pump curve duty from as few as 3 entered points. The estimation method will use the Cubic Spline method to add the additional points. This method should only be used if additional pump curve data is not accurately available.



Compressor Database

Compressors usually do not have a curve like pumps. To simulate them, enter the operating and maximum flow and pressure which must be available from the manufacturer. Pumpsim™ will then create a pump curve using those points.

Compressor

File Edit Estimate

Compressor

Selection

Compressor

Data

Free Air Delivery	100.00	cfm
Operating Pressure	10.00	bar
Max Free Air Delivery	150.00	cfm
Max pressure	12.00	bar
Cost	500	\$

Ok Cancel

3.3 Fluids

Pumpsim™ can simulate a variety of fluids, newtonian (water), non newtonian (cement) and compressible fluids such as air. It is possible to have several fluids in one model but two consecutive pipes need to have the

same fluid.

Newtonian fluid

Newtonian fluids are defined by their density and viscosity. Pumpsim™ is pre-configured with some common fluids by default but more fluids can be added in the presets.

A slurry calculator is also available to simulate Newtonian slurry. First select the Liquid Data, either from the database or enter some custom values directly. Then enter the Solid data characteristics. The calculated density and viscosity results will be entered in the table by clicking Apply.

Slurry calculator		
Liquid data		
Select Fluid	Fresh Water	
Liquid density	998.00	kg/m3
Liquid viscosity	1.02	mPa.s
Solid data		
Solid density	2,500.0	kg/m3
Weight concentration	10.000	%
Result		
Density	1,061.8	kg/m3
Viscosity	1.15	mPa.s
<input type="button" value="Apply"/> <input type="button" value="Cancel"/>		

Non Newtonian fluid

Non newtonian fluids are fluids where the viscosity varies with the applied stress or force. Two types of non Newtonian fluids are available: Bingham plastic and Power law.

- **Bingham plastic** fluids are defined by their density, plastic viscosity and yield stress. Cement and paste fill are common Bingham plastic fluids.
- **Power law** fluids are defined by their density, apparent viscosity and flow behaviour index. Glycerine and oobleck are common power law fluids.

Compressible fluid (gas)

As gas density is variable depending on the pressure, different data needs to be entered to calculate the gas density during the simulation using the ideal gas law: the specific gas constant. The specific heat will be used when we include heat in the simulation.

3.4 Scenarios

Scenarios (Pumpsim™ Advanced only)

Scenarios are a feature of Pumpsim™ Advanced and are used during the dynamic simulation in order to automatically trigger actions in the model. Many scenarios can be added to a simulation. You can choose which scenario to run from the main Scenarios form.



Adding Scenarios

Scenarios are used during the dynamic simulation to simulate scripted events. Different scenarios can be created and selected in the main screen.

Scenarios can be renamed by changing the name in the menu.

	Id	Event						
▶	0	If Lower tank water level below 1.0 m then stop Main pump.	⚙️	+	✖	📄	⬆	⬇
	1	If Lower tank water level above 4.0 m then start Main pump.	⚙️	+	✖	📄	⬆	⬇
	2	If Upper tank water level below 2.0 m then stop Jumbo drill.	⚙️	+	✖	📄	⬆	⬇
	3	If Upper tank water level above 3.0 m then start Jumbo drill.	⚙️	+	✖	📄	⬆	⬇
*			⚙️	+	✖	📄	⬆	⬇



Edit event.



Add a scenario or event.



Delete a scenario or event.



Copy and edit a scenario or event.



Move up

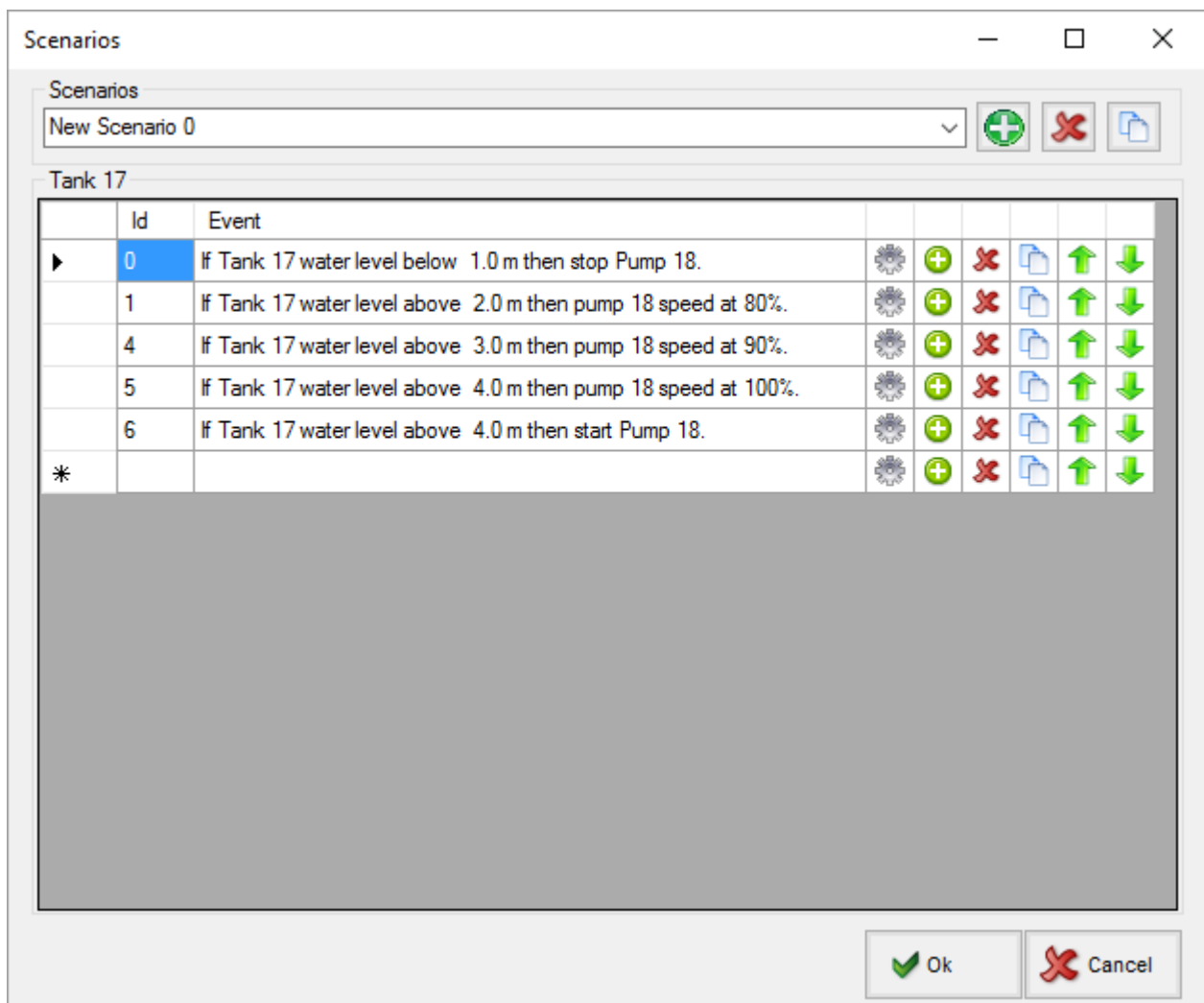


Move down

Right clicking on the scripts provides the following options:

- **Show same source only:** show events which contain the same source object. Example: show all elements with "Upper Tank".
- **Show same action only:** show events which contain the same action object.
- **Show all**
- **Copy:** copy one or several events.
- **Paste** (in any scenario)

***Hint:** The order of events is important; Pumpsim™ will execute each event in order. In the example below, the pump will run at different speeds depending on the tank water levels (80% between 2m and 3m, 90% between 3m and 4m). The order of events can be changed by moving events using the arrows or the keyboard arrows.*



Add/Edit Events

Edit Scenario

Event

Above

Lower tank

Tank: 17

Water level: 4.0

80 %

Action

Start

Main pump

Object: 18

If Main pump water level above 4.0 m then start Main pump.

Ok Cancel

Events:

- **At:** the event will happen at a specified time.
- **Above/below:** refer to the water level of a tank. Can be set using percentage or height.

Actions:

- Start/stop a pump or demand node.
- Open/close a valve or pipe.
- Change the speed of the pump. *Caution: it does not start the pump.*
- Change the opening percentage of a manual valve.



Click to select an object on the screen. It will exit the selection after clicked.



Find the selected element.

4. Building A Model

A model must have a number of key components to successfully run a simulation.

Some simple rules for constructing a model are:

- All pipes in a model must be connected at both ends to another pipe, unless connected to the surface or to a tank or a to spray or 'blanked' off as a closed end.
- Any fixed flow must not restrict another fixed flow elsewhere in a model, or be overly restricted by an impassible high resistance.

Errors contributing to the above conditions account for around 90% of simulation errors and problems noted in Pumpsim™ (from Chasm Consulting observations)

Overview

There are many different ways to build a computer model. Pumpsim™ utilises a visual approach to creating models, and the fundamental structure (shape) of models can be built either by hand with the mouse or imported from a CAD program.

While a model can be built schematically in Pumpsim™, it is highly recommended building a **true-to-scale 3D model** if possible to allow Pumpsim™ to automatically use parameters such as size, length and depth for simulation.

The information in this chapter explains only the basic techniques to establishing working models. To create a truly representative model, pipe sizes and resistance must be accurately established and many other important factors such as junctions considered.

For further information:

- Check the [Pumpsim™ website \(www.pumpsim.com\)](http://www.pumpsim.com) for newsletters and tutorials.
- Seek a suitable Pumpsim™ training course to undertake advanced training in the product.

4.1 Initial Model Construction

Pipes can be constructed in a number of ways.

- Manually draw scaled pipes with the mouse using the toolbar functions.
- Manually draw schematic pipes with the mouse using the toolbar functions.
- Import pipe data from a formatted spreadsheet using the TXT import function.
- Import graphics data from a **DXF / DWG / Datamine / Surpac** graphics file into Pumpsim™, and convert

the data to pipes. Most site planning and CAD packages will have the ability to export data to one of these formats.

Regardless of the method used, in summary the recommended steps for constructing a Pumpsim™ model are as follows:

1. Construct the pipes either manually using the **DRAW** function, or via the **IMPORT** function, ensuring all pipes join correctly to each other. Filter tools are available in Pumpsim™ to assist with ensuring pipes are joined together.
2. Add tanks and use the **EDIT** button to edit the pipes that are open, and mark **OPEN** option on the form.
3. Again, using the **EDIT** button, set the correct size and shape for all pipes. Insert any flow controls or regulators in the model, and specify pipe characteristics such as roughness and resistance.
4. Place nodes to increase the accuracy of the model.
5. Finally, press the **SIMULATE** button to show the result of the model construction. If everything has been constructed correctly, the flow data and arrows should show the result of the simulation.
6. If some pipes are dead ends, then simulation warnings can be prevented by setting the **"CLOSE"** option in the pipe **EDIT** form.
7. If any error or warnings show, then correct the mistakes individually or grouped as required.

4.2 Manual Scaled Construction

To manually construct pipes, simply use the **construction tools** (the **draw**, **move**, **copy** and **delete** buttons) from the toolbar.

Pipes can be drawn freehand on the screen with the mouse, using the coordinates displayed in the status bar to guide the pipe location. Additionally, coordinates for pipes can be entered manually using the **coordinate entry** function or by simply clicking on a pipe end while in the draw mode.

This method is generally suitable for small models (less than several hundred pipe segments). Most pipe models are generally fairly tolerant of misaligned or slightly misplaced pipes, providing approximate pipe lengths are close to reality. In many cases, to assist with clarity, pipes can be deliberately moved aside, and a fixed length function used to override the automatic length calculation to set actual lengths.

If the model is required to be true to scale, then more accurate construction may be required, and it is often better to import actual mine designs from CAD programs into Pumpsim™ to use as a template to constructing a true to scale model.

4.3 Manual Schematic Construction

In some cases, a model schematic can be constructed to simulate a model. A model schematic may look nothing like a network, but simply represent pipes by a series of conveniently located two dimensional lines. Correct pipe lengths can be entered by fixing the length under the **Edit Box**.

Warning: *Schematic Models, although convenient in simplifying models, have major restrictions which can affect their functionality in Pumpsim™. Pumpsim™ uses true pipe locations to calculate elevation, schematic models are largely unsuitable for Pumpsim™ models.*

4.4 Spreadsheet Text Import

Imports model data from **TXT** format (text format with fields delimited by a TAB character). This format is widely available through most spreadsheets and can be read into word processors. The standard format for a **TXT** file can be saved from Pumpsim™ under the **Save As** command. Most components (such as pump database and colour settings) can be excluded from a **TXT** file, leaving only the main file component if desired.

A Pumpsim™ text file must have at least the first row **Header** line, which defines the text file as Pumpsim™ compatible and a **Completion** line 'END' (the last line in a Pumpsim™ saved **TXT** file).

Load any Pumpsim™ file saved as a **TXT** file to view the structure in a compatible editor (for example Microsoft Excel™).

4.5 Layers

Layers are a way to individually layer or identify and view parts of a model separately from other parts of the model. Some examples may include a drainage system, a pumping station or any other collection of objects. Note that there is no requirement for the user to utilise viewing layers. They are only present to allow easier manipulation and viewing of a model. The **Mine** example in the **FILE > LOAD DEMONSTRATION** menu shows an example of layers.

Layers work by allowing the user to overlay multiple layers of pipes or information on a screen. By doing this, unnecessary detail can be turned off so only pipes of interest are viewed.

Layers may be made active or displayed using the **Display Manager**. Layers names can be changed from the **Edit Box** by clicking the button adjacent to the layer name.

Primary Layers

Primary Layers consist of up to 250 layers. It is intended that the **Primary Layers** be used for identifying types of pipes. For example, Layer 1 could be Primary Shafts, Layer 2 could be Main Pipes, Layer 3 - Raises, Layer 4 - Stopes, Layer 5 - Minor Pipes etc.

Secondary Layers

Secondary Layers also consist of 250 discreet layers. It is intended that the **Secondary Layers** be used for isolating parts of a model that could be viewed independently from the rest of the model. As mentioned above, this could include working areas, stope, raises, declines or any other feature of interest.

There is no limitation on the number of pipes belonging to a layer, however a pipe can belong to only one secondary and one primary layer. Primary layers and secondary layers can be set and viewed independently of each other.

See the **Site example** that comes with Pumpsim™.

***Hint:** To quickly save and recall a layer state, use the **Save View** function to save and recall views with the **Primary** and **Secondary Layers** settings.*

Using Layers

The easiest way to use layers may be to create the model first, and then edit pipes to change the layers to the desired number. All pipes will default to **Layer 1** of both **Primary** and **Secondary Layers**.

***Hint:** The **Edit Box** has a function which will select and group all similar pipes (for example all round pipes with a diameter of 3.0m). This can provide a quick way to group edit and change layers for multiple pipes.*

New pipes constructed will inherit the layer numbers of the pipe they are constructed from. If no pipe exists, new pipes will use the default layers set in the **Display Manager**.

***Hint:** Because primary and secondary layers can be used together, this creates an opportunity to create a filter using both layer systems. If, for example, primary layers were assigned to different regions of a site (North, South, East and West for example), and secondary layers were defined as different functions (Shafts, Ramps, Ore drives for example), then by individually setting the primary and secondary layers, the following examples could be easily viewed:*

- All ramps in the North site.
- All ore drives in the East and West sites.
- All ramps and shafts in all sites.

Exclude from simulation

A layer can be excluded from the simulation. This can be useful for drawing other parts of the environment not designed for simulation, such as electricity cables.

4.6 DXF Graphics Import

STEP 1 – Create and Import a DXF file into Pumpsim™

This method relies on passing line or string graphics data from a **CAD or Site Planning package** to Pumpsim™. The data may simply be the centre lines of basic piping or site designs. To allow Pumpsim™ to efficiently use this data without excessive post-import editing, the following criteria should be considered:

- **Only import the minimum data required.** Excessive detail or headings which do not form part of the model should not be imported if possible. In many cases, it may be quicker and more accurate to construct a skeleton model consisting of interconnected lines within the Site Planning or CAD package, before importing the model into Pumpsim™. The skeleton lines should simply trace over existing site workings or designs.
- **Try to ensure the lines connect, so Pumpsim™ can identify junctions through which liquid can flow.** Lines that simply cross each other without a junction, or which terminate close to each other but do not join, will not carry flow, and may result in a no exit/entry error in Pumpsim™ which will need to be corrected.

When importing data, the lines can be converted to pipes during the import process by selecting the option **"Convert to pipes"**, or selectively after import using the **DRAW > CONVERT** tool. To use this tool, click or drag a box around the lines which are to be converted. The second option has the advantage of overlaying the updated DXF files with new development over the existing Pumpsim™ pipes, and selectively choosing where to update the Pumpsim™ model.

***Hint:** To efficiently add DXF data to Pumpsim™, keep a 'layer', 'object' or 'file' dedicated to Pumpsim™ data within your Site Planning or CAD package. As new pipes or designs become available, simply export the additions as separate DXF files, and import them into Pumpsim™ to add to the existing model.*

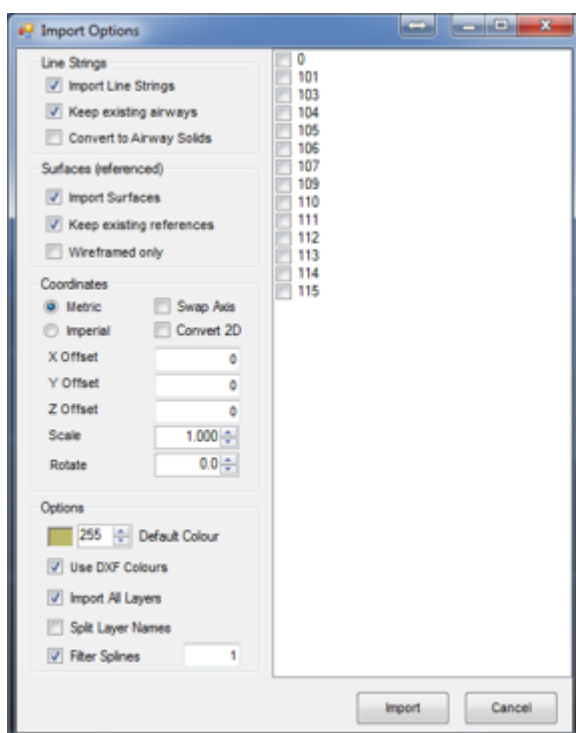


Fig. 4.6.1: The import options box is utilised when importing files into Pumpsim™

Warning: Three dimensional solids (such as survey mine solids) will import into Pumpsim™, but cannot directly be converted to pipes. They can however be used as a guide to manually draw and construct new pipes in the model.

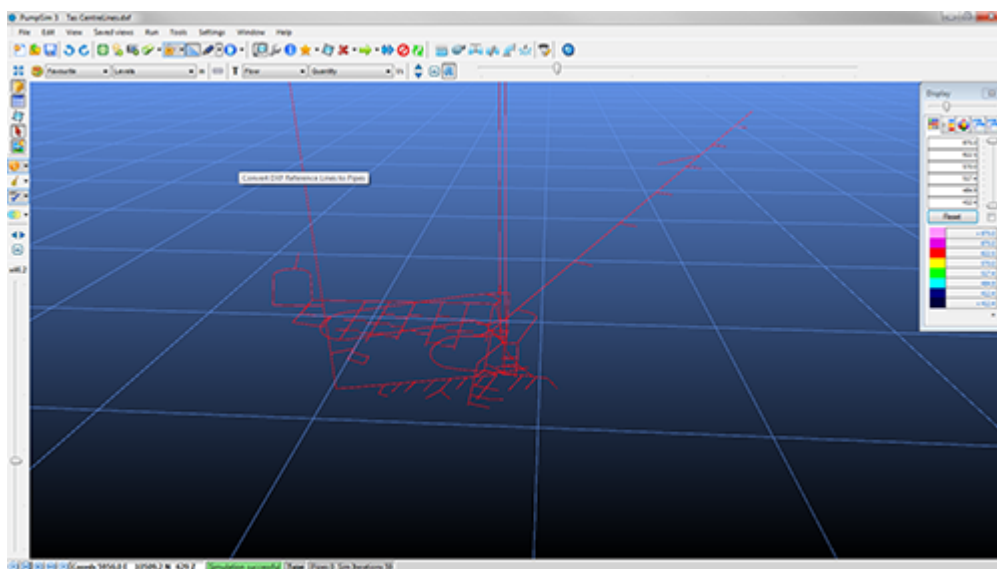


Fig. 4.6.2: Three dimensional solids that have been transported into Pumpsim™.

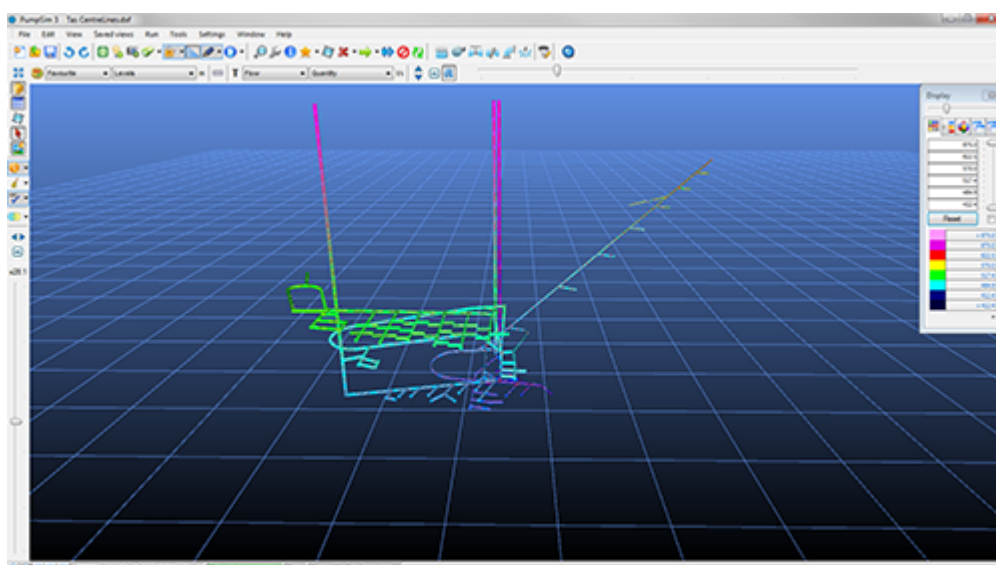


Fig. 4.6.3: Pipes that have been constructed using the three dimensional solids as a line guide.

STEP 2 – Modifying and Validating Import Data

Once imported into Pumpsim™, the program will assign default pipe sizes and shapes to all the imported pipes. Ensure you have default sizes and shapes set in the **Settings** menu that will approximate the typical sizes that are imported. This will prevent having to re-edit of ALL of the pipe sizes and shapes (although this can still be quickly done using the **group edit** command).

Import DXF Graphics Solid or Wireframe Data

Unlike line string data, solid or wireframe graphics cannot be directly converted to pipes. The data can still however be used to help construct pipes, by either using the referenced (imported) graphics as guides to manually draw pipes, or by using a special function in Pumpsim™ to group the data and construct a pipe of best fit through the data.

OPTION 1 – Turn on Edit > Lock on References menu option and select the DRAW button.

Simply use the mouse to DRAW pipes over the top of the imported graphics. The pipes will 'connect' to the same position as the reference graphics (providing the Edit > Reference Lock is enabled), allowing the pipe network to be drawn in true 3D coordinates.

OPTION 2 – Convert ANY (references to pipes). Use the Draw Sub-Option > Convert Any.

This option converts groups of referenced graphics to an expected pipe path, and may be of use to attempt to convert large amounts of wireframe or multiple string (for example outlines) to an expected pipe. It is likely that significant editing may be required after using this function to adjust pipe locations and correct errors. For further information on this method see Convert Any option.

Correcting Errors

To quickly sort and validate the imported pipes, click on **"Flow Simulation"**. While the model is unlikely to correctly simulate at this stage, this function will check the pipes for duplicates and unconnected pipe ends, which will speed up the process of establishing a working model.

In most cases, errors will result from pipes not exactly joined into other pipes, or remaining unconnected as dead ends, resulting in no exit or entry errors. If many of these can be corrected before starting a flow simulation, this will improve the ease of creating a working model.

Filtering Tools

To help tidy up models and correct initial errors, a number of tools are provided in the **TOOLS > FILTER** menu (bind disconnected pipes, find duplicated pipes etc).

Alternatively, for pipes which are disconnected, use the **TOOLBAR > MOVE** function to connect the pipe ends onto other junction ends. If this is done correctly, moving the node after joining will then result in all pipe ends moving simultaneously.

The following tools can be found under **Tools > Utilities > FILTERING TOOLS**

BINDING

- **Bind Nodes** (use a value up to 4-5m or more to bind ends that are close to each other).
- **Bind Junctions** (Binds loose nodes that are close to a passing pipe which has no corresponding node or junction).
- **Bind Intercepts** (find and/or join pipes that cross each other, but have no joining nodes)

SIMPLIFY

Removes a lot of the small pipes and unnecessary detail. In many cases, importing DXF files directly from a site planning will create an overly detailed model file, which will slow down the system graphics and simulation, resulting in a more unwieldy model.

For example, a nice smoothly rounded decline pipe which produces many pipe segments is simply not required for accurate pumping network simulation and in most case can be designed with less segments and detail. The **SIMPLIFY** function in Pumpsim™, will sort through a model, identifying unnecessary detail, which can be removed without adversely affecting simulation.

Alternatively, pipes can be manually simplified by using the toolbar **DELETE** function to remove node junctions between pipes.

DUPLICATES

Find and/or remove any duplicated or overlapping pipes.

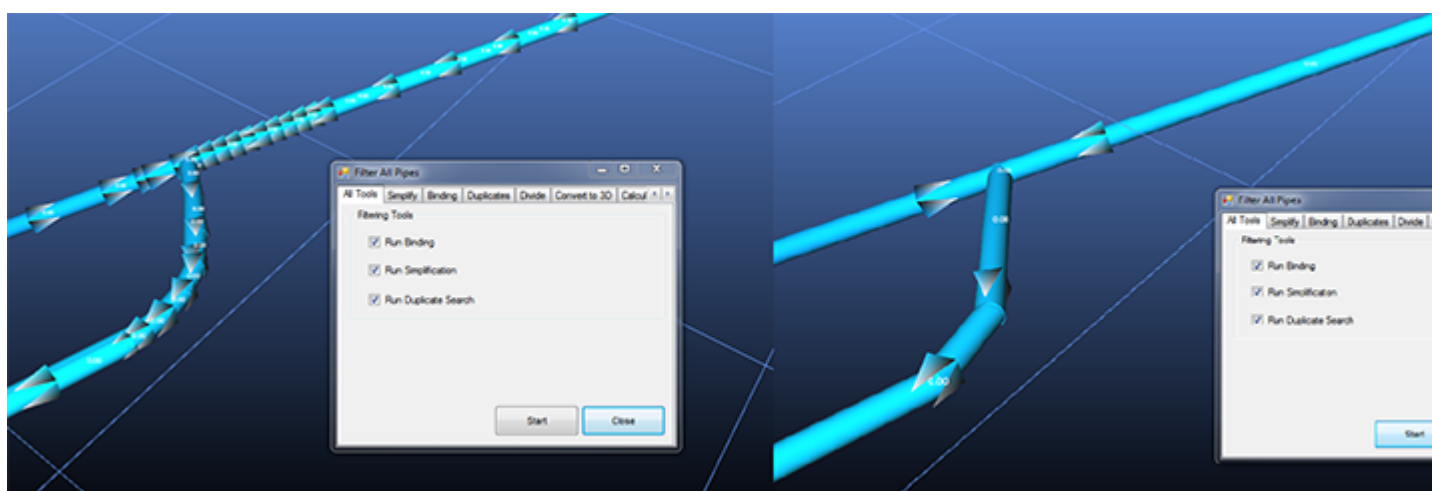


Fig. 4.6.4: Simplification of pipes.

Importing as Reference Graphics

Importing graphics items can serve two functions. Importing line string graphics can allow Pumpsim™ to directly create new pipes, using the line strings as centrelines for the development. The centrelines can be converted to pipes during the **import** function, or later on by selectively clicking or fencing the centrelines with the **Add > Convert** function. Further utilities under the tools menu allows users to quickly turn the new pipes into viable models.

***Hint:** Converting all imported centrelines to pipes can be an issue if some model pipes already exist and may be duplicated by this function. Instead, loading centrelines as a reference graphic without converting to pipes can allow the user to check where extensions or changes to a site model may be, particular if some of the centrelines may overlap existing pipes. The reference lines will extend from existing pipes showing where extensions to the model may be required. The referenced centreline can then be selectively clicked or fenced to convert to pipes using the Add > Convert function.*

A second option allows importing DWG or DXF files as 'references', which places the graphics within the model, but does not affect or interact with the model. Examples may include site surface terrain, orebodies, actual development solids etc. This option allows the reference graphics to be used as a guide to building new pipes, or simply enhances the display with more information about the environment. Any solids or wireframes that are present in the DXF will automatically be imported as a 'reference' graphic, as Pumpsim™ cannot build new pipes from graphical solids (only centrelines).

Imported reference files can be merged with an existing model, and new pipes built with DXF centre lines can be set with default attributes (pipes sizes etc) before importing. Attributes can easily be changed later within the program.

***Hint:** File imports support drag-and-drop. Simple grab the file from a Windows folder or directory and drop it on the Pumpsim™ screen. More than one file can be imported at a time by using the SHIFT or CTRL key to select and load the files.*

Import Options

Pumpsim™ will search a DXF file for both line strings and solids, as well as layers within the DXF file. The import options indicate what action to take if these items are found.

Layers

The right hand column lists available layers in the DXF file which may be imported. Only select the layers required to be imported into Pumpsim™.

Centrelines

Centrelines (lines or polylines) can be either converted to pipes, which will carry flow within a model, or simply as a reference graphical line string, which will not affect a model, but which can be used to help guide and place manually constructed pipes.

- **Import Centrelines** – Disabling this function will tell the program to ignore any line strings found.
- **Merge Data** – Adds the new imported data to the current model
- **Import as reference only** – Tells the program NOT to build pipes out of centrelines, but simply import as a graphical reference. A centreline can be converted later to a pipe using the Add > Convert function.
- **Surfaces (referenced)** – Surfaces or 3D solids cannot be directly converted into pipes (unless a 'convert all' option is attempted), however 3D data can provide a useful reference to manually developing Pumpsim™ pipe. In addition, it can provide an important visual aid to viewing models within an actual environment.
- **Import Solids** – Disabling this function tells the program to ignore and not load any solid graphical data found in the reference file.
- **Merge References** – Adds the imported data to any existing referenced solid graphical information already within the model.
- **Wireframed Only** – Adds the imported data as a wireframe line solids, instead of a polygon solid.
- **Metric or Imperial Coordinates** – Most imported files do not have an internal specification of whether the units are Imperial or Metric. Pumpsim™ will assume the coordinates are the same as currently used in the Pumpsim™ file, however if the coordinates system are mismatched, then the imported data will be displaced to incorrect coordinates (for example a metric DXF imported into an imperial Pumpsim™ model). This setting allows you to override the default Pumpsim™ setting to ensure coordinates will match up.
- **Offset X, Y, Z** – Allows a DXF file to be imported with an offset from the original file coordinates. This may be useful to match existing coordinates used in Pumpsim™, or when a 2D file single elevation file needs to be moved to a new elevation in Pumpsim™.
- **Convert 2D** – Flattens the entire DXF file on to a single elevation or level. Occasionally, flat 2D CAD files may have different elevations set in the file. When these files are used in a 2D CAD package, the elevations difference may not be noticed, however when loaded into Pumpsim™, the elevation differences may become troublesome in a 3D working environment. If the OFFSET Z option is entered, then the imported elevation will be at this number when this option is set.

- **Scale** – Allows the DXF import data to be scaled to a different size. This is sometimes useful if the original DXF data has been scanned or converted from a PDF file at an arbitrary scale and no longer matches the true scale.
- **Rotate** – Rotates the DXF data around a zero based origin by a specified number of degrees.
- **Default Colour** – Allows the user to select the colour of the reference data to import.
- **Use DXF Colours** – Colours the imported graphics to the original colour (if selected), or the user selected colour (if not selected)
- **Split Layer Names** – Imports the reference graphics into Pumpsim™ with each layer as a different reference name, instead of a single reference name encompassing all layers. The reference names can then be individually managed in the Reference Manager function under the File menu.
- **Filter Splines** – Imported splines (smooth curves) can have hundreds of data points to make smooth curved lines. This can consume excessive memory in Pumpsim™. Select this function to reduce the imported data requirements, and approximate the curves with a series of straight lines.
- **Import STR (Surpac Strings)** – Similar to DXF functionality, Pumpsim™ will attempt to import line strings and offer to directly convert to pipes, or import them as a reference for later use.
- **Import DM (Datamine Strings and Wireframe Solids)** – Similar to DXF functionality, Pumpsim™ will attempt to import line strings and wireframe solids from Datamine compatible files.

4.7 Simulation



Steady State Simulation

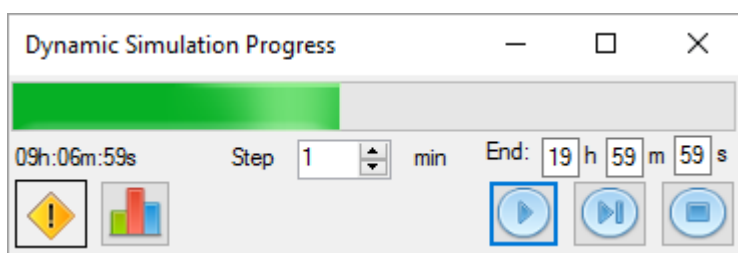
The **Steady State Simulation** simulates the model with fixed characteristics. Tank water levels stay the same during the simulation. If the simulation is not successful, an error message will appear to try to help resolve the issue. The simulation can also be successful with warnings, which imply that something during the simulation was abnormal. Warnings can be due to design issue, too high/low pressures, or due to the model (eg. pipe not properly connected). See appendix E for more detail.

The Pump Priming Simulation simulate the part of the model containing a pump and check if they will be able to run without the help of suction. It is advise to run this tool if the pump needs to pump uphill and then downhill. The normal simulation might show that with the help of the suction on the downhill side, the pump will work properly but until the pipe is full of water, there won't be suction.

***Hint:** It is important to have a correctly functioning **Steady State Simulation** before starting the **Dynamic Simulation**.*



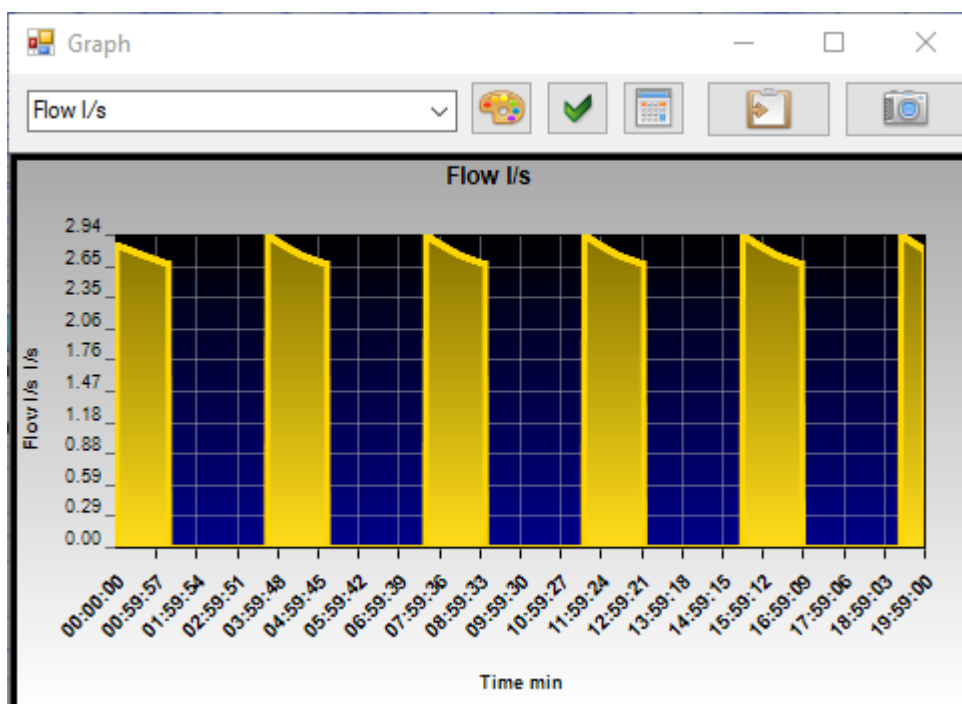
Dynamic Simulation (advanced only)



The **Dynamic Simulation** will be launched when the **Simulation** button is clicked. For each time step, a simulation will run and the tank water level will change depending on flow in and out of the tank. The selected scenario can be changed in the toolbar.

The **Time Step** is a **maximum** time step; meaning that it can be reduced if an event occurs within the time step. In this case the time step will run until the event and stop. Example: the time step is 1 minute but a tank will be full after 30 seconds, so this time step will be reduced to 30 seconds.

After the simulation, the **chart** can be displayed by clicking on . Data will be recorded at every fitting (pump, valve, tank etc). The user can also place monitors on the model. The axis range can be adjusted by right clicking on the graph.



Change the color of the graph. The color will be kept for future graphs.



Update with the latest simulation data.



Different data can be displayed as well as a summary.



Copy the data to the clipboard. It can then be copied to other spreadsheet programs, for example Microsoft Excel.



Takes a snapshot of the graph.

4.8 FAQ

Below are some of the common issues found when constructing models in Pumpsim™. For more help when a warning or error shows, please refer to Appendix E.

The pipe has no flow.

Some common reasons are:

- Presence of an excluded pipe in the line. Select all the pipes in the section and click on **EDIT**, the **"excluded"** check box should be unchecked.
- Some pipes might not be properly connected. Use **SELECT > ALL** to see if all the required pipes are connected.
- The inlet pipe could be connected above the water level of the tank.

More flow is leaving the tank than entering.

During the steady state simulation, all flows are based on the system as it is, the water level will stay the same. It provides data from the tanks at a specific level. The **INFO** box of the tank will give you information about the filling/emptying rate of the tank. A warning can be shown by changing the tank tolerance (**Settings > Flow Simulation**); a warning will show if the tank will empty or fill quicker than the tolerance. Setting the tolerance to 0 shows no warning.

During the dynamic simulation, the water level will rise until full, then the water will flow out and be lost from the system. Once empty, no water will come out of the tank.

Why is there cavitation?

Cavitation occurs when the pressure is too low and water vaporizes. It is usually due to a high pressure drop in pipes, which is often linked to a high flow rate. It is likely that a valve or device is missing in the line, often at the end. For example, an open pipe might be missing a demand point or a valve.



It is important to note that the results given when cavitation occurs are likely incorrect. A cavitation problem needs to be fixed for a correct simulation.

How can there be negative pressure?

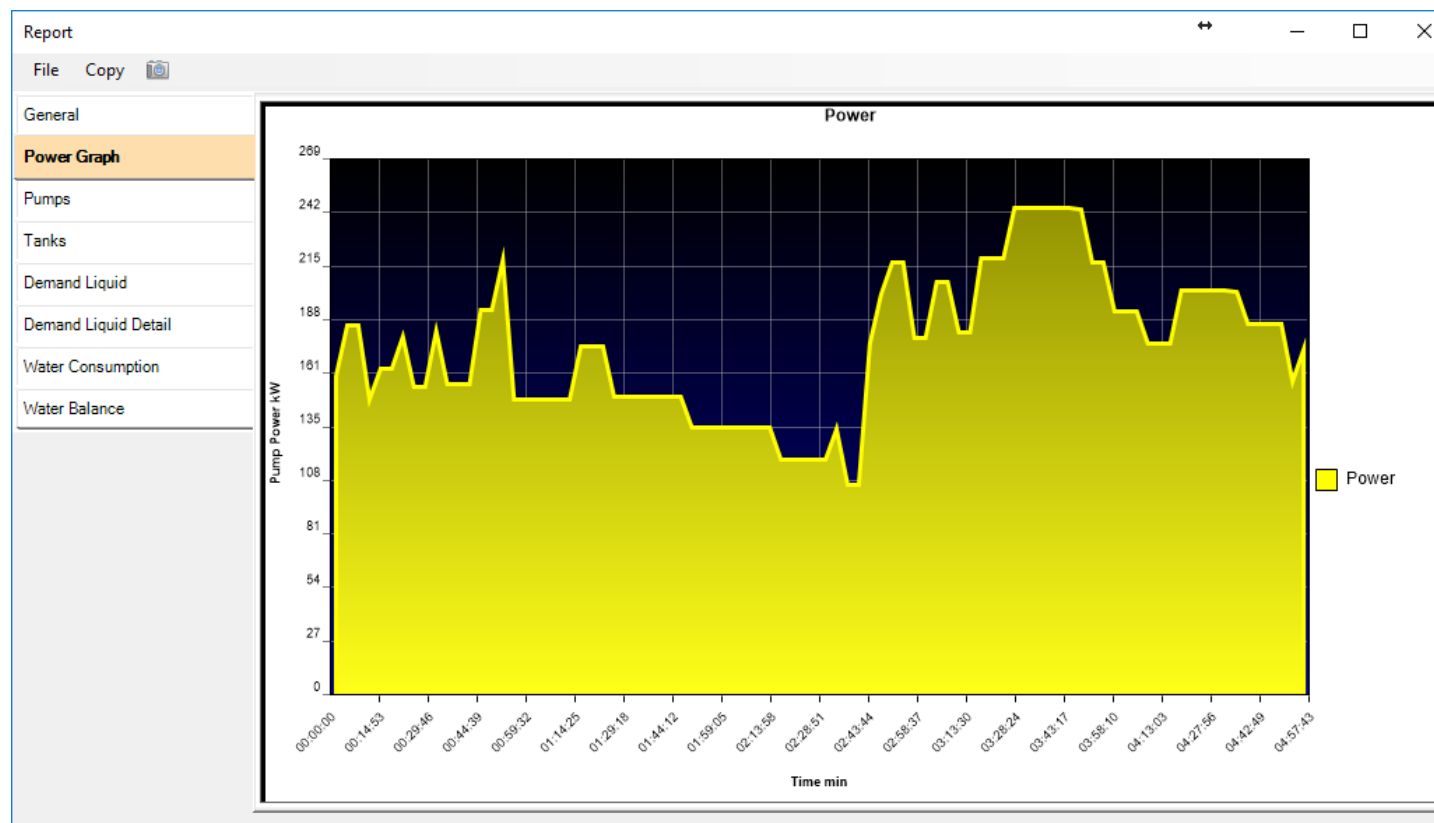
The pressure is a **gauge pressure**, it is the absolute pressure minus the atmospheric pressure.

4.9 Dynamic Simulation Report

Dynamic Simulation Report

The dynamic report is a summary of the dynamic simulation. It allows a general overview of the model and helps identify possible issues in the system. All data can be copied to the clipboard using File>Copy to clipboard. Values can be sorted to highlight a specific issue.  Opens a graph of the selected element.  Centers the screen on the selected element.

- **General** – Gives the total flow and power through the model
- **Power** – Graph of the power consumed over time



- **Pumps/Compressors** – Lists the pumps in the system, displaying the number of times each pump switched on/off and the percentage of the time the pump was on.
- **Tank/Receiver** – List the containers in the model, displaying the number of times each one empties

and the percentage of the time where the container is filling up.


- **Demand** – List all different types of demand devices and display the amount of water consumed for each.
- **Demand Detail** – List all demand points with their minimum and maximum pressure.
- **Water Consumption** – Plot the volume of water used over time.
- **Water Balance** – Plot the amount of water lost or gained by the model.

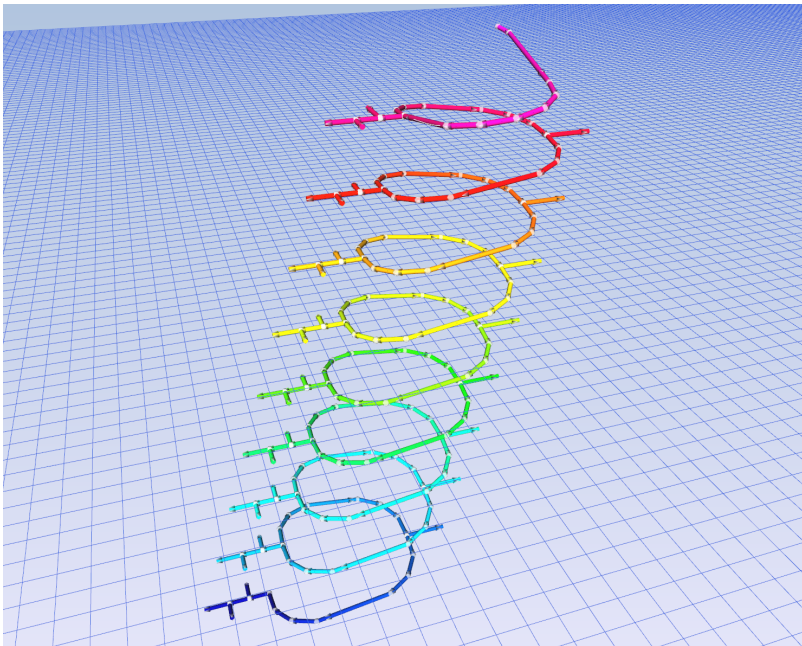
4.10 Step by step guide

Step by Step Guide




In this guide we will be building a model using a simple dxf with centreline. It will include water supply and de-watering. This model is built during standard Pumpsim™ training.

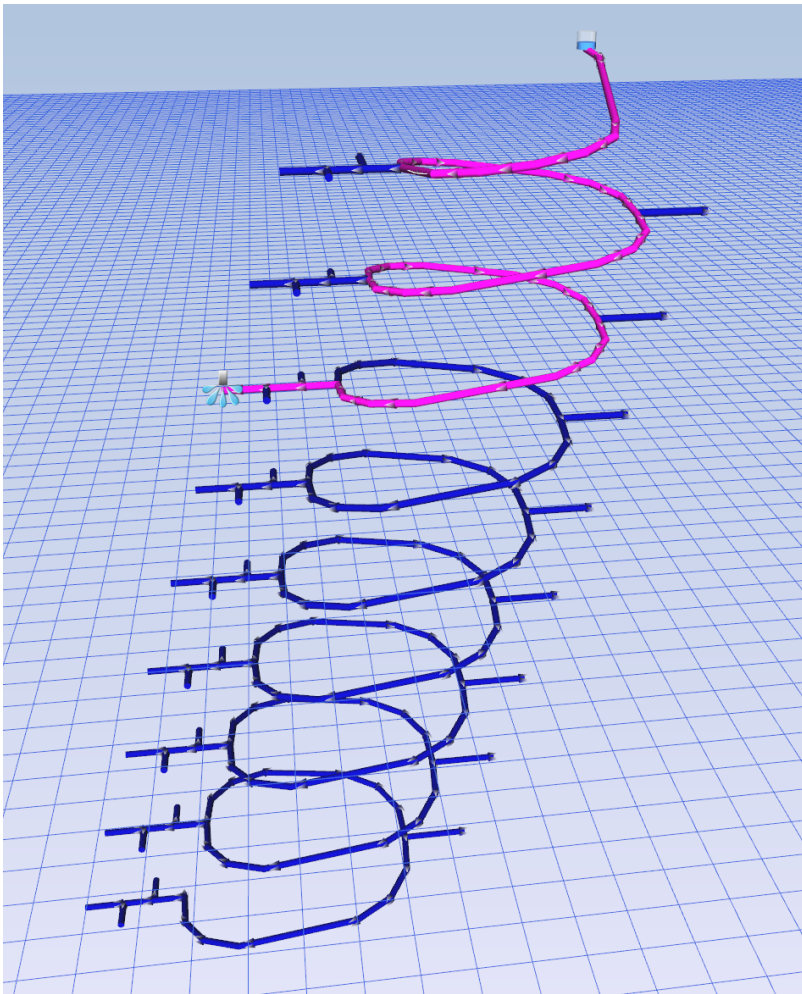
Building the Main Skeleton using DXF


1. First go to File>Import and select your dxf file.
2. Click on import.
3. To transform all centreline to dxf (pipes?), go to the draw tool  drop down and select convert centreline.
4. Loop around the model to create pipes on the whole model.
5. It is good practice to run a simplification/binding tool after importing from a centreline dxf, as the pipes might not all be connected. To do this, go to Tools>Filter and simply run with the default setting.

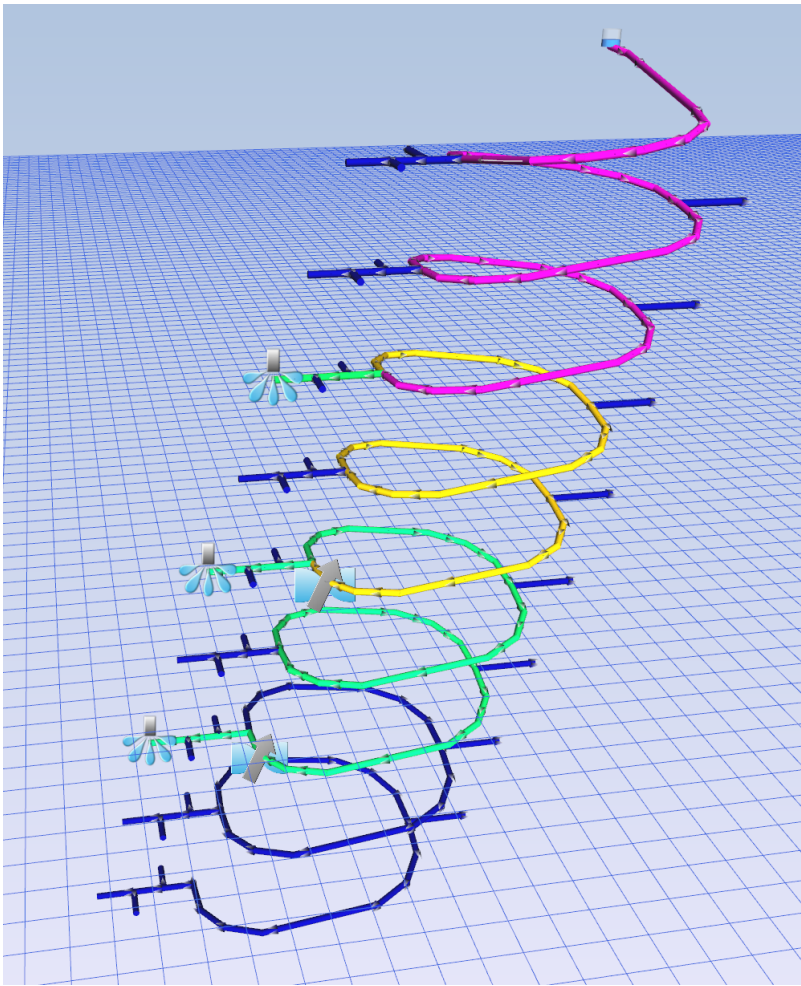


Water Supply

1. First place a tank at the top of the model by clicking on the pipe end.
2. Place a demand device  on a loose end in the middle of the model. See screenshot.
3. To add a Jumbo for example, click the preset button  and add a Jumbo. If unsure about the discharge coefficient, click on the calculator and enter operating pressure (7 bar) and flow (5 l/s). Minimum pressure 2 bar and maximum pressure 10 bar.
4. Then select the Jumbo in the drop down and press OK.
5. Simulate . There should be a lot of warnings, they will be due to the end pipes not being set to open or closed. To close them all, click on select all in the warning box and edit.
6. Select close in the edit box. The simulation will now run fine.
7. Select the edit tool and loop through the model to edit pipes. Change the pipes to Poly PN12 100 mm.



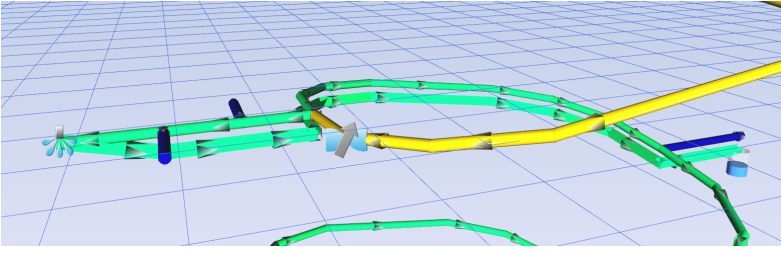
8. Add 2 more demand devices at the lower level and run a simulation.
9. Warnings should appear showing that the pressure is too high. We will use a Pressure Reducing Valve (PRV) set to 6 bar to reduce the pressure.
10. Click on the automatic valve tool , and add it before the junction going to the jumbos. Select the PRV option at 6 bar. Simulate.



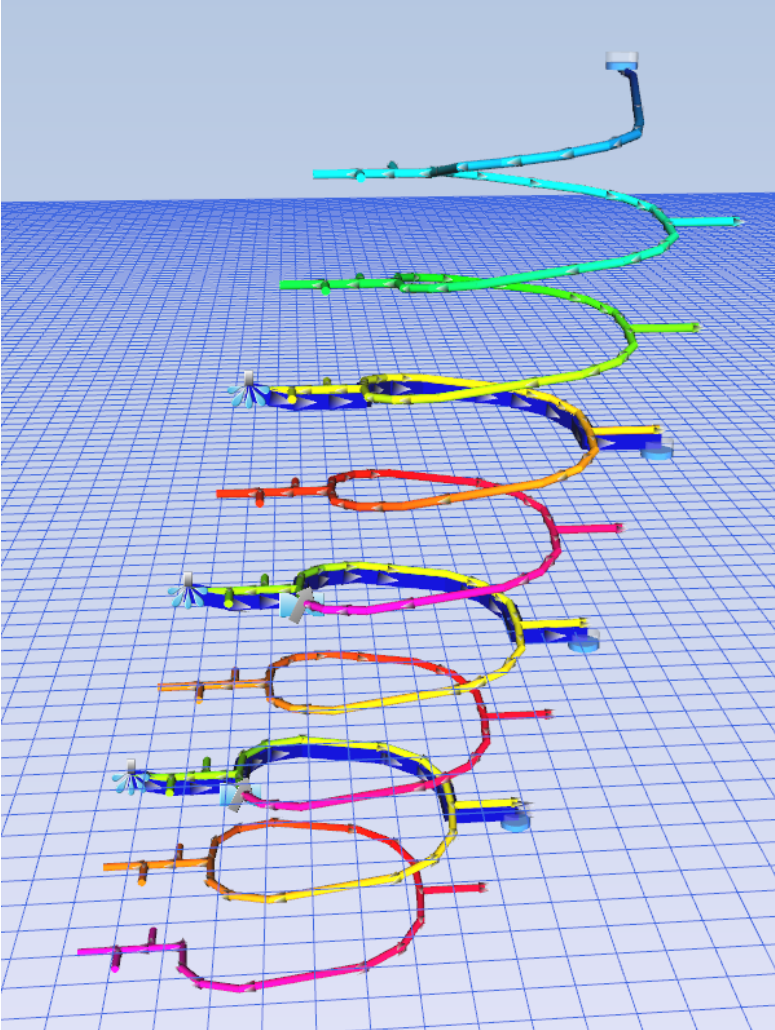
Channels

Channels are used to bring the water from the jumbos to dams where the water can be pumped from. The channels will follow a similar path to the pipes along the decline.

1. Select the pipes along which the channel will be running using the select tool.
2. Click on duplicate then click on one of the pipes and type -3 m in offset Z.
3. While the new pipes are selected, double click on edit.
4. Select channel, rectangular and enter 500 mm for both height and width.
5. The channel needs to be connected to the jumbo, to do that click on the move function then select the channel end and move it to connect to the jumbo.
6. Place a tank at the end of the channel. Make sure the channel connects to the top of the tank by making the "height of the pipe connection" equal to the height of the tank. Simulate.



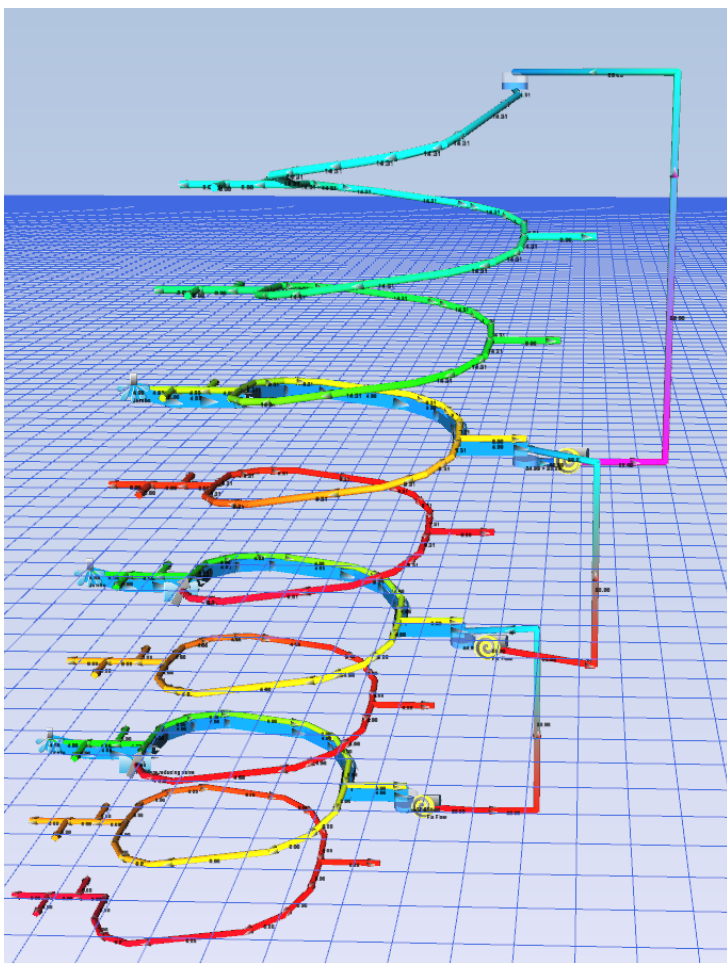
7. Repeat for the other jumbo.



Pumping

The water will now be pumped from tank to tank to the surface.


1. Draw some pipes from the lower tank heading away from the decline.
2. Click on the last pipe and enter 65 m in offset Z.
3. Join the pipe to the tank at the same level.



Dynamic simulation

If we run the dynamic simulation now, the tanks will be empty and the pumps will run dry. To avoid this, a script can be setup for the pump to stop when the water level drops too low and to start up again when the water level is high.



1. Open the scenario editor
2. Enter a scenario name, e.g. "Pump management".
3. Then click the  button in the Events table.
4. Select Below for the Event.
5. Click on the red arrow then select the lower level tank.
6. Enter a water level (1 m) or percentage (20 %).
7. Select Stop as the Action.

8. Click on the red arrow then select the lower level pump.
9. Press OK and then add another event using the same tank and pump combination, but this time set the pump to start when the water level is above 80 %.
10. Redo for the upper level pump station.
11. Edit the surface tank and check the "Hold Level" option for now. This will keep the water level in this tank constant during the dynamic simulation.
12. Run the simulation.
13. To increase the length of the simulation, use the settings menu found below the Dynamic Simulation button.

5. Configuration

5.1 Presets

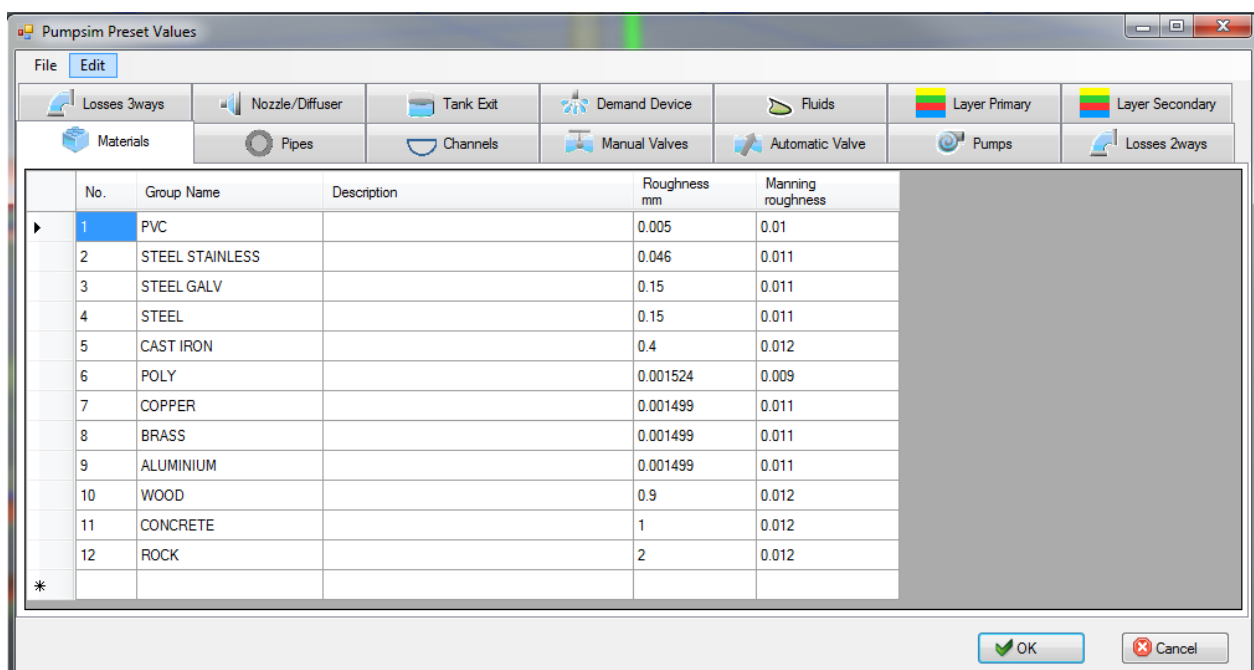


Fig. 5.1.1: Preset values box of pipes in Pumpsim™.

Preset Values

Preset values provide a quick and convenient way to specify pipe characteristics and parameters that may be commonly used in the model. Examples are pipe type, roughness, minor losses or valves.

The preset table also provides access to model primary and secondary layer names and colours, as well as water types, pumps and numerous other items used in Pumpsim™ simulation.

Any preset value which is changed will be applied to **ALL** pipes using that preset value. For example if a model has 10 pipes using a resistance preset called 'Rubber Flaps' and the resistance for 'Rubber Flaps' is changed in the preset box, then ALL 10 pipes using rubber flaps will have the new resistance applied when simulation is next performed.

Most items can be deleted by selecting the entire row (or a selection of rows) and pressing **DELETE**, or by selecting and deleting individual values. If a preset is currently in use, a warning may appear stating the fact, and the result it will have on the model. Ideally, presets should not be removed if they are currently in use.


Sort Order

The sort order of the preset values displayed in the spreadsheet, and displayed in forms in other parts of the program can be specified. All columns in the preset form can be sorted by Ascending or Descending by clicking the column header once or twice. For example, resistances can be sorted by name (by clicking the 'Resistance Name' column header, or by value (by clicking the value header).

In addition, row items can be manually sorted, by selecting the row and pressing the **UP** or **DOWN** arrow on the keyboard. The sort order will be retained next time the form is accessed and will be saved with the file.

Accessing Presets

Preset items are accessed from the main **SETTINGS** menu, however most preset items can also be directly

accessed from the **edit forms** by pressing . When activated from an **edit form**, the currently used preset will be highlighted in the **Preset** form.

Preset Types

Materials

Different material can be set up. Entering the absolute **Hazen Williams roughness** (for pipe) and the **manning roughness** (for channel).

Pipes

Pipe presets allow a specific size of pipe to be set with pre-defined dimensions, material and nominal pressure. See [Pipes and Channels](#) for more information.

Channels

Channel presets allow a specific size and shape of pipe to be set with pre-defined dimensions, material and nominal pressure. See [Pipes and Channels](#) for more information.

Valves

Valves have to be predefined with a type, a flow factor (Kv) (or flow coefficient Cv for imperial) value and the size. The type will change the equation that link the opening percentage and the flow factor at a opening. See [Valves](#) for more information.

Automatic valves

This category is for valves that will adjust during the simulation, such as Pressure Reducing Valve (PRV), Fix Flow, Ratio Reducing or No Return valves. Info and size can be added as well. See [Valves](#) for more

information.

Pumps

Present a summary of current pumps, and the basic characteristics of those pumps (eg diameter, water density for the curve etc). While the pump curve data cannot be directly edited from this screen (you will need to go to the pump database function), the pump names can be changed as well as the other pump parameters. Pumps can also be fully deleted from the model. A read only column states the number of pumps in use in the model. See [Pumps](#) for more information.

Losses 2ways

Fitting connected to 2 pipes only, such as elbow or bend. A different resistance on the other direction can be set. If the reverse resistance is 0, it will consider both resistance equal. See [Junctions](#) for more information.

Losses 3ways

Fitting connected to more than 2 pipes, such as tee or wye. See [Junctions](#) for more information.

Layer Primary, Layer Secondary, Fluids

The form shows current names for these items, and in the case of layers, whether they are being used or not. The names can be changed at any time, and new layer names and water type names can be added. Colour can be changed by clicking on the colour box. There are currently 250 layers and 25 pipe types reserved for use.

Tank Exit

Specific resistance at the exit or entry of a tank (set on the **Pipe Edit box**).

Spray

The discharge coefficients used for the simulation. It can be calculated from an array flow/pressure by clicking on the **calculator** column. See [Sprays](#) for more information.

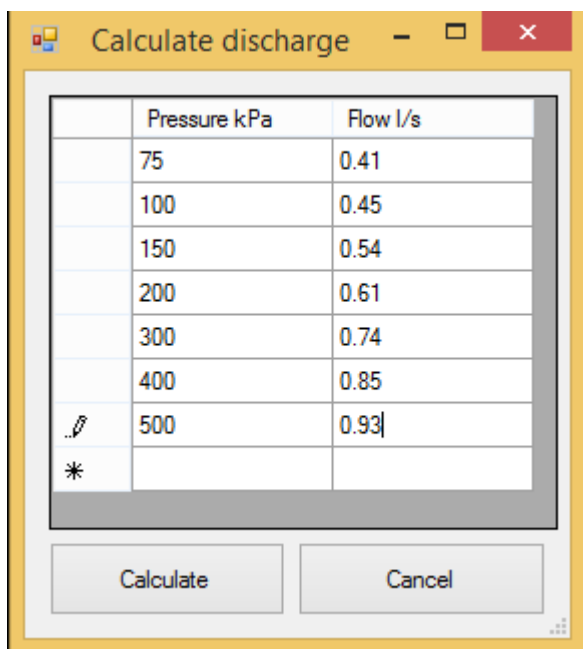


Fig. 5.1.2: The discharge coefficients calculated from array flow/pressure.

Nozzle / Diffuser

The resistance will be calculated after entering the angle and both diameters. See [Junctions](#) for more information.

5.2 Settings

Settings provide control over a large number of parameters used in Pumpsim™ for simulation, graphical display and file handling. **Settings** are normally saved specifically within the Pumpsim™ model file, however they can be shared with other files using the **INHERIT** or **MASTER LINK** functions.

Costing

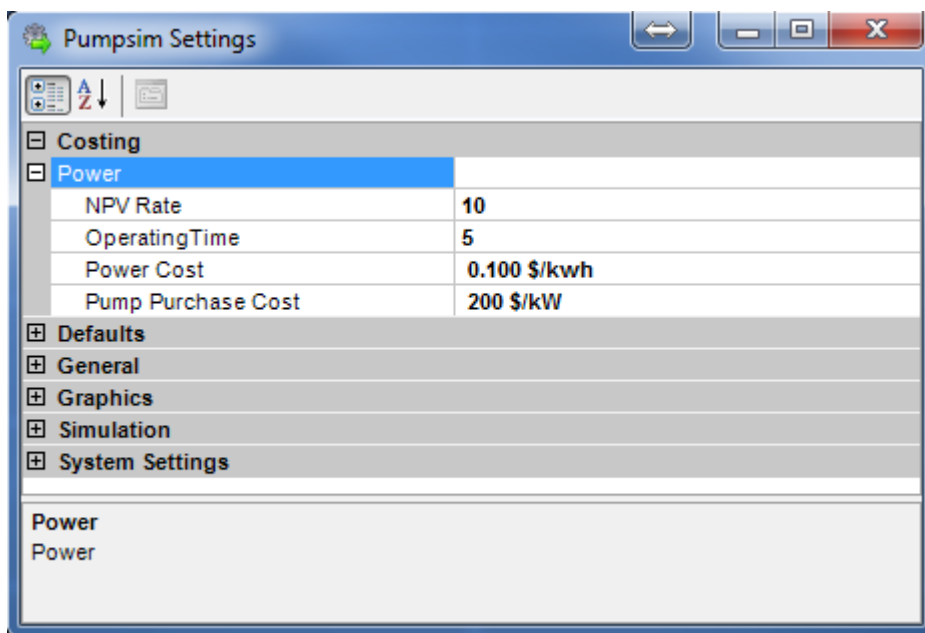


Fig. 5.2.1: The costing menu within the Pumpsim™ settings window.

Power

- **NPV Rate** – Discounted cost rate (in whole % per annum) to apply to future costs to allow calculation of present value.
- **Operating Time** – System running duration.
- **Power Cost** – Cost of electrical power.
- **Pump Purchase Cost** – Estimate cost of pump purchase depending on pump power.

Defaults

Set default settings and waterway sizes when first building a model, as well as file saving and loading behaviour in Pumpsim™.

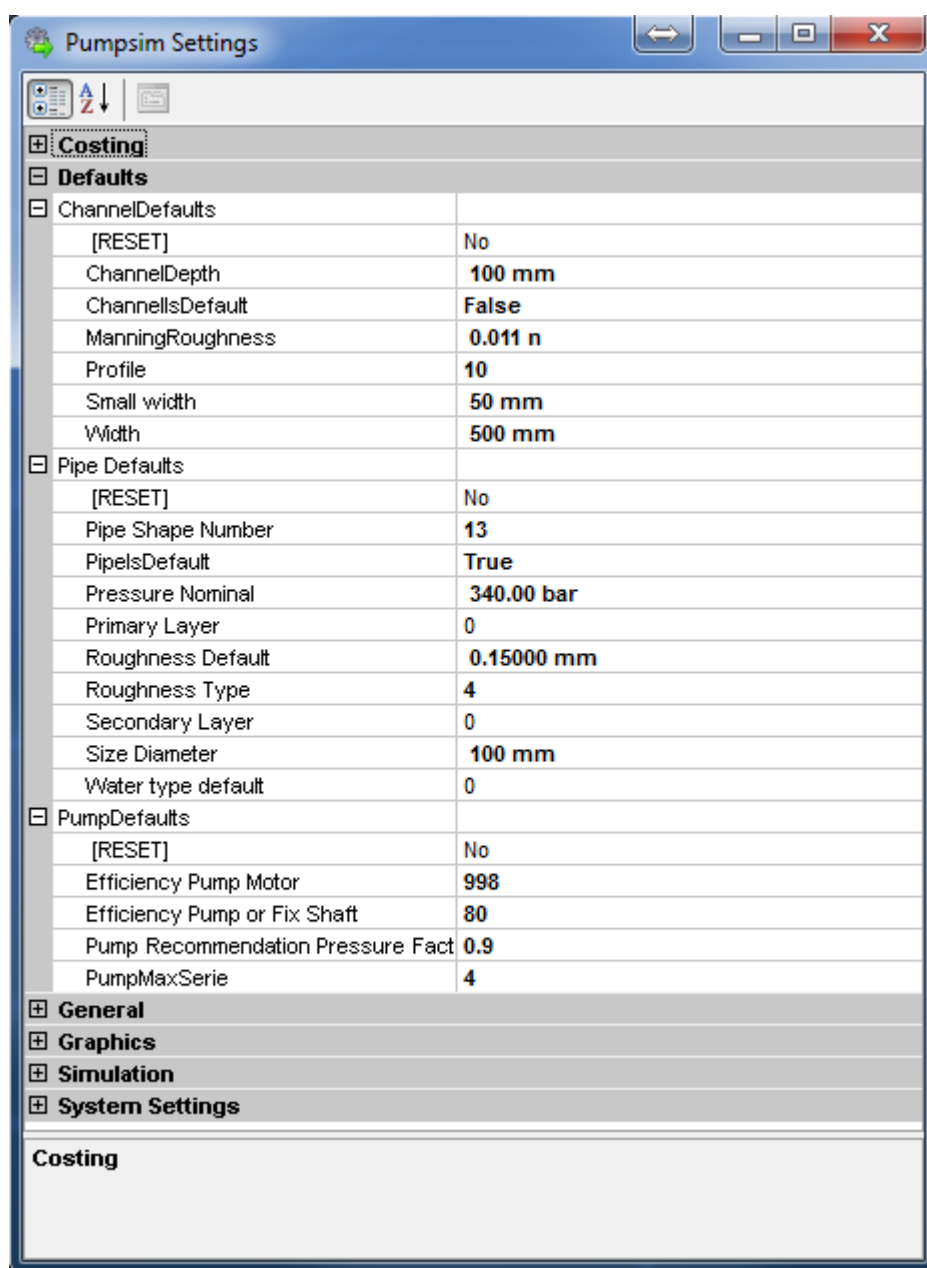


Fig. 5.2.2: The default settings menu within the Pumpsim™ settings window.

Channel Defaults

- **Channel Depth** – Depth of channel.
- **ChannelsDefault** – If true, the default drawing object will be a channel. Will change **"Pipe Is Default"** to **FALSE** under **"Pipe Defaults"**.
- **Manning Roughness** – Manning roughness of the channel material.
- **Profile** – Profile of the channel (rectangular, triangular, trapezoid or semi-circular).
- **Small Width** – Bottom width of a trapezoidal channel.

- **Width** – Top width of a channel.

Pipe Defaults

- **Pipe Shape Number** – Default shape of the pipe.
- **Pipe Is Default** – If true, the default drawn object will be a pipe. Will change "**ChannelsDefault**" to **FALSE** under "**Channel Defaults**".
- **Pressure Nominal** – Default maximum pressure that can be supported by that pipe.
- **Primary Layer, Secondary Layer** – Default layers to apply to new waterways.
- **Roughness Default** – Roughness
- **Roughness Type** – Roughness
- **Size Diameter** – Default size of new pipe. Imported DXF and text files without specific waterway size will also be set to these values
- **Water Type Default** – Default fluid type.

*Hint: Pipe defaults are normally only applied if pipes are constructed without connection to other pipes. Where new pipes are connected to existing pipes, they will INHERIT the settings of the waterways the new pipes are connected to. This behaviour can be modified from the **EDIT > NEW PIPES** menu item.*

Pump Defaults

- **Efficiency Pump Motor** – Default pump motor efficiency to apply to calculated power to estimate the electrical power absorbed by the pump motor. In most cases it will be around 95% for direct drive electrical motors, or as low as 80 – 85% for offset or gear drive pumps.
- **Efficiency Pump or Fix Shaft** – Default efficiency factor to apply to motor shaft power calculations for fixed quantities and pumps without efficiency or power curves.
- **Pump Recommendation Pressure Factor** – Pressure difference allowance in pressure for the recommendation.
- **PumpMaxSeries** – Maximum number of pumps in parallel or series.

General

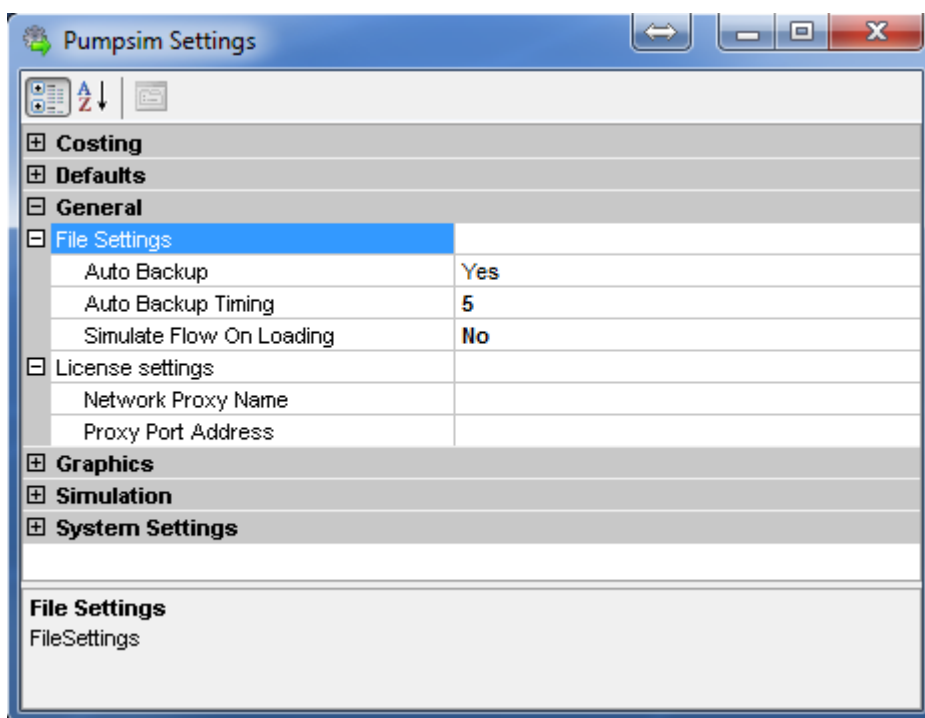


Fig. 5.2.3: The general settings menu within the Pumpsim™ settings window.

File Settings

- **Auto Backup** – Forces Pumpsim™ to make a backup of the current model file at regular intervals. If the program crashes, or is forcibly exited, the backed up model will be loaded automatically the next time Pumpsim™ is run.

Warning: If the model has become corrupted for some reason, there is a possibility that the auto-saved model may also be corrupted. For this reason, it is recommended that a model be regularly saved to ensure that good working copies are available as backups if required.

- **Auto Backup Timing** – Set interval of backup timing in minutes ('5' = backup every 5 minutes)
- **Simulate Flow On Loading** – Automatically performs a simulation when the file is loaded. This will update simulation summary parameters and a number of calculated heat parameters; these parameters are only available after a simulation.

License Settings

Alters license activation settings which may be required to allow Pumpsim™ to access the internet to validate licenses. Pumpsim™ normally uses Microsoft Internet Explorer settings to access the internet, however on occasion third party programs like Pumpsim™ may be denied direct access, requiring the **proxy name** and **proxy port address** to be entered manually.

These values can also be adjusted in the **License Manager** form.

Graphics

Control the various aspects of the screen graphics and presentation.

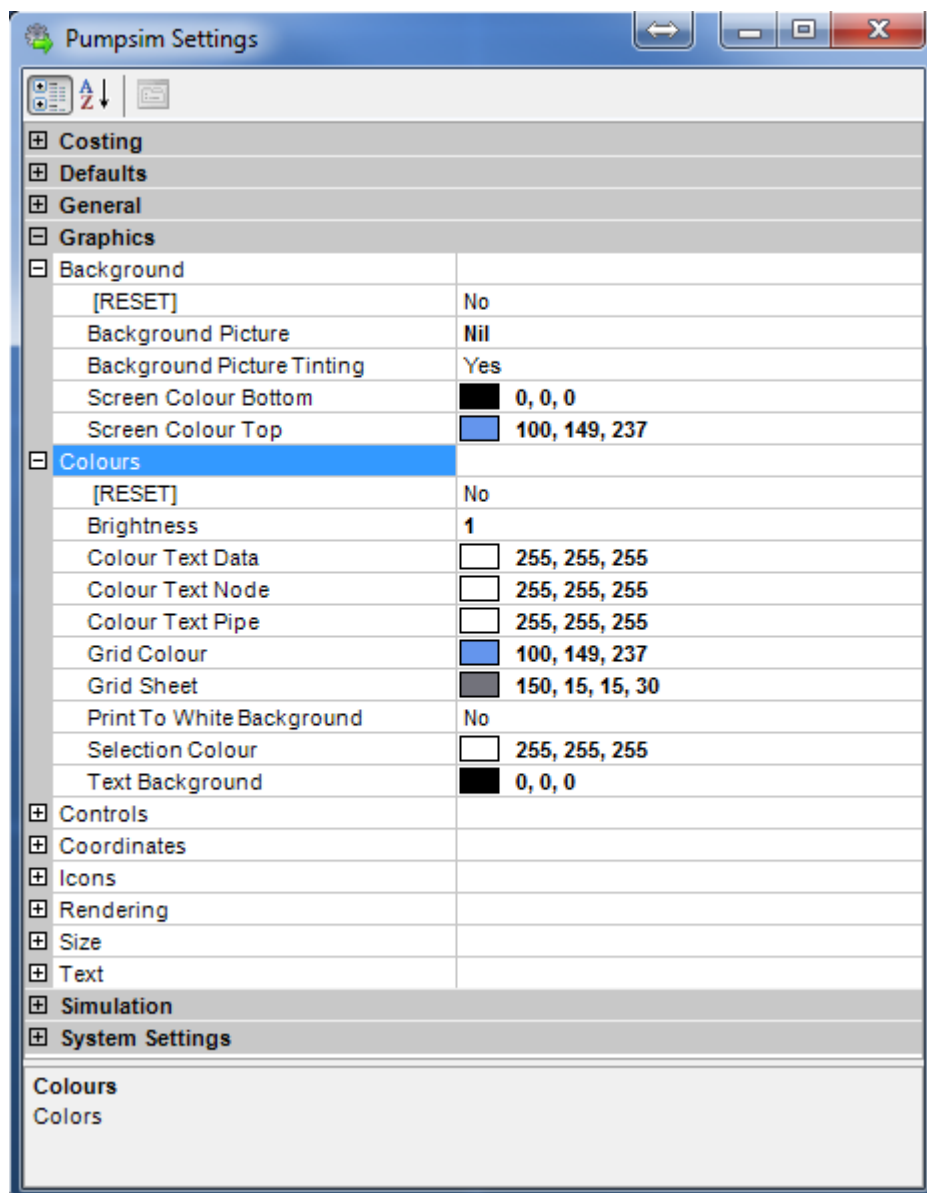


Fig. 5.2.4: The background and colours menu within the Pumpsim™ settings window.

Background

- **Background Picture** – Sets a preset or custom picture to the background of the visual display. This may provide some visual flair for presentations or may simply suit the users preferences. An option exists to use a preset background (current **CLOUDS** or **GROUND**), or a custom background picture can be displayed by simply dragging a picture file from windows on to the screen.
- **Background Picture Tinting** – Tints the background picture using the Screen Colour Top and Bottom

settings. Note that a black screen colour will totally hide the pictures, while a white screen colour will fully show the picture in natural colours.

- **Screen Colour Bottom and Top** – Colour of upper and lower halves of background. The colours are smoothly blended to provide a gradient effect. Lighter colours may be more appropriate for presentations and reports.

Colours

- **Brightness** – Controls the relative brightness and intensity of colour display on the screen.
- **Grid Colour** – Colour of grid lines.
- **Grid Sheet** – Colour of semi-transparent edit plane sheet shown during vertical drawing operations.
- **Colour Text Data** – Colour of the waterway data. Text colours may need to be adjusted to provide contrast if the background colours are adjusted.
- **Colour Text Waterway** – Colour of the waterway text names and error messages. Text colours may need to be adjusted to provide contrast if the background colours are adjusted.
- **Colour Text Node** – Colour of the waterway node names. Text colours may need to be adjusted to provide contrast if the background colours are adjusted.
- **Print to White Background** – Instructs the program to make the background white when printing a graphics model to a printer or saving the image to a file.
- **Selection Colour** – The colour of selected waterways. The default colour is yellow, however sometimes this colour may not be as visible with lighter colour backgrounds.
- **Text Background** – The colour behind text on the screen if transparent text is not used.

Controls

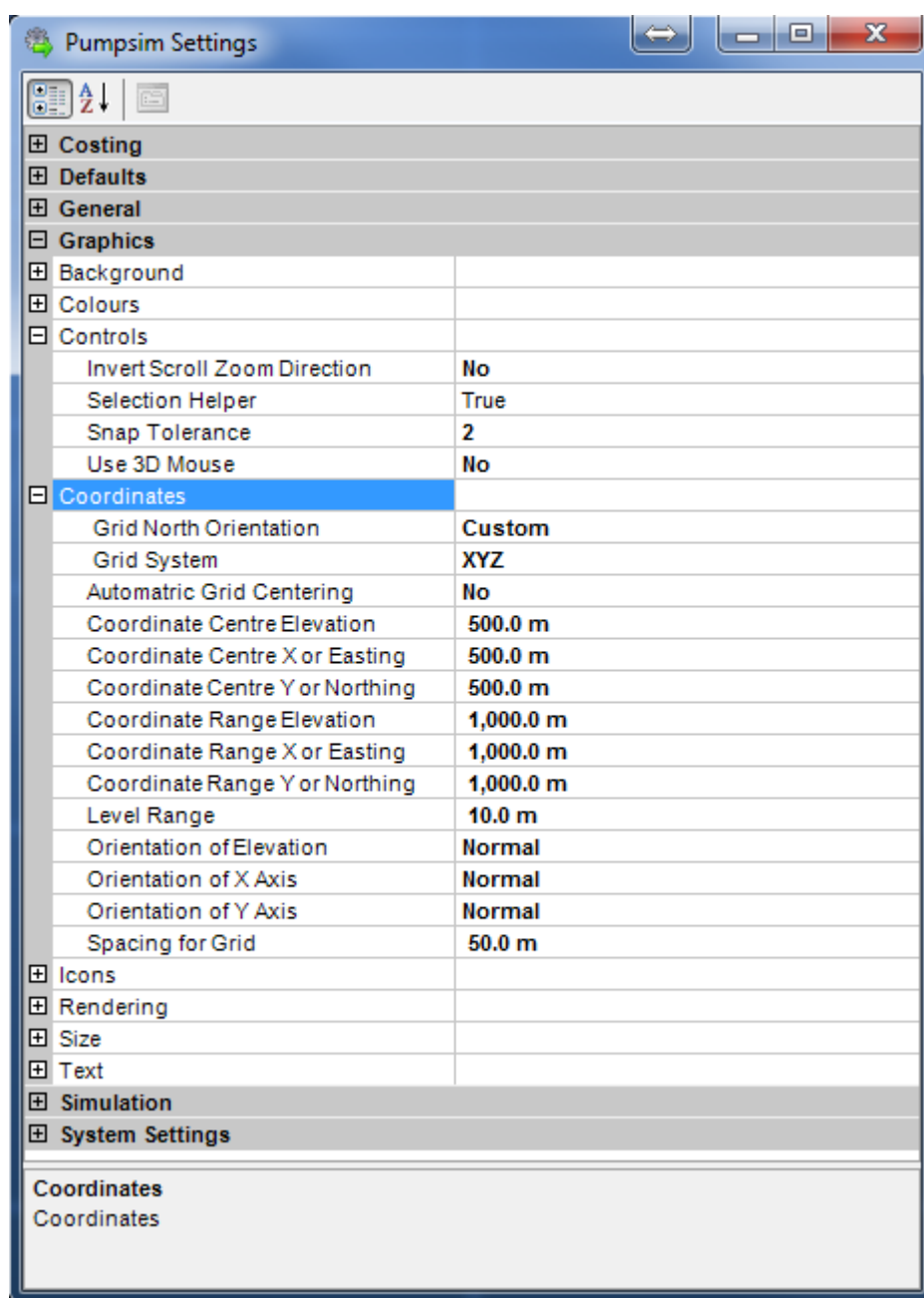


Fig. 5.2.5: The Coordinates menu within the Pumpsim™ settings window.

- **Invert Scroll Zoom Direction** – Invert the direction of the scroll zoom using the mouse.
- **Selection Helper** – Highlight the object under the mouse. Should be disabled in large models as it can slow down the graphics.
- **Snap Tolerance** – Adjust the willingness of the Pumpsim™ cursor to adjust or join to other waterways while drawing or moving waterway items. For finer control (less propensity to connect to close waterways) reduce this number (minimum 1, maximum 100)
- **Use 3D Mouse** – Enable use of a Connexion™ 3D mouse. This type of input allows for models to be rotated, panned, and zoomed with a separate controller, leaving the regular mouse free to operate

menus and selection duties. This option will have no effect if a Connexion™ 3D mouse is not present.

Coordinates

- **Grid North Orientation** – Defines the direction of grid north on the screen. Therefore, if grid north direction is defined as **UP**, then the **Northing** coordinate numbers will start lower from the screen bottom, to higher numbers towards the screen top. If a **GRID NORTH** system is not chosen, then this setting will show **CUSTOM**.
- **Grid System** – Most mines/models use a **GRID NORTH** system. Pumpsim's coordinate convention when using grid north coordinates is **Easting, Northing** and **Elevation** in that order. If another type of system is used, then an **XYZ** system can be specified, which allows for orientation of coordinates in any direction.
- **Automatic Grid Centering** – Automatically calculate model bounds and grid location.
- **Coordinate Centre** – Defines the centre of the grid system from which the grid lines will be drawn. Note that this can only be specified if **Automatic Grid Centering** is turned **OFF**, otherwise the centre numbers will be automatically adjusted when the model is viewed or reloaded.
- **Level Range** – The range of elevations to show around a single selected level. Selected levels may be chosen from the **RIGHT CLICK** mouse popup menu, and limit the range of waterways data shown on screen.
- **Orientation of Elevation, X Axis, Y Axis** – This option should only be used for **CUSTOM** grid orientation. **GRID NORTH** orientations do not require these settings to be adjusted.
- **Spacing for Grid** – The spacing between grid lines on the screen.

Icons

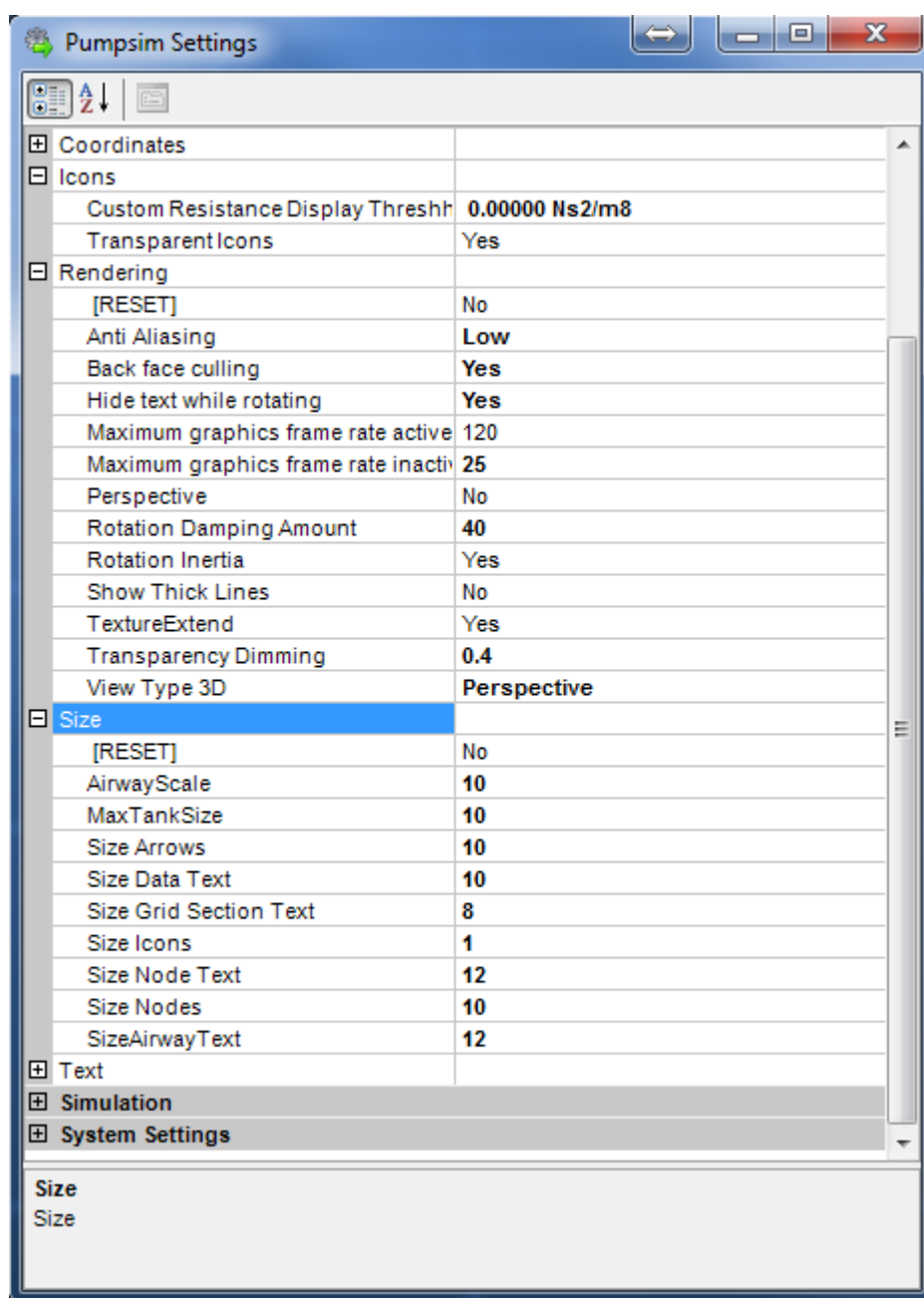


Fig. 5.2.6: The Rendering menu within the Pumpsim™ settings window.

- **Custom Resistance Display Threshold** – Controls the display of resistance icons where waterways have been assigned a custom resistance. Only waterways with a resistance above the threshold will have an icon displayed.
- **Transparent Icons** – Shows transparent sides for icons over waterways.

Rendering

- **Anti-aliasing** – An advanced graphics option (not supported by all graphics card) which smooths the appearance of the edges of solids to give a more natural look. The option may significantly slow some

graphics cards or cause graphics irregularities. By default, it is set to **FALSE**.



Fig. 5.2.7: On the left is an image of a modelled pipe before anti-aliasing is applied. On the right is the same pipe with anti-aliasing on. The right pipe is significantly smoother.

- **Back face Culling** – Removes hidden surfaces from the display. This may make older graphics cards more responsive in displaying complex graphics.
- **Hide Text While Rotating** – Hides text while model is rotated or zoomed. Very large models with lots of text can slow or make smooth rotation movements unresponsive. Use this option to hide text and make rotation movement much smoother.
- **Maximum / Minimum Frame Rate** – The rate at which to update the screen graphics. To conserve laptop battery power, the minimum frame rate option is available to reduce power when Pumpsim™ is not the active program in Windows.
- **Perspective** – Apply perspective to 3D view.
- **Rotation Inertia / Damping** – Pumpsim™ model rotation results in a short period of continued rotation after the mouse is released. This is purely for visual appeal and is provided to give models the illusion of 'weight' and solidity. The speed at which rotation is damped can be adjusted or turned off using these settings.

Hint: To allow the model to rotate freely without stopping, choose a damping level of zero (0), or HOLD THE CTRL key down when rotating a model with the mouse.

- **Show Thick Lines** – Hide or shows arrow, node and text data by default. This can be used to improve display clarity by removing unnecessary graphical details. Note that this function can be overridden from the **View Menu** options or the **Right Click Context Menu** options for text and other graphical items.
- **Texture Extend** – If texture extend is set then it will duplicate the picture to fill where the picture does not extend. If it is not set, black will appear around the initial picture.
- **Transparency Dimming** – Level of dimming (0-1) from original colour of inactive pipes if transparency flag is set.
- **View Type 3D** – View type, perspective or orthogonal.

Size

- **Waterway Scale** – Scales waterways graphically so they appear larger or smaller than the specified dimensions. This does not change the calculated waterway size. A factor of '1' is normal scale. Factors larger than one will enlarge waterway graphics while factors smaller than '1' will shrink waterway graphics. This function may make very large extensive models larger and easier to see, or conversely can make very small model (eg lab scale apparatus) also easier to see.

- **Maximum Tank Size** – Maximum size which will be use to show tank.
- **Size Data / Node / Waterway Text** – Size of waterway text displayed on model. Larger text sizes are generally clearer to read, however excessive data may clutter display.
- **Size Node / Icons / Arrows** – The size of the nodes, icons and arrows. Note that the size reduces in wireframe mode to improve display legibility. In Solid mode, Icons and Nodes will not be reduced smaller than the waterway size.

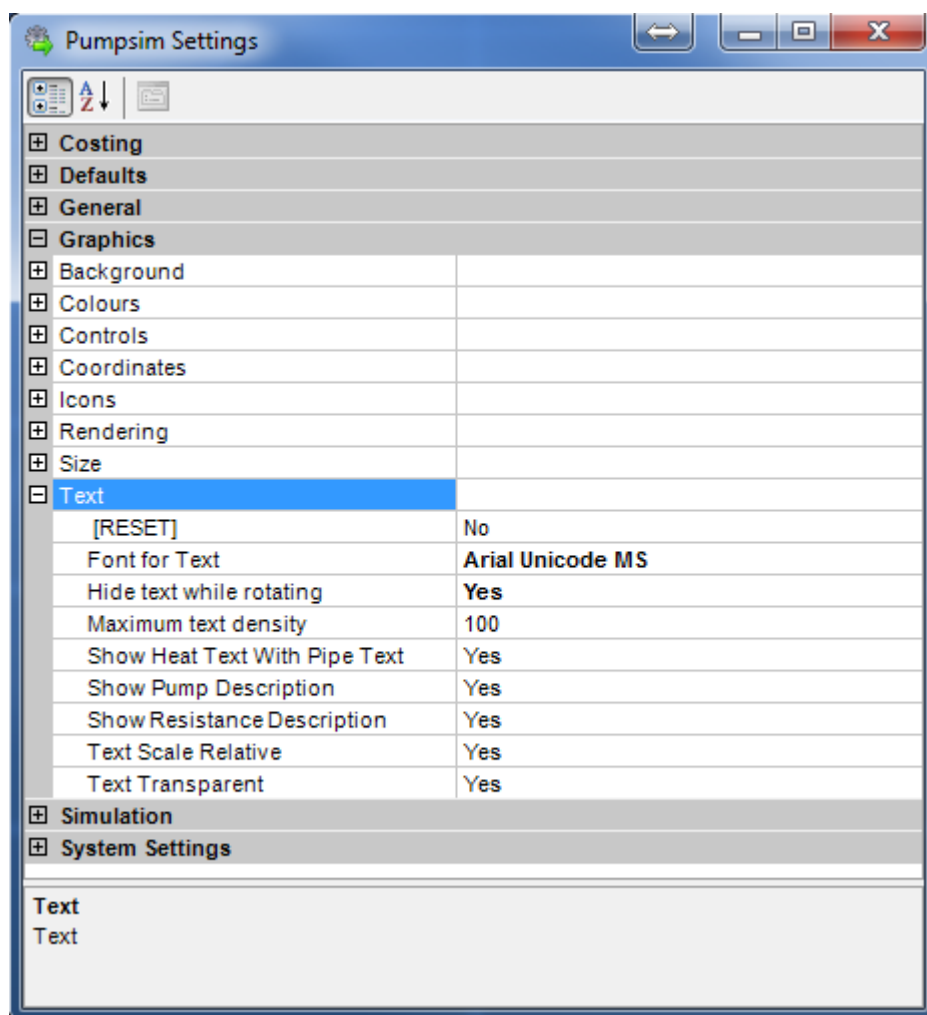


Fig. 5.2.8: The text menu within the Pumpsim™ settings window.

Text

- **Font for Text** – Allow selection of the type of FONT to display on the screen.
- **Hide Text While Rotating** – Display text on icons with heat sources.
- **Maximum Text Density** – Change how much text can appear on the screen
- **Show Heat Text with Pipe Text** – Shows the thermodynamic setting names on the waterways they are attached to.

- **Show Pump Description** – Display text showing pump name description
- **Show Resistance Description** – Show the description of the resistance
- **Text Scale Relative** – Reduces the size of text labels relative to distance away from view camera. This make close waterways larger and more legible and distant waterway text smaller.
- **Text Transparent** – The background for text characters is shown as a solid colour, or transparent. Turning off text transparency can improve clarity of text graphics, but will obscure the waterway graphics behind the text.
 - **[TRUE]** – Text labels backgrounds are transparent and show graphics under the label.
 - **[FALSE]** – Text label backgrounds are solid and do not show graphics underneath. In some cases, this may improve the legibility of text.

Simulation

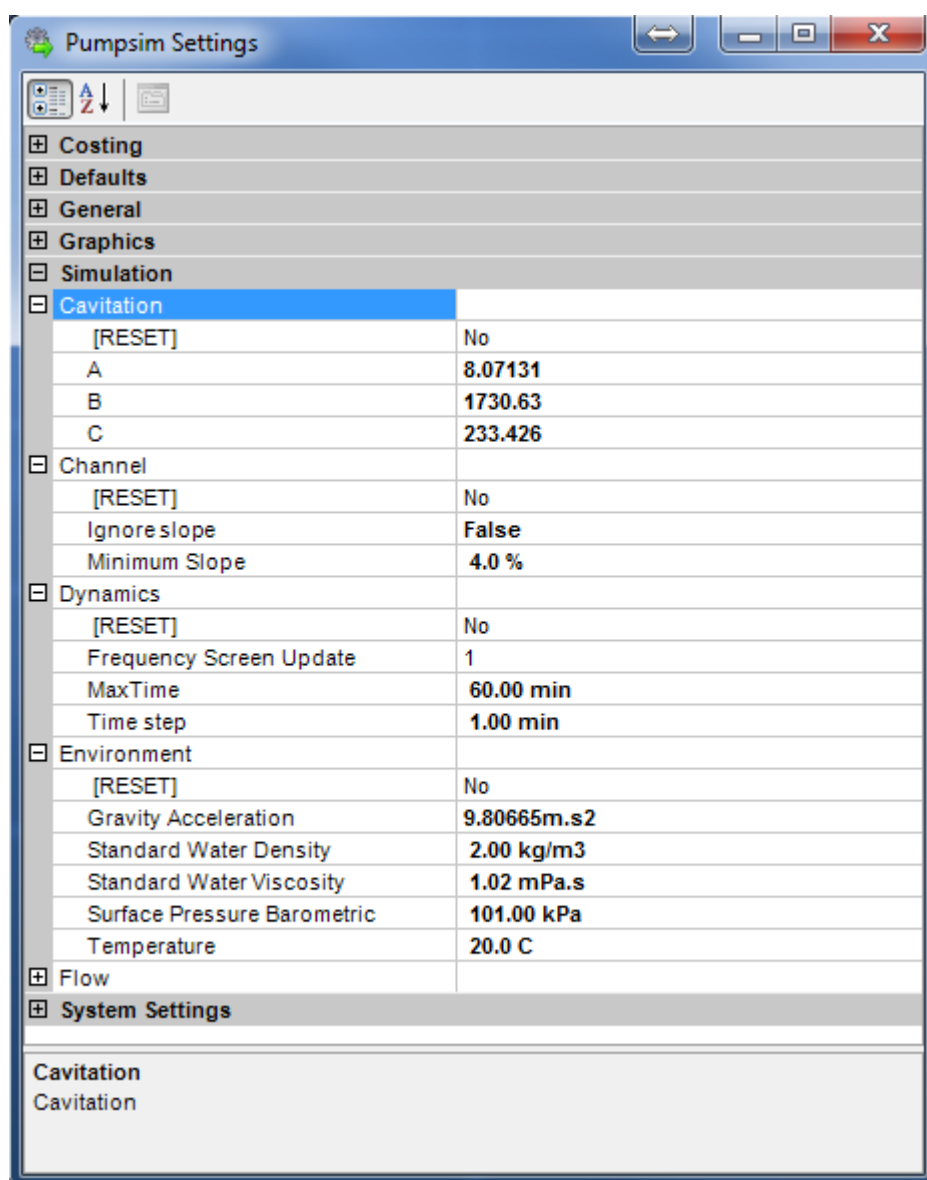


Fig. 5.2.9: The Simulation menu within the Pumpsim™ settings window.

Cavitation

Parameters for cavitation simulation

- **A, B, C** – Fluid constant of Antoine equation to calculate the vapor pressure.
- **Simulate cavitation** – Enable or disable showing warning for cavitation.

Channel

Parameter for channel simulation

- **Ignore Slope** – Do not use the slope of the channel but only the direction, the direction of each

channel must be correct for simulation. The depth will be calculating using the minimum slope.

- **Minimum slope** – Channel with a slope lower will be assume flat.

Dynamics

- **Frequency Screen Update** – Set the number of iterations between screens update.
- **Max Time** – Maximum time for simulation.
- **Time Step** – Increment of time for which the simulation is run.

Environment

- **Gravity Acceleration** – The standard gravity acceleration.
- **Standard Water Density** – Pressure outside the system, ie: at the water level of a tank or at the exit of a pipe.
- **Standard Water Viscosity** – Pressure outside the system, ie: at the water level of a tank or at the exit of a pipe.
- **Surface Pressure Barometric** – Pressure outside the system, ie: at the water level of a tank or at the exit of a pipe.
- **Temperature** – Pressure outside the system, ie: at the water level of a tank or at the exit of a pipe.

Flow

Simulation waterflow settings directly influence how the waterflow simulation operates.

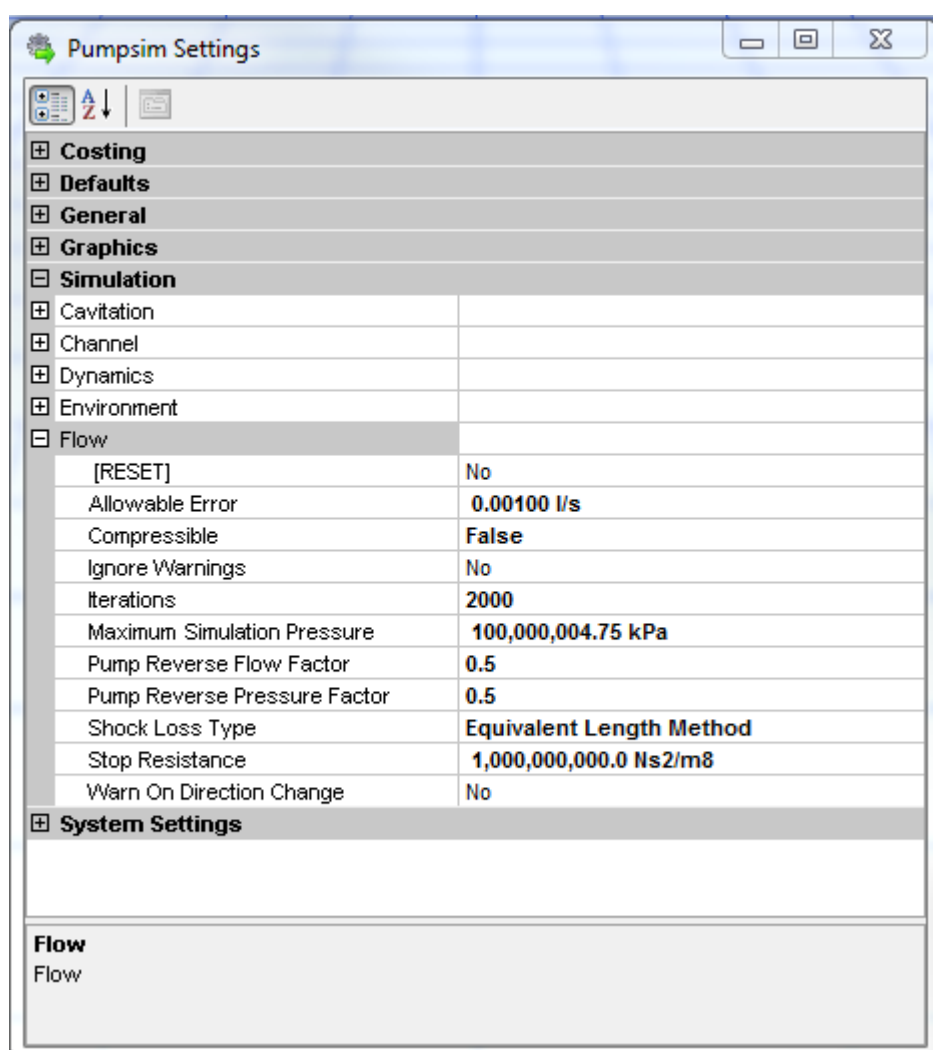


Fig. 5.2.10: The Flow menu within the Pumpsim™ settings window.

- **Allowable Error** – Defines the level of accuracy Pumpsim™ must resolve down to before an acceptable solution is displayed. This should normally be set to less than 0.1 m³/s error. If a final analysis is required it may be advisable to set this to 0.01 m³/s or lower. The smaller the value, the more accurate the simulation process, but the longer it may take.
- **Ignore Warnings**
 - **[TRUE]** – Pumpsim™ will ignore all warnings related to **No Entry** or **No Exit** errors found during simulation (waterways with no other waterways joining)
 - **[FALSE]** – Pumpsim™ will only ignore waterways which have been set to **Allow Closed End** in the **Edit Box**. Any other 'orphaned' waterways will cause the simulation to show warnings.
- **Iterations** – Sets the number of attempts Pumpsim™ can take to achieve an acceptable error, before the program abandons the simulation process.
- **Maximum Simulation Pressure** – The maximum pressure the simulation will allow between waterways before a simulation error is reported. This error may indicate unreasonable fixed flows or resistances which interact and cause large pressure changes in the model.
- **Pump Reverse Pressure Factor, Pump Reverse Flow Factor** – Defines the default reverse pump

performance relative to the original pump curve for the pressure and quantity of a pump running in reverse. These factors decrease the performance of pumps set to run in reverse (for emergencies for example) in the **Edit Box**. Note that the default values may be overridden by values directly placed in the **Pump Database** for individual pumps.

- **Shock Loss Type** – Defines the shock loss method to use. Pumpsim™ can be set to calculate shock losses using the equivalent length method, or the **shock factor (K)** method. Shock loss calculations are necessary to estimate pressure loss due to water turbulence caused by a change in waterway direction, a junction or a change in pipe size. Note that changing this value in an existing model will result in Pumpsim™ requesting to recalculate the shock losses using the alternative system.

The equivalent length method requires the user to estimate an equivalent extra pipe length required to approximate pressure loss due to shock. The **shock factor (K)** method uses a calculated factor derived from both empirical and calculated changes in waterway areas and velocities. Both methods are described and utilised in an array of relevant texts.

Once the method is set, the **Edit Box** will require an appropriate shock loss value for each pipe. The **Edit Box** can accept a manually entered number, but also has a number of preset values, as well as an **AUTO** function which will force Pumpsim™ to attempt to calculate a shock loss factor or an equivalent length.

- **Stop Resistance** – Defines the minimum resistance at which Pumpsim™ will completely stop all flow in a pipe. All preset resistances above this value will cause flow in the set waterway to completely stop. Only one **Preset Resistance** value should be greater than the **Stop Resistance** value. This function artificially restricts flow and has the potential to cause simulation problems if used on too many waterways. The simulation will check and ensure that only one (or nil) preset resistances are greater than this value.
- **Warn On Direction Change** – If set to **TRUE**, and a waterway is specified in the **EDIT** form with a **FIXED DIRECTION**, Pumpsim™ compares waterway directions before and after a simulation and alerts the user which waterways have waterflow that has changed direction during simulation.

System Settings

Pumpsim™ program settings control over-arching settings which influence all parts of the program.

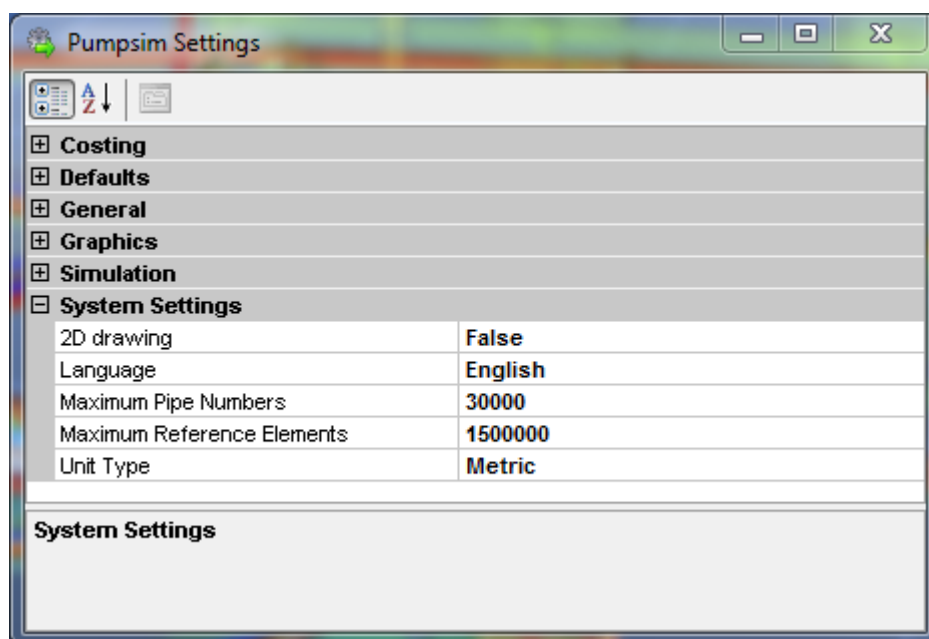


Fig. 5.2.11: The System Settings menu in Pumpsim™ settings window, that controls overarching features of the program.

- **2D Drawing** – Allow the user to draw a model in 2D. The elevation is still use for the simulation.
- **Language** – Set the language for all menus, dialog boxes and other text within Pumpsim™.
- **Maximum Pipe Numbers** – Sets maximum pipe numbers allowed to be constructed in a model.
- **Maximum Reference Elements** – Maximum number of reference element.
- **Unit Type** – The type of units to use in Pumpsim™. The program operates natively in **SI METRIC**, with underlying calculations performed in metric units. To display imperial values and accept imperial input, set the unit type to **IMPERIAL**. The imperial setting uses a conversion table to calculate the conversion from metric, and can be customised to suit site preferences. The imperial conversion table can even be set to use a combination of **METRIC** values by setting the program to **IMPERIAL** and then using the **SETTINGS > UNITS** menu to individually specify what units will remain **ALWAYS METRIC**.

Warning: The conversion table has set limits of decimal accuracy to convert from metric to imperial and back again. If the decimal accuracy is too low, some accuracy may be lost in the conversion process, and the value input as an Imperial number, may be returned slightly differently.

6. Appendix

[Appendix A - Glossary of Terms](#)

[Appendix B - Summary of Data Types](#)

[Appendix C - Icon Picture Guide](#)

[Appendix D - Display Problems](#)

[Appendix E - Simulation Errors](#)

[Appendix F - Shortcut Keys](#)

6.1 Appendix A - Glossary of Terms

A glossary of commonly used terms.

Branch	A pipe or channel carrying flow
Channel	Conduct open at the top.
TXT	TAB Separated Value file - a text file format used by Pumpsim™ when saving uncompressed files. This format can also be read into Spreadsheets, Word Processors and databases.
DXF	Drawing Exchange Format - An AutoCAD graphics file format that can be imported and exported by most CAD and Mining Software packages. Pumpsim™ can also import or export data in this format.
Endpoint / Start point	The start or end of a pipe, defined by an x , y and z coordinate, which gives its position in space in three dimensions.
Flow behaviour index	Variable of the power law.
Friction cost	Similar to friction power, with the kilowatt value converted to a dollar cost per annum, based on the default power cost.
Friction factor or k factor	Atkinson's friction factor, describing the roughness or unevenness of a wall. It directly affects the resistance of a pipe. Friction factors are measured to a specific density, normally normalised to 998kg/m ³

Friction loss	A component of pressure drop along a pipe caused by the pipe resistance.
Friction power	Derived from the friction loss, this estimates (in kilowatts), the amount of energy lost due to the resistance of a pipe on the flow.
Hardy-Cross Method	The simulation method used by Pumpsim™ to perform the calculation of flow in a model. It uses an iterative estimation method that adjusts the flows through a model until the estimation errors lie within acceptable limits. Pumpsim™ uses a modified method which takes into account density changes and mass flow balances.
Junction	A point defining where two or more pipe ends share the same position. Flow at this point can split or join, depending on flow direction.
Load (Pressure)	Estimation of how much load or weight a pipe pressure loss would develop across a resistance.
Network	An interconnected series of objects that together form the model of a flow design.
Node	A point defining the end or junction of a pipe(s).
Pan	The action of sliding the model graphics to a new screen position by using the right mouse button.
Pressure loss	The loss of pressure along a length of pipe due to friction loss, pumps, fixed pressures or fixed quantities etc.
Resistance	A value describing the difficulty fluid will have moving down a pipe. It is derived from a combination of pipe size, friction factor, length, density and viscosity.
Shock Loss	A factor that estimates the effect a change in drive direction or size has on flow. Any such change will increase turbulence in an flow, and hence develop energy losses that can be equated to an increase in pipe resistance. Pumpsim™ uses shock loss as an extra length added on to the original drive length (i.e. the higher the shock loss, the greater the equivalent length), which in turn increases the overall pipe resistance.
Thermal Conductivity	Thermal conductivity, k , is the property of a material that indicates its ability to conduct heat. It appears primarily in Fourier's Law for heat conduction.

Density	The density of a material is defined as its mass per unit volume. The density influences many psychometric properties.
Viscosity	The viscosity is the fluid resistance to sheer stress.
Yield stress	The minimum amount of stress required for Bingham plastic to start deforming.

6.2 Appendix B - Summary of Data Types

Summary of the major data types.

Available from the **Data Selection** menus in the menu bar. Note that additional data and data subsets are available in the spreadsheet view.

Flow

Quantity	The volume flow rate of liquid/slurry through a pipe.
Velocity	The average speed of flow through a pipe across the cross-sectional area.
Mass Flow	The mass flow through a pipe. This is a function of volume flow and density. Note that this will not necessarily be equal to the quantity x wet density value, because the mass flow does not consider the moisture component that is included in the density value.
Reynolds	The Reynolds describe the effect of viscosity on the fluid. If it is lower than 2100 it will be a laminar flow, if it is higher than 4000 it will be turbulent

Pressure

Pressure Total Loss	The total pressure loss along a pipe between entry and exit. Pressure loss may consist of pipe friction pressure losses, due to wall friction factors and additional resistances placed in the pipe.
Pressure Gauge	The pressure inside a pipe, corrected by the atmospheric pressure. The correction removes the effect of elevation on pressure, and permits a relative assessment between pipes at different elevations.

Pressure Friction Loss The loss of pressure along a pipe due to wall friction. This ignores any other resistances in the pipe such as doors or stoppings.

Note that by default, atmospheric density is adjusted for temperature gradient (lapse) rate, in much the same way as auto compression can drive changes in underground temperature at different elevations. Inlet pipes at different elevations will have different temperatures which are derived from the standard elevation and default surface temperature. This function can be turned off (surface temperature adjust) in the settings if desired.

Pipe Attributes

Roughness	Roughness of the pipe; this value also represents the material.
Dimensions	The size of a pipe width and height as the centre section.
Perimeter	The perimeter of a pipe, calculated from the centre height and width, taking into account the pipe shape profile. The perimeter can also be set directly from the EDIT menu.
Area	The area of a pipe, calculated from the centre height and width, taking into account the pipe shape profile. The area can also be set directly from the EDIT menu.
Length	The true length of a pipe, used to calculate pipe resistance and heat transfer. It does not include additional shock lengths if specified.
Coordinates	Report the coordinates of a pipe in eastings, northings and elevation.

Descriptors

Pipe Number	The internal array number of a pipe, referenced by Pumpsim™. Pumpsim™ will report this number in error descriptions. The pipe number may change as pipes are added or removed from the model.
Pipe Name	The name of a pipe branch. The name can be entered in the pipe EDIT box. The name does not affect simulation but may assist the user in identifying or finding pipes.
Unique ID Number	A unique number assigned to every pipe, that does not change when pipes are added or removed. As a result, the numbers may not be sequential and may have large gaps in the sequence.
Pipe Type	The descriptive name of the pipe type set in the EDIT box.

Profile	The descriptive name of the profile if set.
Friction Factor	The Darcy Weisbach friction factor.
Primary Name	The name and number of a pipe primary layer.
Secondary Name	The name and number of a pipe secondary layer.

6.3 Appendix C - Icon Picture Guide

Pipe Icons

A number of icons are used in Pumpsim™ to represent various pipe attributes. The icons are displayed on top of the **pipe solid** or **wireframe**, and indicate the presence of a specific item. Multiple attributes in a pipe represented by icons are shown side by side.

- Icons can be turned off or on from the **Display Manager** or from the right mouse button context menu.
- Icons can be moved along pipes by selecting the **MOVE** function from the toolbar and dragging the icons along the pipe with the left mouse button.
- Icons can be copied to other pipes by selecting the **COPY** button function from the toolbar and dragging the icon to a new pipe.

Customized Icons

Pump, **resistance** and **heat icons** can be customised in Pumpsim™. Providing the icon is established with a **PRESET** name, a picture file can be dragged on to the icon externally from the Windows File Manager. The picture file can be formatted as a JPG, PNG or GIF file. In addition, the GIF format also supports animated pictures which will be displayed as an animated icon in Pumpsim™.

Pumps

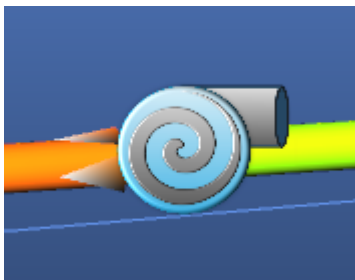


Fig. C.1: Pump icon.

Pump icons are used to **represent the presence of a pump**. Pumps can be shown in three different colours.

- **Blue** – normal operating pump
- **Green** – Fix Flow
- **Red** – Fix Pressure

Valves

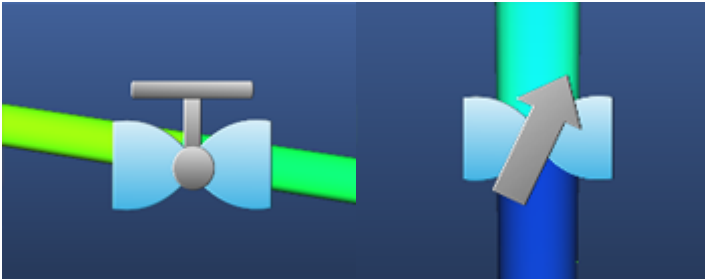


Fig. C.2: Valve icons

Valves icons are used to **represent the presence of a valve**. Valves can be shown in three different colours.

- **Blue** – normal operating pump
- **Green** – Fully open valve
- **Red** – Fully closed valve

Bend



Fig. C.3: Bend icon.

Indicate a pressure loss due to a bend.

Tee junction



Fig. C.4: Tee junction icon.

Indicates a Tee junction.

Diameter change

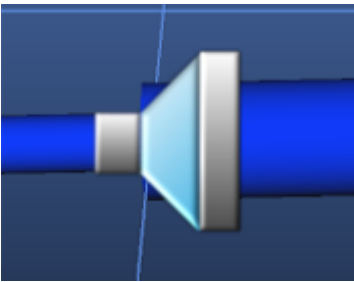


Fig. C.5: Diameter change icon.

Indicate a pressure loss due to a change in diameter.

Spray



Fig. C.6: Spray icon.

Indicates a spray.

Blocked Pipe

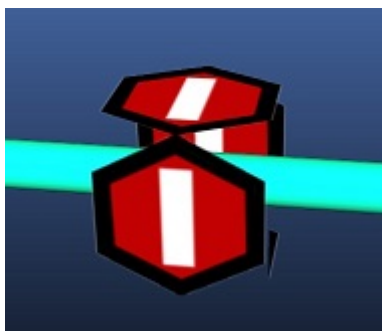


Fig. C.7: Blocked pipe icon.

Blocked icons indicate a pipe blockage or very high resistance has been placed and cannot travel freely through the pipe. Blocked resistance values are set in the **PRESET** menu. If blocked resistance values are above the maximum resistance specified in the **Settings**, then absolutely no flow will be permitted along the pipe.

6.4 Appendix D - Display Problems

Occasionally the user may find that the graphics displayed on the computer screen are not as intended. The following section will step through some common problems and resolutions.

Hardware Problems

A large variety of graphics cards exists on the market today. While each type of card should perform in the same way (although some faster than others), there are many different manufacturers and software drivers that may cause differences in expected performance.

Chasm Consulting tests a wide variety of different cards to ensure optimum performance where possible.

If an issue exists, often updating the software driver from the card manufacturer can help resolve issues. Due to the low price of 3D graphics cards, if the computer or graphics card is very old (greater than 5 years), it may be better to upgrade the computer or replace the graphics cards with a newer type.

Screen fails to display after coming out of 'sleep / hibernation'

The screen remains blank, or shows corrupted data when resuming from screen savers or hibernation. This is caused by the graphics card failing to send the correct message to Pumpsim™ to enable Pumpsim™ to reset the card and restore the graphics.

Resolution – In most cases the screen can be recovered by using the **VIEW > REFRESH** menu item. In extreme cases the software may need to be shutdown and restarted. Updating the graphics cards software drivers can often help in this case. Alternatively, disable the screen saver / hibernation, and allow the computer to simply turn off the monitor instead.

Anti-aliasing does not work / is very slow

Only newer graphics cards fully support this image-smoothing feature. Older graphics cards, particular Intel motherboard types, do not support it. Low power graphics solutions from ATI or NVIDIA may support it, but graphics performance will slow.

Resolution – No solution. Install a newer video card which supports full screen anti-aliasing or turn off anti-aliasing in the settings.

Custom Icons cannot be used

Only newer graphics cards fully support custom icons. Older graphics cards (particularly Intel) may not have sufficient memory pointers to support the placement of potentially tens of thousands of icon images on the screen, therefore support for these cards has been disabled.

Resolution – No solution. Install a newer video card.

Software Problems

Occasionally, the screen will not display graphics as expected. This can stem from a wide range of settings in Pumpsim™ which may not be set correctly to display the desired graphics.

No Graphics Shown on Screen

The pipes may be hidden or invisible, or located off the screen.

Resolution – Use the **VIEW > FIT ALL** command to make sure the graphics are relocated to the screen. If no graphics show, use the **VIEW > SHOW ALL**, to turn on all hidden layers and levels. If still no graphics show, ensure that pipes are still present in the model, by running the **RUN > SUMMARY** menu option and ensuring a number of pipes are present.

Finally, make sure that options such as **VIEW > HIDE ZERO FLOWS**, and **VIEW > HIDE EXCLUDED** are not turned on.

I can see pipes, but no text / arrows / nodes

The screen may only display the raw pipes, but no other information.

Resolution – The option to display text or arrows may be turned off (press 'T' or 'A' respectively to turn back on), or use the sidebar toolbar menu. If this does not work, the **LIMIT** command may be turned on. This command hides text / nodes / arrows if the pipes is not specifically set on the **EDIT** form to show data. Turn the **LIMIT** option off using the 'L' button.

6.5 Appendix E - Simulation Errors

The following messages may be shown if errors are present during model simulation. To select and view a pipe with an error, simply click the error value in the **Error List Box** at the lower right corner of the screen. To select multiple pipes for editing in Pumpsim™, click on the items (hold **CTRL** to select multiple items) and click **SELECT**.

Warnings

Warnings can generally be ignored, as the simulation will complete despite the warnings. They may however be a sign of a more serious problem, and should still be investigated.

Pipe will be full/empty in hh:mm:ss

The flow in and out of the tank is unbalanced. The warning will only show if the time is lower than a tolerance. The tank tolerance can be changed in the **Settings > Flow Simulation** menu. If the tolerance is set to 0, no warning will show.

Resolution – None required. However the system might need modification (bigger pump?)

No entry pipe or surface connection

There is no fluid entering the pipe, therefore no flow can occur. The pipe is not connected to the surface or marked as a 'closed end'. The program can be set to ignore these warnings in the **Settings > Flow Simulation** menu.

Resolution – Simply connect a pipe to the entry, connect the pipe to the surface, or marked as a closed end in the **Edit Box**.

No exit pipe or surface connection

As with no entry warning.

No entry or exit

As with no entry warning description.

Pipe stopping redundant

A pipe stopping has been placed in series with another pipe stopping. Only one stopping is required.

Resolution – None required, although a stopping can be removed if desired.

Stalled pump

The pump is operating at a too high pressure / low volume, beyond the limits of the pump curve.

Resolution – None required, however pumps should not be made to run outside the curve. Pumpsim™ must 'guess' a result, which could lead to errors.

Low pressure pump

The pump is operating at too low pressure / high volume, beyond the limits of the pump curve.

Resolution – None required, however pumps should not be made to run outside the curve. Pumpsim™ must 'guess' a result, which could lead to errors or water balancing issues. To remove the warning, increase the resistance the pump operates at, or extend the pump curve down to zero pressure. Changing the amount of RPM can also solve this warning.

Negated pump

The pump offers no additional pressure to the model, and may actually be retarding flow.

Resolution – None required, however pumps should not run as negated. Pumpsim™ must 'guess' a result, which could lead to errors or water balancing issues. In most cases, pump negation occurs as a result of other pumps drawing water through the pump at a flow above the maximum pump curve quantity. To remove this warning, consider removing or switching off the pump.

Pipe not full

The pipe might not be full

Resolution – None required, however the pressure calculation may not be accurate for this part.

Channel overflow

Too much water flowing down the channel or drain hole too small.

Resolution – None required, but increasing channel or drain size may solve this problem.

Node not connected

A node is not connected to any pipe.

Resolution – None required, but it may be a design error.

Errors

Errors will generally halt a simulation process, and must be corrected before a valid simulation can occur.

Pipe attempted to reverse

A pipe set as 'restricted reverse' in the **Edit Box** was attempted to be reversed by the simulation. As a result, the simulation is halted.

Resolution – This function is designed to warn the user about pipes which should not have reversed flow. To remove this error, simply unclick the option in the **Edit Box** for the pipe.

Duplicate pipe

A pipe with the same entry and exit location as another pipe has been created.

Resolution – Allow Pumpsim™ to delete the duplicate pipe.

Fixed flow cannot be achieved

A pipe has a fixed flow placed in line with another fixed flow, resulting in a conflict between the two flows, or a fixed flow is placed in a location where it cannot draw water.

Resolution – Remove one of the fixed flows, or ensure a path between the flows exists to balanced the flow difference. Ensure there are no breaks in any pipes leading to or from the fixed flow, through which flow cannot travel.

Fix being over-restricted

A fixed flow encounters too much resistance to flow the required quantity causing a massive buildup in pressure.

Resolution – Check to ensure pathways before and after the fix are connected to the model or surface, and can freely flow, and no blocked resistances are present in the pathways before or after the fix. Pipes that are not exactly connected cannot carry flow. While a no entry/exit warning will normally highlight these pipes, if 'allow closed end' has been selected, then no warning will show, and the problem may be difficult to find. Try the wireframe mode with nodes displayed if this is a problem.

Operating pump blocked

A pump has been placed in a pipe through which there could be no flow.

Resolution – As with 'fix stopped', check to ensure restricting blocked resistances are not present, and the pipes before and after the pump can carry water.

Convergence problem

The model cannot find a resolution in the model flow balance.

Resolution – Ensure pumps are not running stalled or negated, and no unusual pressures or flows exist within the model.

Overpressure across resistance

Pressure building up unacceptably over a high resistance pipe, or a pump or fixed flow running into a high resistance pipe.

Resolution – Investigate and remove the source of the high pressure (pump or fix) or remove the high resistance causing the buildup in pressure.

Operating pump restricted by resistance

A resistance has been placed in the same pipe as a pump, and the pump is being unreasonably restricted.

Resolution – Turn the pump OFF instead from the **EDIT** form, or remove the pump. If the pump is turned OFF, click on **SELF CLOSING** if you want to ensure water does not leak back through the pump.

Fix being over restricted

Similar to overpressure across resistance, but the error code will target the pipe with the pump or fix, not the resistance.

Resolution – Changing the pump or fix, or alternatively removing the high resistance will correct the problem.

Unresolved pressure or flow

Pumpsim™ cannot resolve of pressure or flow volume during simulation.

Resolution – Ensure no unusual pipes, pump pressure or pathways exist within the model.

Meshing error

Pumpsim™ cannot create a workable mesh of pipe loops (meshes) required for flow and pressure balancing using the Hardy Cross method.

Resolution – Ensure that there are not too many pipes connected to the SURFACE. In addition ensure that pipes are physically connected – use the **TOOLS > FILTER** option if necessary to allow Pumpsim™ to ensure a connection.

No channel exit

Pumpsim™ cannot find the exit channel.

Resolution – Only one exit channel is allowed, it can be an open end, link to a tank or to a drain hole.

Junction in channel

A junction is present in a channel.

Resolution – Delete junction.

A channel cannot be restricted at the exit

A fix flow is present at the end of the channel section.

Resolution – Delete the fix. Fix flow can only be used at the entrance of the channel.

Channel try to reverse

Channel try to reverse during the simulation.

Resolution – Check where the flow try to flow. It could be due to channel with the wrong slope.

Only one downstream channel allowed

Several downstream pipe.

Resolution – Diverging flow are not allowed in Pumpsim™. A way to simulate similar effect can be to separate one of the downstream pipe and to place a fix flow.

No upstream flow entering connection

All the channels are diverging.

Resolution – A channel probably has too high a slope. Adjust the channel or change **SETTINGS > SIMULATION > CHANNEL > Minimum Slope** so the channel can be considered flat and be reversed.

No downstream connection

All the channels are converging.

Resolution – A channel probably has too high a slope. Adjust the channel or change **SETTINGS > SIMULATION > CHANNEL > Minimum Slope** so the channel can be considered flat and be reversed.

End of channel missing (surface, tank or drain)

A channel section must have only an end.

Resolution – Specify where the water will flow by putting a surface (with no pump or valve), a tank or a drain at the end of the section.

Too many pipes entering the drain, only one allowed

Only one pipe or channel can enter a drain.

Resolution – Join the pipes before the drain and link to the drain by a small channel section.

Wrong tank connection

Connection with the tank below the water level.

Resolution – Move the channel, tank or water level so the water can enter the tank.

6.6 Appendix F - Shortcut Keys

A number of keyboard strokes exist to assist in rapid selection of commonly used functions. Most menu items have the shortcut key next to the menu item. Below is a general summary of common key strokes

CTRL N	New File
CTRL O	Open File
CTRL M	Merge File
CTRL I	Inherit File Attributes
CTRL S	Save File
CTRL F	File Note
CTRL C	Clone Attributes
CTRL V	Apply Attributes
CTRL L	Lock On Target (Toggle)
CTRL T	Lock On Transparency (Toggle)
CTRL Q	Quick View
LEFT Arrow	Move to previous Quick View
RIGHT Arrow	Move to next Quick View
HOME	Fit All
INSERT	Insert Node
PAGE UP	Select / Move Up one elevation level (if defined)
PAGE DOWN	Select / Move Down one elevation level (if defined)
END	Show ALL elevation levels
DEL	Enter delete mode (select / fence ways to delete)
BACKSPACE	UNDO
CTRL Z	UNDO
TAB	REDO

F1	HELP
F2	Plan / Section Views (Toggle to cycle through views)
F3	FIND (Toggle to repeat last find command)
F4	FIND and HIGHLIGHT found objects
F5	Simulation Flow
F6	Simulation Thermodynamic
F7	Hide / Show Zero Flow
F8	Hide / Show Excluded
F9	Transparencies on/off
F12	Save As
T	Show / Hide text data
L	Show / Hide limited data
Z	Zoom Mode
D	Draw or Add pipe mode
E	Edit Mode
S	Select Mode
M	Move Mode
C	Copy Mode
B	Block Mode (for blocking / unblocking)
R	Reverse Mode (for reversing objects)
Z	Zoom Out
P	Perspective / Orthogonal Mode Toggle
W	Toggle between Wireframe and Solids
G	Display / Hide Grid
N	Display / Hide Nodes
A	Display / Hide Arrows
I	Display Hide Icons