



PIPENET® - Leading the Way in Fluid Flow Analysis

CONTENTS:

Welcome!

Latest release of PIPENET!

User Experience

Data Window as a Spreadsheet

Resetting the Datum for Node Elevations

More about Forces and Piping Design

Interfacing with CAD and 3D Modellers

Initial State in PIPENET Transient module

Updated Sunrise Systems web site!

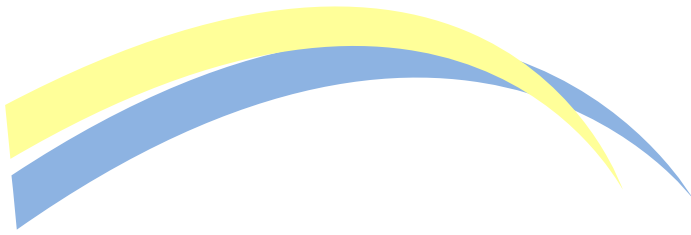


We are delighted to send you the latest edition of **PIPENET News!**

If you are already a **PIPENET** user we will give you news which will further increase your enjoyment and effectiveness with **PIPENET**. If not, we will give you more reasons why you should be!

PIPENET 1.5.2 OUT NOW!

Our current and best release yet is **PIPENET 1.5.2**. If you have any questions, please contact us at pipenet@sunrise-sys.com



Latest Release 1.5.2

Since its release at the start of 2011, **PIPENET 1.5** has been extremely successful. We described its contents in an earlier newsletter to you.

We have produced two service packs since its launch. Our latest release is **PIPENET 1.5.2**, contains a number of improvements:

Number of Decimal Places Displayed for Dimensionless Quantities

There are a few units such as “nozzle deviation” that do not have dimensions (in the case of “nozzle deviation” the number is a percentage). In **PIPENET Vision 1.5.2**, the number of decimal places shown for these quantities in schematic drawings is reduced to 2 which makes the drawings much more legible.

The new releases also contain a number of smaller fixes and improvements, and support for new features by our Marketing Partners (such as new operating system releases by Microsoft).

MUS!

MUS (Maintenance Update and Support) is an excellent affordable service giving customers first class technical support and upgrades. Up to date versions of our training materials, bulletins, demonstrations and other support materials are also available free of charge to **MUS** customers. Enjoy the reassurance of having the latest software at your finger tips with outstanding support!

User Experience

Here are some recent ones from our user base:

“PIPENET has really turned out to be a great tool for our Firewater System Design. I highly recommend it.” Mickey

“I think your latest release is great!” Mike

“You are a magician! Thanks a lot and good week end!” Giovanni

“Many thanks and much appreciation for your timely advice to resolve the problem. We were stuck with that case, but now we are going ahead with the other cases.” Sarath

PIPENET has been in engineering use for more than 26 years. It is your Natural Choice!

Using the Data Window as a Spreadsheet

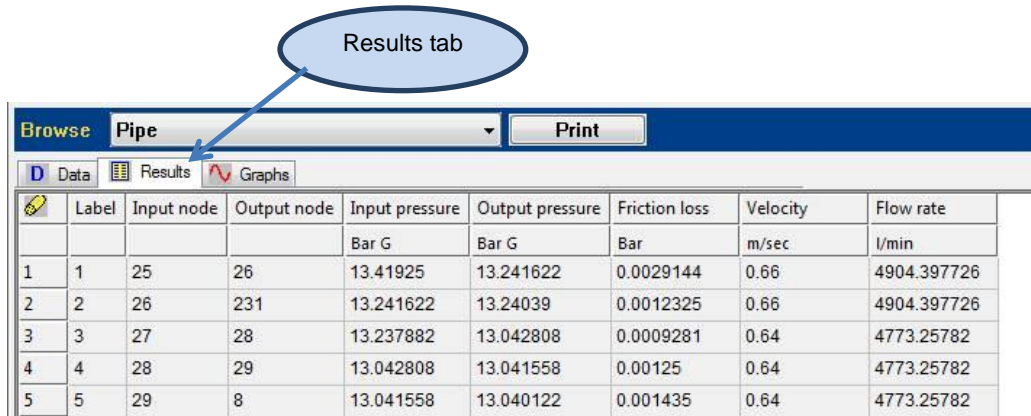
Customers have asked us how they can use **PIPENET** to perform “spreadsheet” type operations within **PIPENET**. In other words, they want to view tables and schematic drawings of data, and make **PIPENET** manipulate the data in an easy-to-use way.

Users often think of this as being a kind of clipboard/cut-and-paste operation, but actually the **PIPENET** Data Window provides an excellent way to do this. The data window in **PIPENET** enables the user to do much more. Some examples are shown below. The same technique can be used for virtually any parameter in the data window. For example, it is possible to locate the most remote nozzle on the schematic just by a few mouse clicks.

1. The **PIPENET** Data Window behaves somewhat like a spreadsheet. Global editing and sorting are two powerful and useful functions. As it is integrated within **PIPENET**, you can do things with it which you cannot do with a separate spread sheet.
2. To take a particular example, let us look at the maximum velocity in pipes which is usually of fundamental interest to users. In **PIPENET** this can be done quickly and we can go straight to the pipe with the maximum velocity.
3. Once the calculation has been performed you can follow the steps show below.
 - (i) In the data window make sure that ‘Pipe’ has been chosen and click on the results tab. The window will look something like this.

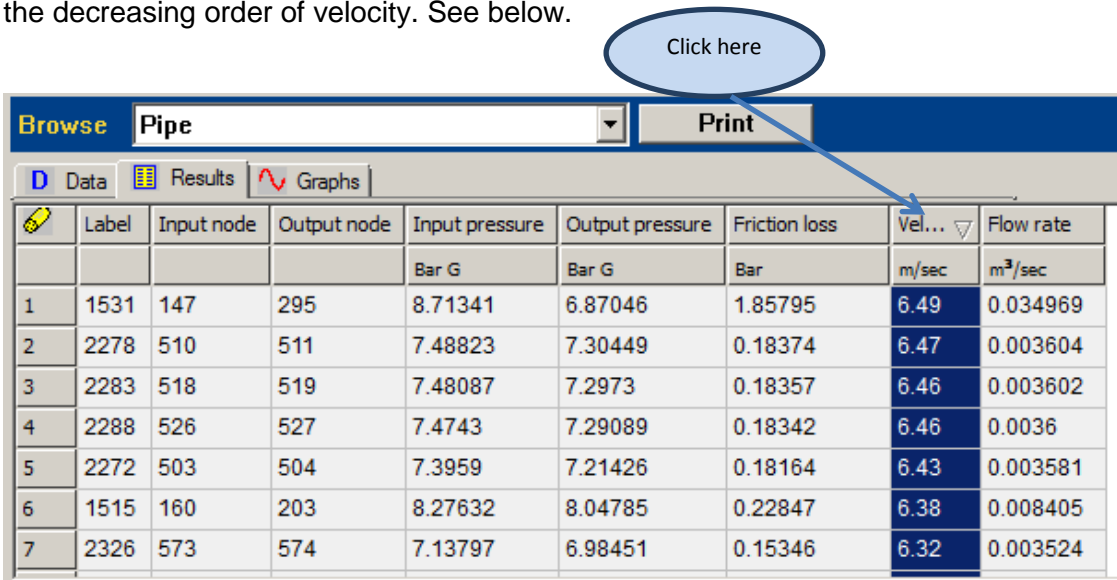
	Label	Input node	Output node	Input pressure	Output pressure	Friction loss	Velocity	Flow rate
				Bar G	Bar G	Bar	m/sec	l/min
1	1	25	26	13.41925	13.241622	0.0029144	0.66	4904.397726
2	2	26	231	13.241622	13.24039	0.0012325	0.66	4904.397726
3	3	27	28	13.237882	13.042808	0.0009281	0.64	4773.25782
4	4	28	29	13.042808	13.041558	0.00125	0.64	4773.25782
5	5	29	8	13.041558	13.040122	0.001435	0.64	4773.25782

PIPENET – The Engineer’s Choice!



	Label	Input node	Output node	Input pressure	Output pressure	Friction loss	Velocity	Flow rate
				Bar G	Bar G	Bar	m/sec	l/min
1	1	25	26	13.41925	13.241622	0.0029144	0.66	4904.397726
2	2	26	231	13.241622	13.24039	0.0012325	0.66	4904.397726
3	3	27	28	13.237882	13.042808	0.0009281	0.64	4773.25782
4	4	28	29	13.042808	13.041558	0.00125	0.64	4773.25782
5	5	29	8	13.041558	13.040122	0.001435	0.64	4773.25782

(ii) Now click on the cell with the heading 'Velocity'. You will find that PIPENET re-orders the pipes in increasing order of velocity. Click on the heading 'Velocity' again. PIPENET will now reorder the pipes in the decreasing order of velocity. See below.



	Label	Input node	Output node	Input pressure	Output pressure	Friction loss	Vel... ▾	Flow rate
				Bar G	Bar G	Bar	m/sec	m³/sec
1	1531	147	295	8.71341	6.87046	1.85795	6.49	0.034969
2	2278	510	511	7.48823	7.30449	0.18374	6.47	0.003604
3	2283	518	519	7.48087	7.2973	0.18357	6.46	0.003602
4	2288	526	527	7.4743	7.29089	0.18342	6.46	0.0036
5	2272	503	504	7.3959	7.21426	0.18164	6.43	0.003581
6	1515	160	203	8.27632	8.04785	0.22847	6.38	0.008405
7	2326	573	574	7.13797	6.98451	0.15346	6.32	0.003524

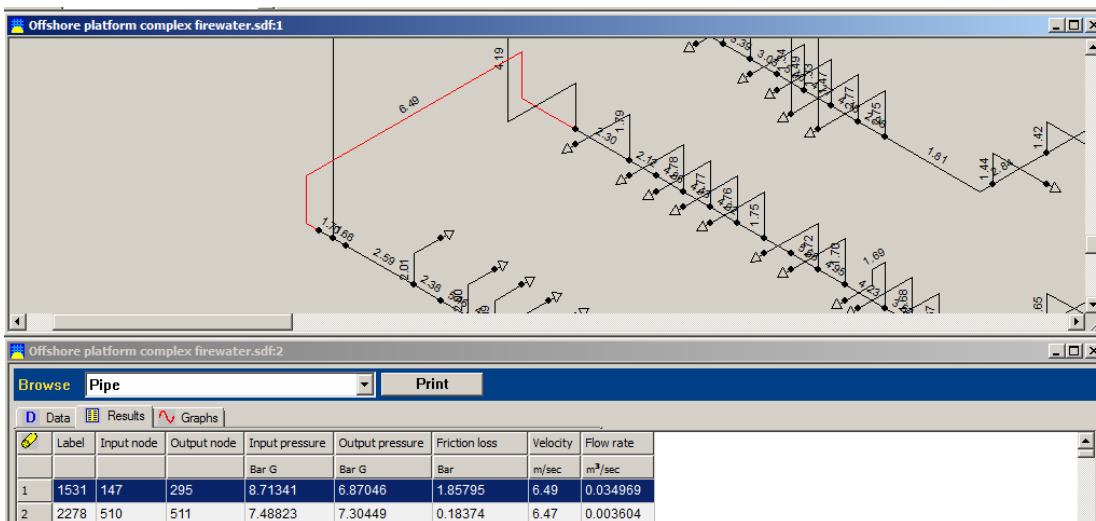
(iii) Now keep your eye on the schematic drawing and click on the left cell of the first row. This is the cell on the left with the number 1. You will see that PIPENET shows you, on the schematic, the pipe with the maximum velocity. Please see below. You can do this for any variable – pressure, flowrate, node number etc. Easy!

You can see that PIPENET takes you straight to the pipe with the maximum velocity. Just with a few mouse clicks a very time consuming and important operation has been performed.

PIPENET – the Name to Trust!

	Label	Input node	Output node	Input pressure	Output pressure	Friction loss	Vel...	Flow rate
				Bar G	Bar G	Bar	m/sec	m³/sec
1	1531	147	295	8.71341	6.87046	1.85795	6.49	0.034969
2	2278	510	511	7.48823	7.30449	0.18374	6.47	0.003604
3	2283	518	519	7.48087	7.2973	0.18357	6.46	0.003602
4	2288	526	527	7.4743	7.29089	0.18342	6.46	0.0036
5	2272	503	504	7.3959	7.21426	0.18164	6.43	0.003581
6	1515	160	203	8.27632	8.04785	0.22847	6.38	0.008405
7	2326	573	574	7.13797	6.98451	0.15346	6.32	0.003524

PIPENET will take you straight to the pipe with the maximum velocity.

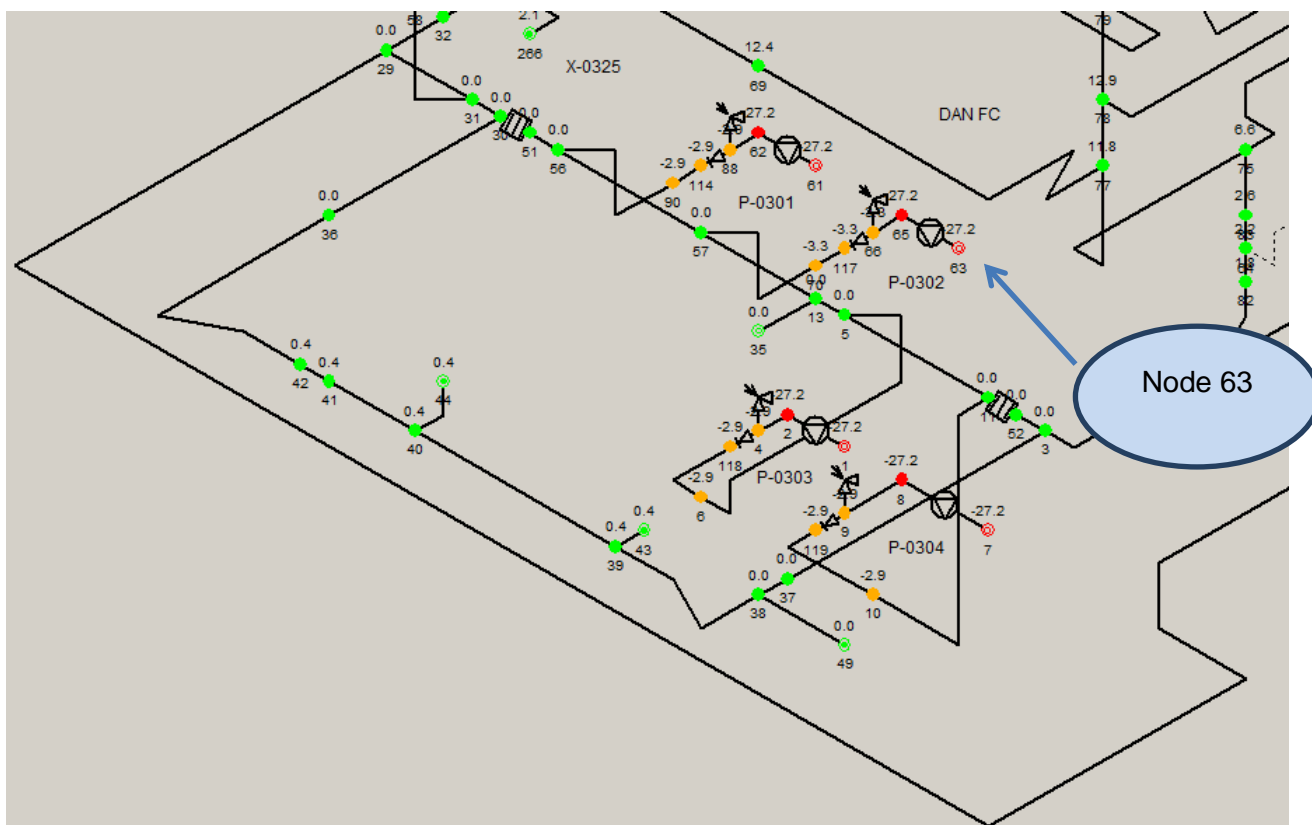


How to Reset the Datum for Pipe Elevations

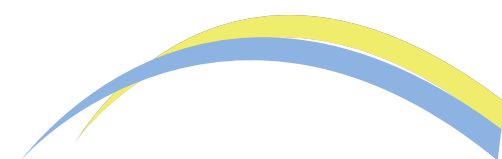
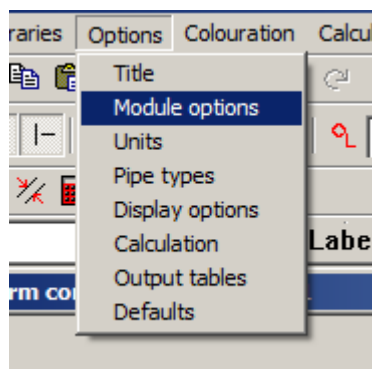
Here a user wants to adjust the network so that the datum used for the elevations is changed. This would be quite straightforward if it was just one item. However, generally the elevations of other parts of the network have to be adjusted consistently so that the calculation can proceed correctly and give the right results. The user may wish to do this because it is necessary to track down the cause of an elevation error. For this purpose it may be useful to have the same datum as the piping isometric drawings.

In the following case the network has four pumps whose input nodes are at an elevation of -27.2 m. This is based on the main deck of the platform being the datum. It is necessary to re-set the input node of the pump to be at an elevation of 29 m. The rest of the network needs to be adjusted accordingly

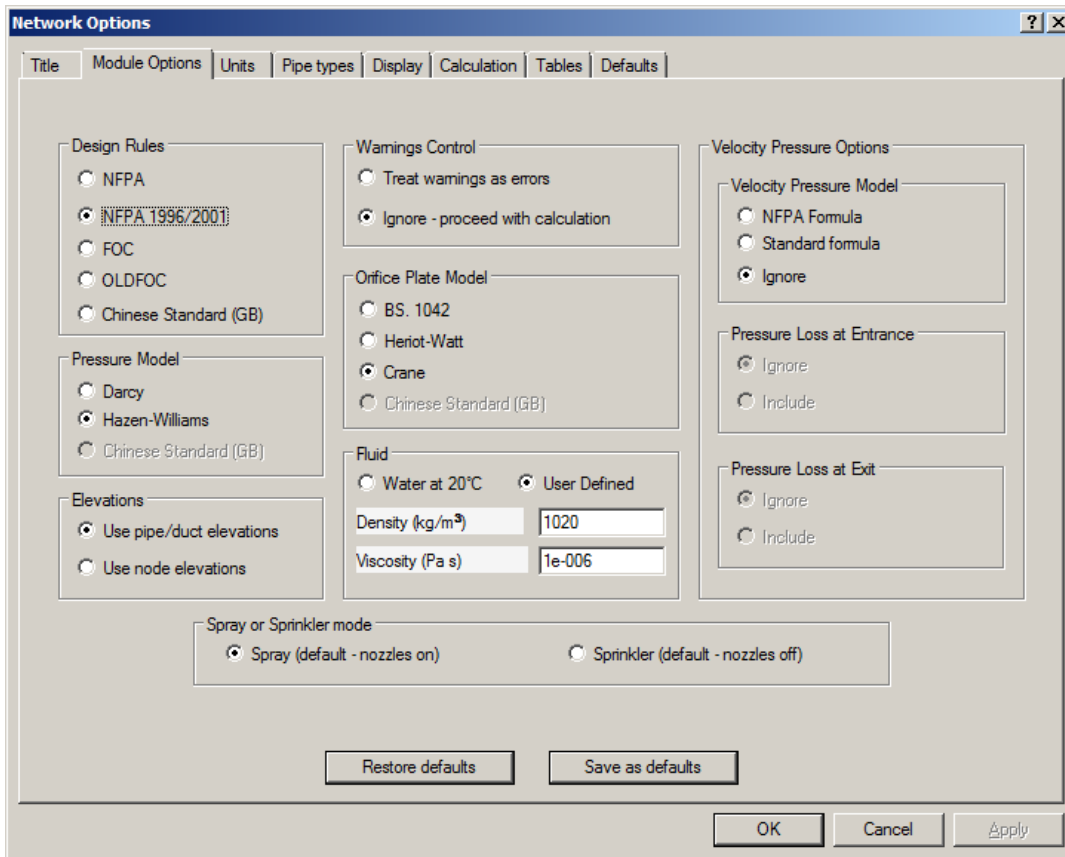
and all the nodes have to be reassigned new elevations.



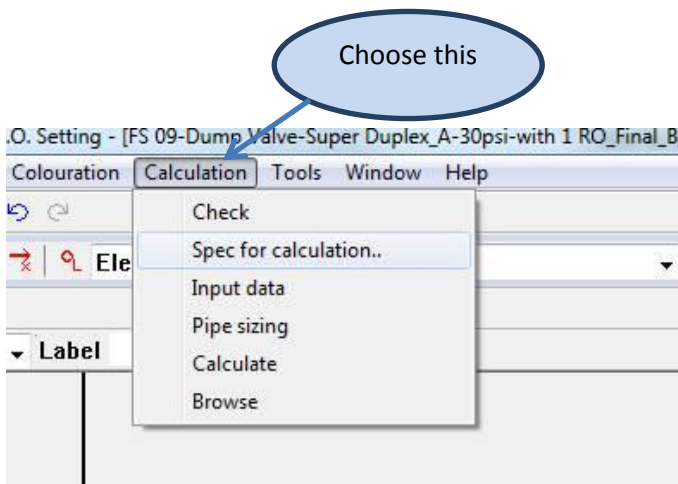
Make sure that you are using pipe elevations. Under Module Options choose the following.



PIPENET – Excellence in Engineering Every Day!

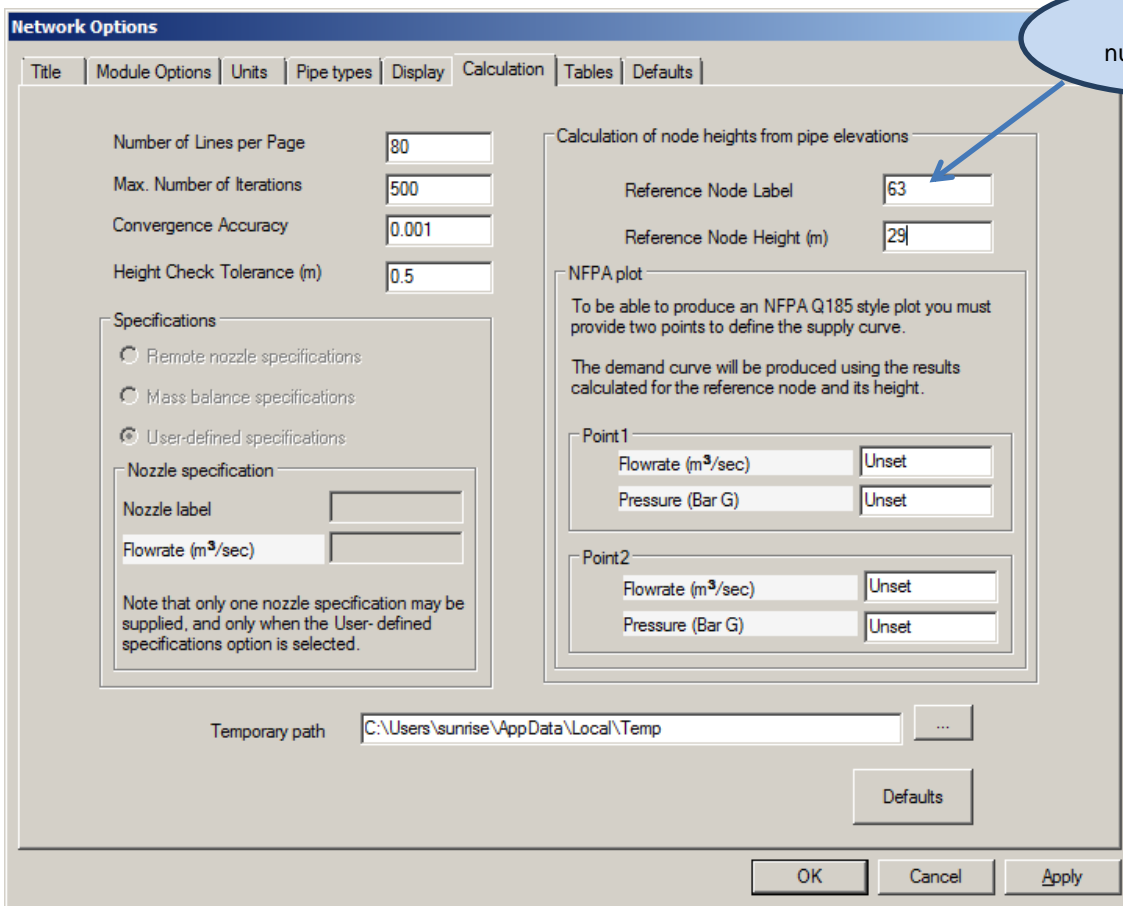


1. Click on Calculation/Spec for Calculation...

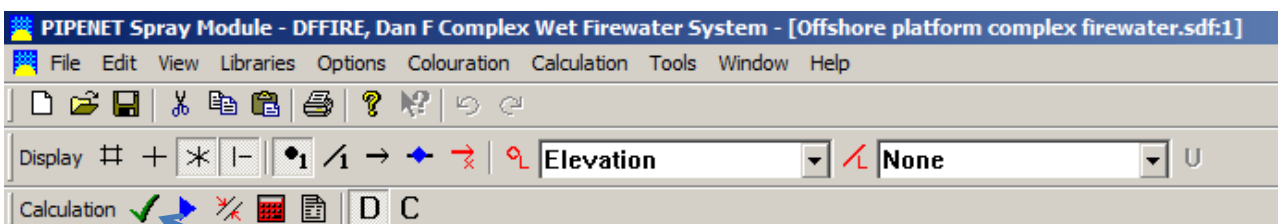


2. In this case the elevation of node 63 to be set at 29 m, and all other nodes must reflect the change which has been made. In other words, the elevations of all other nodes must be reassigned.

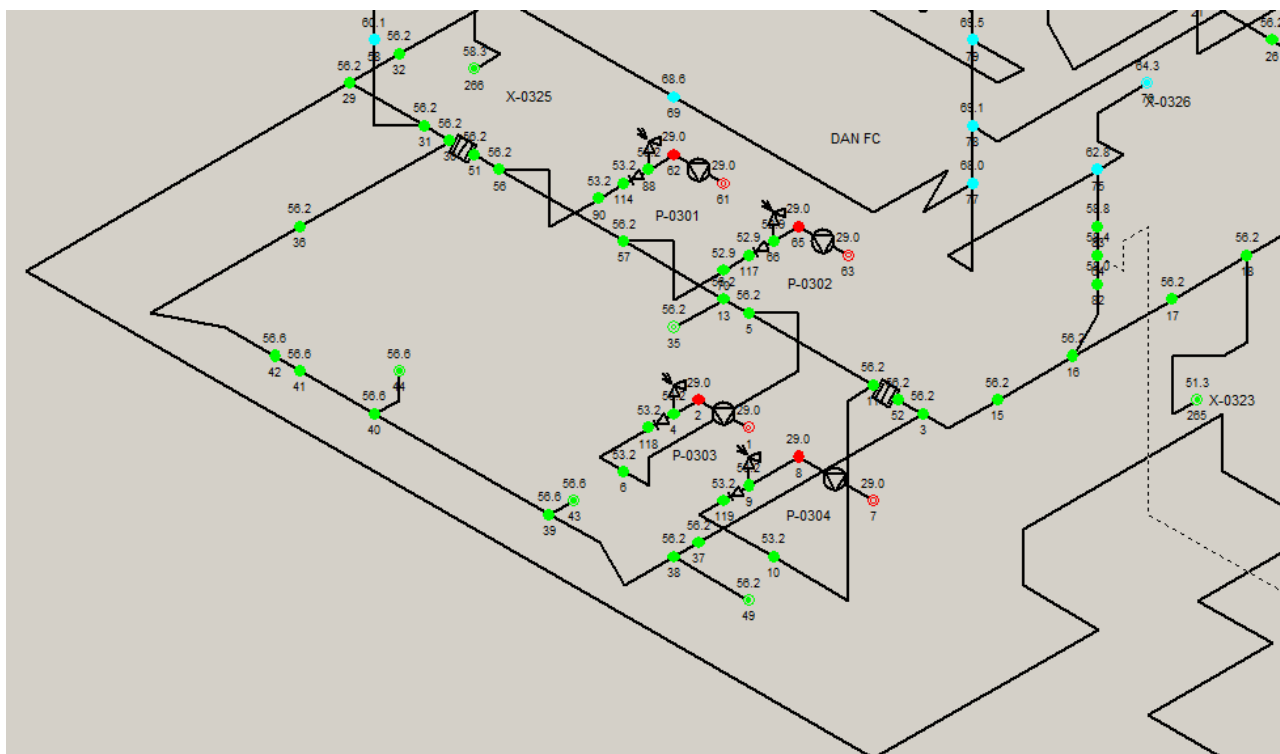
Make the change in the dialog box as shown below.



3. When you click on OK, **PIPENET** will reassign all the node elevations. However the new values for the node elevations will not show on the **PIPENET** schematic until you take one more step. This is because the drawing does not refresh automatically. You must click on the 'tick mark' as shown below



You can see the reassigned node elevations below.

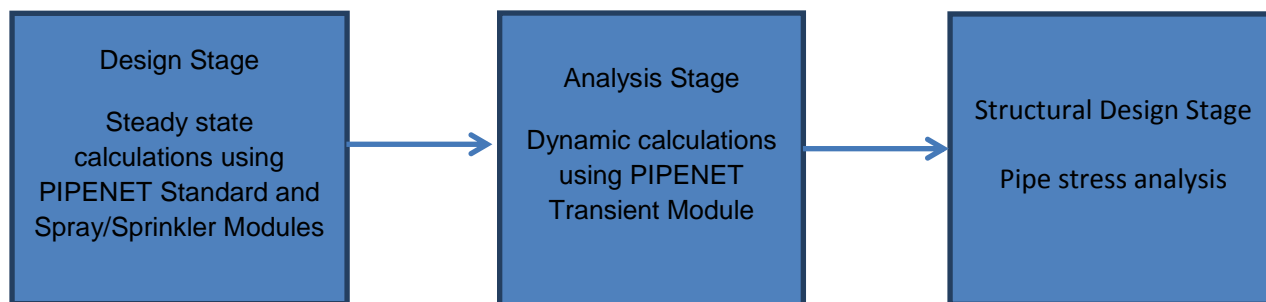


More about Forces

Can PIPENET calculate forces which can be used for pipe support and anchor design?

Yes, of course. We have always recognised the importance of forces. We are the leaders. Even the very first version of **PIPENET** Transient module had the capability of calculating dynamic forces. In **PIPENET** terminology the kind of dynamic force needed for this purpose is called the ‘simple’ force. Essentially, this is the unbalanced force in straight sections of pipe. These are usually one-dimensional forces and are in the direction of the pipes. They are called **simple** forces in **PIPENET** terminology. Pipe stress analysis programs such as Caesar II have interfaces with **PIPENET**. They can automatically import the time history generated by **PIPENET**.

The overall philosophy which is used in **PIPENET** is shown below.



Two very important conceptual differences between **PIPENET** and pipe stress analysis programs are described below.

- (i) If a disturbance is made to fluid in a pipe network system, it will travel throughout the connected parts of the network at the wave speed. In comparison pipe stress is essentially a local phenomenon. It can generally be assumed that forces and stresses do not go past anchors.
- (ii) When it comes to flow analysis the exact 3 D geometry of the network usually is not important. It is the length of pipes and elevation changes which are important. Fittings can be regarded as attributes of pipes. Items such valves which do operate are regarded as models.

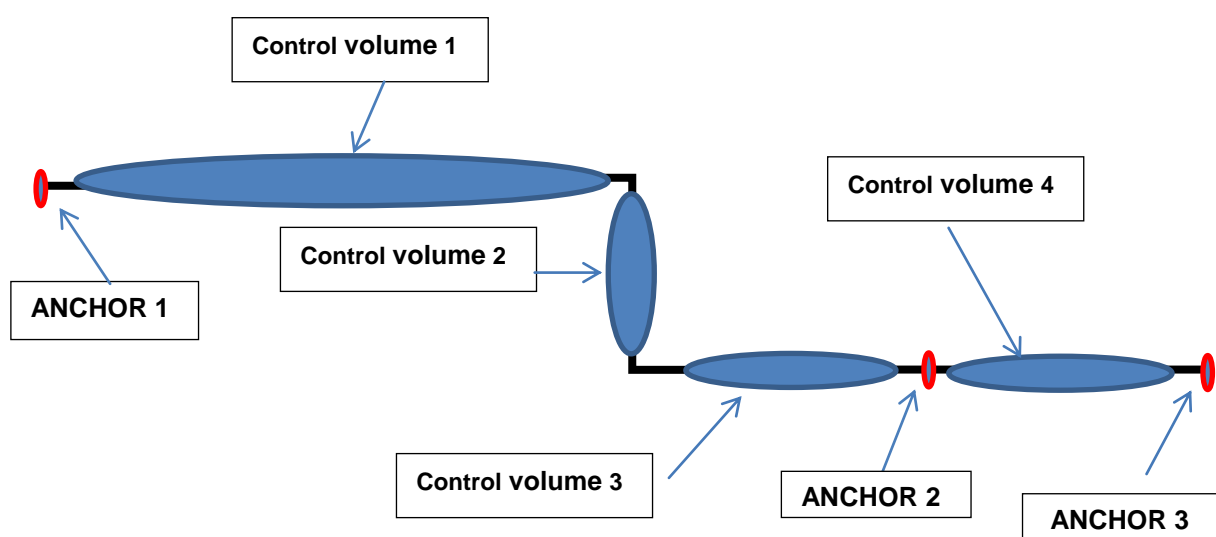
We also recognise that it would be extremely inconvenient for a professional engineer to input the drawing as a piping isometric into **PIPENET**. It could take 10 times as long to input the piping drawing as an isometric into **PIPENET**. That is why the network in **PIPENET** is represented as a schematic drawing. On the other hand, it is essential in pipe stress analysis to work with isometrics data albeit in relatively small parts of the network. The force calculation capability in **PIPENET** is intended to bridge this dichotomy.

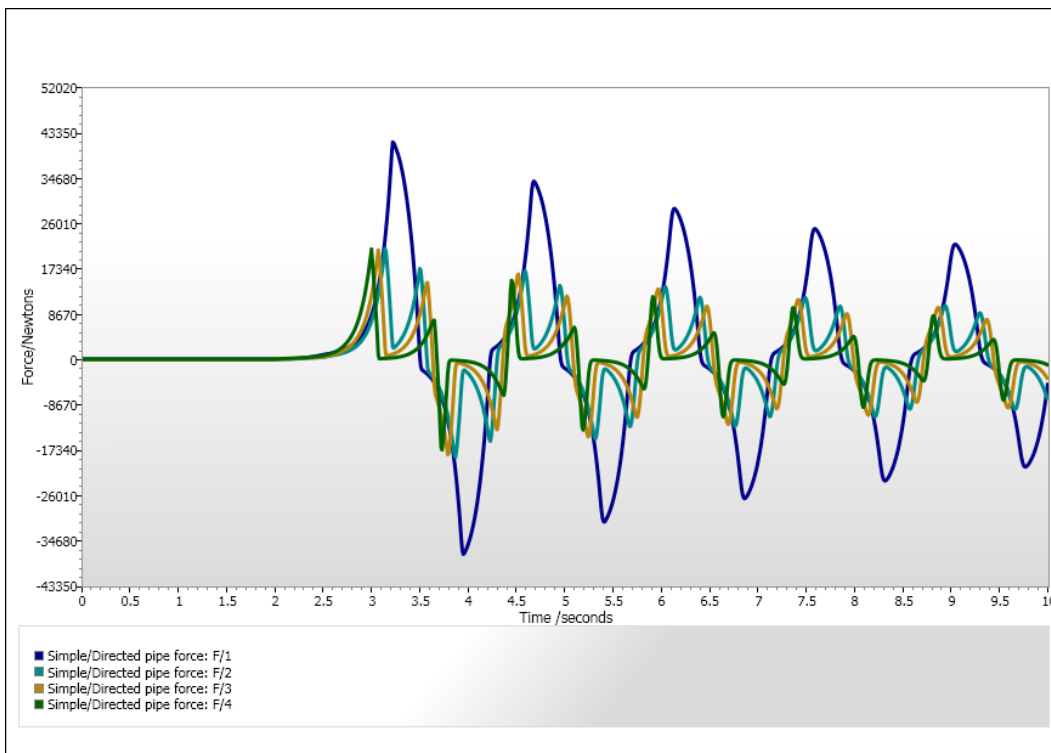
The forces calculation capability in **PIPENET** was developed with pipe stress analysis in mind. For this reason it recognises the following;

- (i) **PIPENET** analysis will have to be carried out by defining the complete network.
- (ii) For each such case the pipe stress engineer would require the forces to be calculated for a large number of load cases. Furthermore, care has to be taken to take into account that the forces should lend themselves to be defined and calculated quickly for each load case. **PIPENET** is designed in such way that the forces for the load cases can be calculated easily without changing the basic network.

Simple Forces

Even in steady state conditions the elbows will experience non-zero hydraulic forces. However the unbalanced forces will be zero because the components will cancel each other. However, under pressure surge conditions the unbalanced forces can be very large.

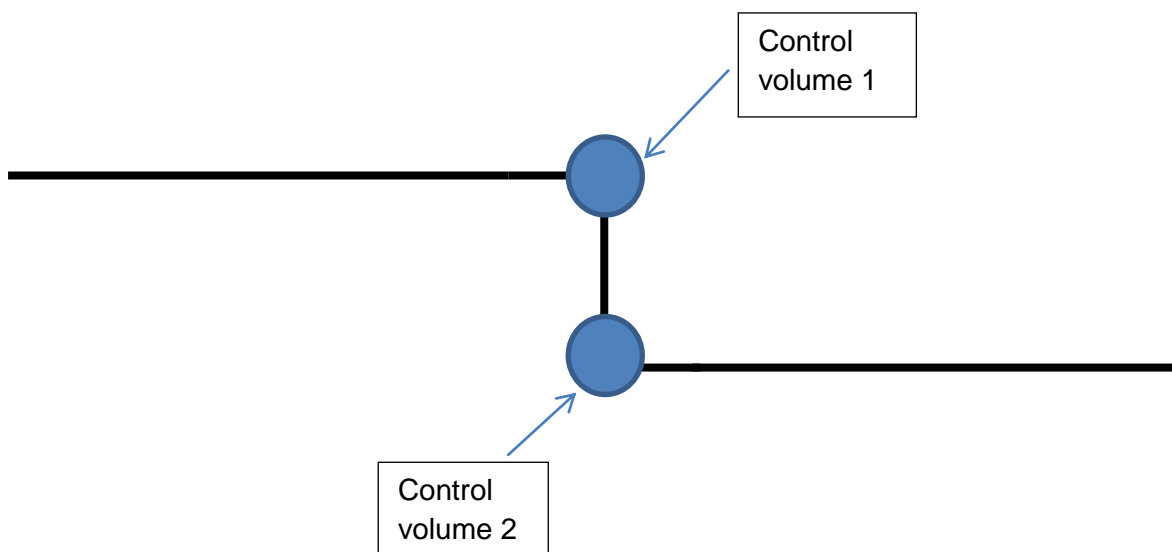


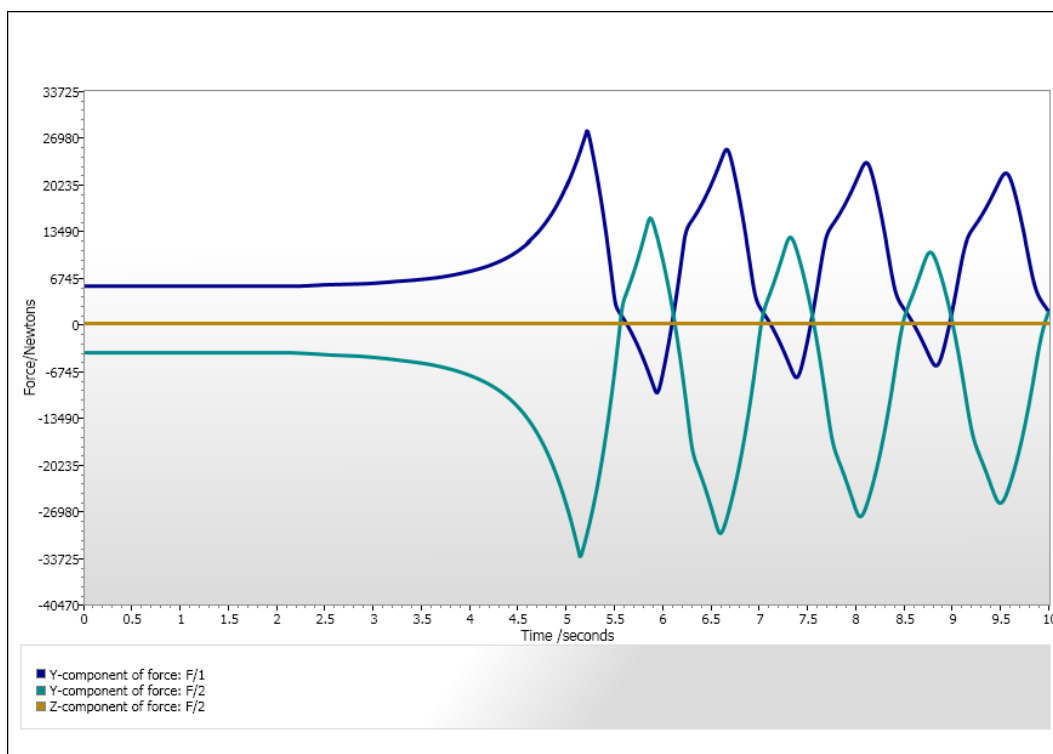


Complex Forces

Complex forces are 3 dimensional forces. So, each force will have 3 components. It is typically associated with a change in direction. The main difference here is that complex forces are 3 dimensional forces. It is necessary to create nodes at the positions where unbalanced forces need to be calculated.

Complex forces again are based on defining control volumes.





Note that in the above graph shows three components of one complex force.

Forces and Pipe Stress Analysis Interfaces

PIPENET can generate an ASCII file specifically for the purpose of interfacing with pipe stress analysis programs. The time step for this can be individually specified. The time history for all the forces defined in the simulation will be output to this file. This file has the extension .frc. An extract from the .frc file for one of the simple forces case is shown below.

Time units : seconds

Force units : Newtons

Output interval : 0.010 seconds

Number of forces tabulated : 4

Number of points per table : 1001

DIFF : F/1

.220000E+1 0.115313E-02

.221000E+1 0.114627E-02

.222000E+1 2.17615

.223000E+1 7.89779

.224000E+1 14.1450

.225000E+1 20.4487

Interfacing with CAD programs and 3D Modellers

We are often asked about whether **PIPENET** can be interfaced with other systems that civil engineers use.

A program such as generic CAD program is not an “intelligent” system - it only deals with graphical entities such as lines and curves. For example, it does not know the difference between a pipe and a cable. **PIPENET** can import a CAD drawing as a graphic underlay, over which the network can be drawn. However **PIPENET** does not assume any special knowledge other than this.

On the other hand, a specialised 3D modeller is intelligent and knows the difference between a cable and a pipe. Some companies have developed interfaces between 3D modellers and **PIPENET**, which can directly create **PIPENET** drawings.

Part 1: Typical Workflow in Engineering

The typical workflow in the engineering design of a pipe network system typically has the following sequence.

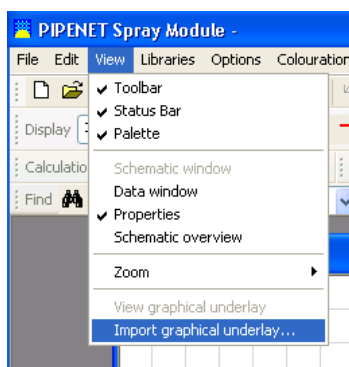
- (i) A plot plan or a general arrangement drawing is given to a design engineer.
- (ii) The design engineer will sketch out a rough layout of the network. He may also indicate piping diameters etc.
- (iii) The piping department will turn this into a piping isometric.
- (iv) The design engineer will do further calculations based on the piping isometric

There will of course be revisions after this. However, the above are the four main steps for creating the basic system. These steps are followed whether the calculations and drawings are done manually or by using software.

Part 2: Typical workflow when using **PIPENET**

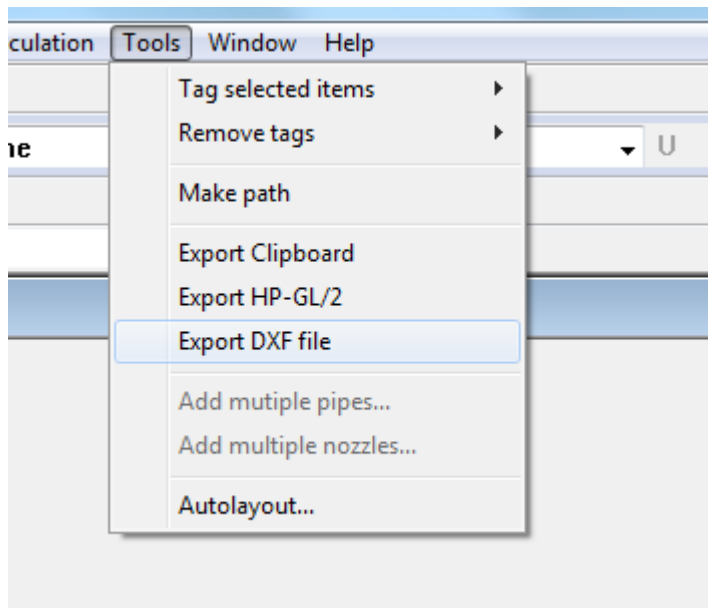
PIPENET is designed to replicate the workflow as described in Part 1 above.

- (i) Import a plot plan or a general arrangement drawing as a .dxf file.



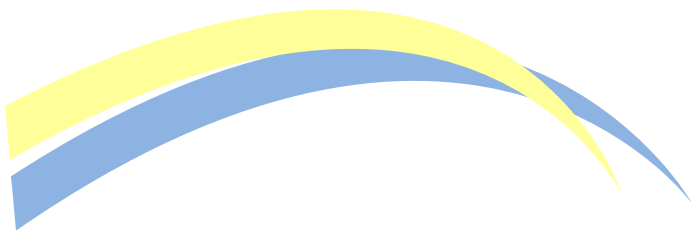
(ii) The design engineer can draw the schematic network in **PIPENET** using this as a backdrop. He also performs preliminary calculations and determines the pipe sizes and so on.

(iii) The **PIPENET** schematic can be then exported as a .dxf file for reading by a CAD program, as below. Under the tools menu of **PIPENET** it is possible to export a .dxf file which can then be imported and viewed in a CAD program. (Please note that it is also possible to export an HPGL file for using with plotters.)

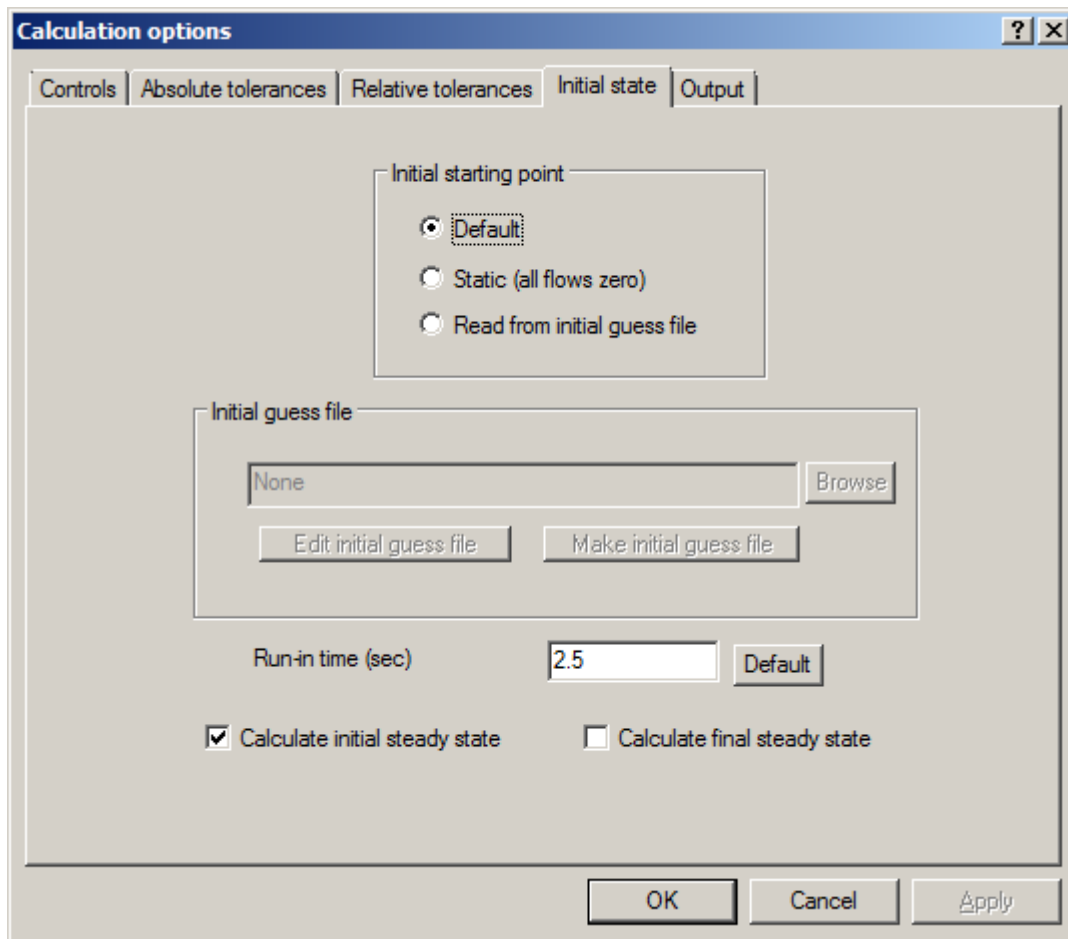


(iv) The piping department will route the pipes in 3D and turn this into a proper drawing. Please note that it will not change the connectivity of the network as it is shown in **PIPENET**. It will only change the attributes such as lengths of pipes, the fittings and so on. For practical purposes the schematic already created in **PIPENET** can still be used.

(v) The user goes back to the schematic he created in **PIPENET** and edits the attributes to reflect the changes to items such as pipe lengths and fittings. He will then have a final schematic drawing with attributes which reflect the piping isometric drawing. He can perform the calculations as required.



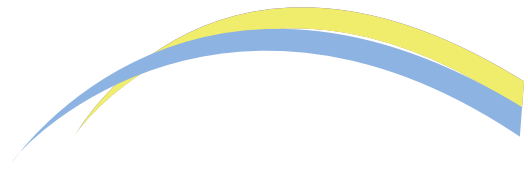
Initial State in PIPENET Transient module



The initial steady state is sometimes difficult to determine. **PIPENET** has a number of tools to assist the user in this. The Initial State dialog box is for this purpose.

The principle of 'Run-in time' is one of the most important to understand. To put it quite simply, it is a very good way of generating sound initial guesses. We can start from the meaning of the term steady state. The mathematical definition of steady state is that the partial derivatives of all the variables with respect to time are 0. These equations are sometimes difficult to solve. In contrast if the state of the system is known at one point in time, it is usually a stable process to take a step in time. So, rather than start with the mathematical definition of steady state we can start with a more pragmatic definition. "A stable condition that does not change over time or in which change in one direction is continually balanced by change in another." Basically, this means if we start with a random state of the system and run in it for an infinitely long time without changing the operating conditions, it will ultimately reach steady state. In fluid flow terms it means if we start with a random state of the system and run it for sufficiently long time, the system will reach its steady state. Run-in time is the time we allow the simulation to run in **PIPENET** using the boundary conditions at time = 0. So, the run-in time can be used in two different ways.

(i) Use a long run-in time which is long enough for the steady state to be reached. The "Calculate initial steady state" button is usually unchecked under these conditions. This is the preferred option if the system has control loops.



(ii) Use a run-in time which is shorter but which would nevertheless produce a good initial guess starting from a random guess. The “Calculate initial steady state” button must be checked under these conditions.

There are many other ways the initial state can be specified or the initial guesses generated. These tools are not often used but they are there if needed.

(i) Then initial guess file has two purposes.

(a) It can be the initial starting point for the simulations. This could for example be used as a re-start file. If this is the case the run-in time should be set to 0 and the “Calculate initial steady state” button is usually unchecked.

(b) The user can specify the initial starting guess for finding the steady state. This could be used in conjunction with the run-in time and/or “Calculate initial steady state”.

The initial guess file has a pre-defined structure and data can be entered/modified by the user. The “Make initial guess file” and “Edit initial guess file” buttons are for this purpose.

(ii) Default.

PIPENET uses its own default values using which the steady state calculation process will start. It is generally use in conjunction with run-in time and Calculate initial steady state button being activated.

(iii) Static (all flows zero)

Remember that static means all flows are 0 and is different from steady state. In some cases it may be known that all the flows will be 0 at the start. In such cases this may be a better to generate the initial guesses on the assumption that the flows are 0.

It is also important to bear the following point in mind. It is always a good idea to run the system in **PIPENET** without introducing events (changes to the boundary conditions) for a certain length of the simulation. We can then assess if steady state has been reached by looking at the graphs.

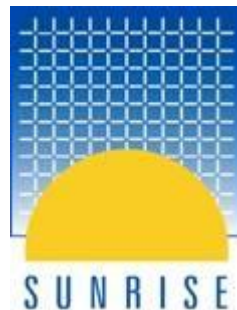
Updated Sunrise Systems Web Site!

Our web site is being improved and will become available shortly. Cosmetically it may look similar. Behind the scenes many excellent **PIPENET** tools are provided at your fingertips. Look out for the release of our updated web site. Go to <http://www.sunrise-sys.com/news.asp>

CONTACT US:

Technical Support: support@sunrise-sys.com

Sales: pipenet@sunrise-sys.com



SUNRISE SYSTEMS LIMITED, CAMBRIDGE CB25 9QZ, UNITED KINGDOM

Web site: www.sunrise-sys.com

E-mail: pipenet@sunrise-sys.com