PIPENET® NEWS

AUTUMN 2018

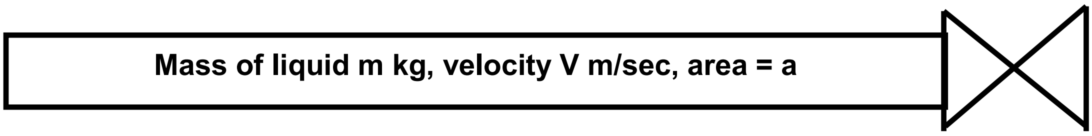
LEADING THE WAY IN FLUID FLOW ANALYSIS

IN THIS ISSUE…

Fundamentals of Transient Analysis: Intuitive Explanation of Pressure Surge and How It Is Calculated in PIPENET

PIPENET Transient Module is widely used around the globe for pressure surge analysis. Engineers have mastered its functions and can successfully perform rigorous transient analysis without difficulties. However, it is always important to keep in mind the basic principles which explain why pressure surge appears, in the first place. Such fundamentals help to understand intuitively the pressure surge phenomenon.

INTUITIVE EXPLANATION OF PRESSURE SURGE



Initial momentum of liquid	=	m x V	Valve closes in t sec
Momentum after t secs	=	0	
Change in momentum	=	m x V – 0	
Rate of change of momentum	=	m x V /t	
Force	=	m x V / t	
Increase in pressure	=	m x V / (t x a)	

WHY CAN WE NOT USE THIS IN ENGINEERING?

1. It does not consider the influence of elasticity, compressibility and wall thickness.
2. It assumes that the flowrate changes uniformly with time.
3. It cannot model several events taking place.
4. It cannot be used for items other than pipes.

JOUKOWSKY EQUATION

The fundamental equation in water-hammer theory relates pressure changes, Δp, to velocity changes, Δv, according to

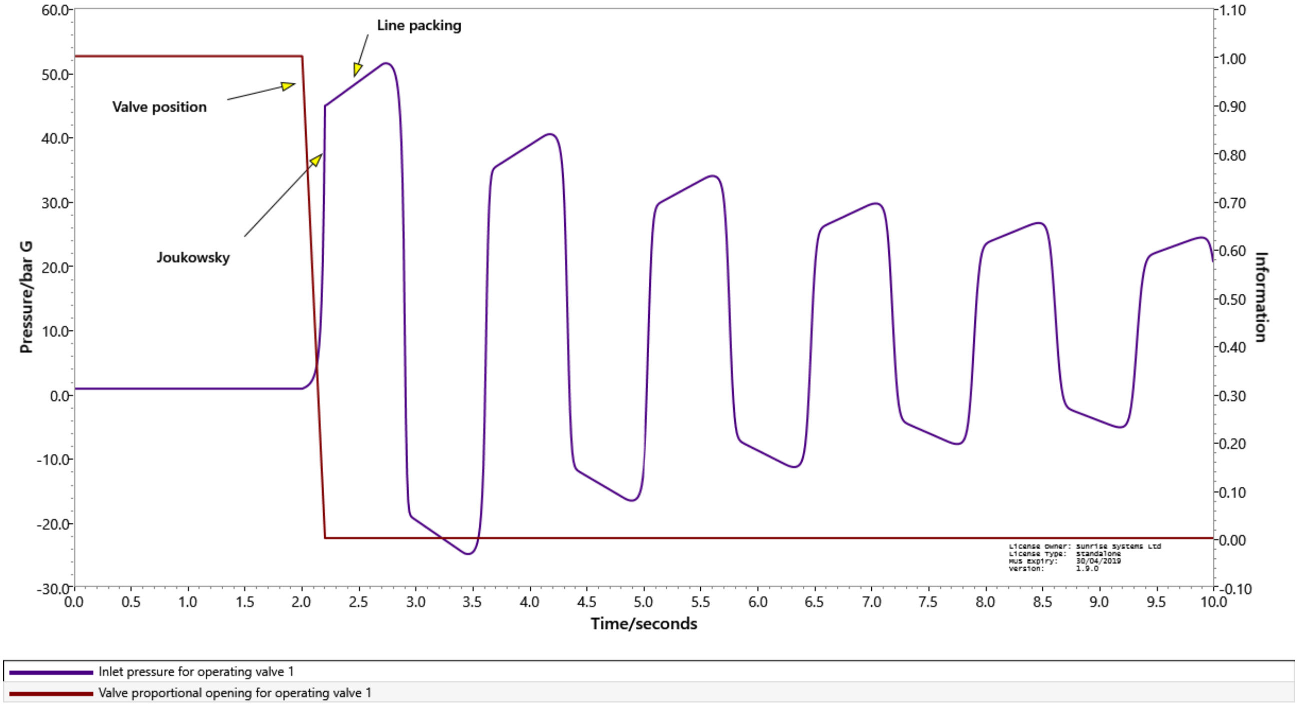
$$\Delta P = -\rho c \Delta v$$

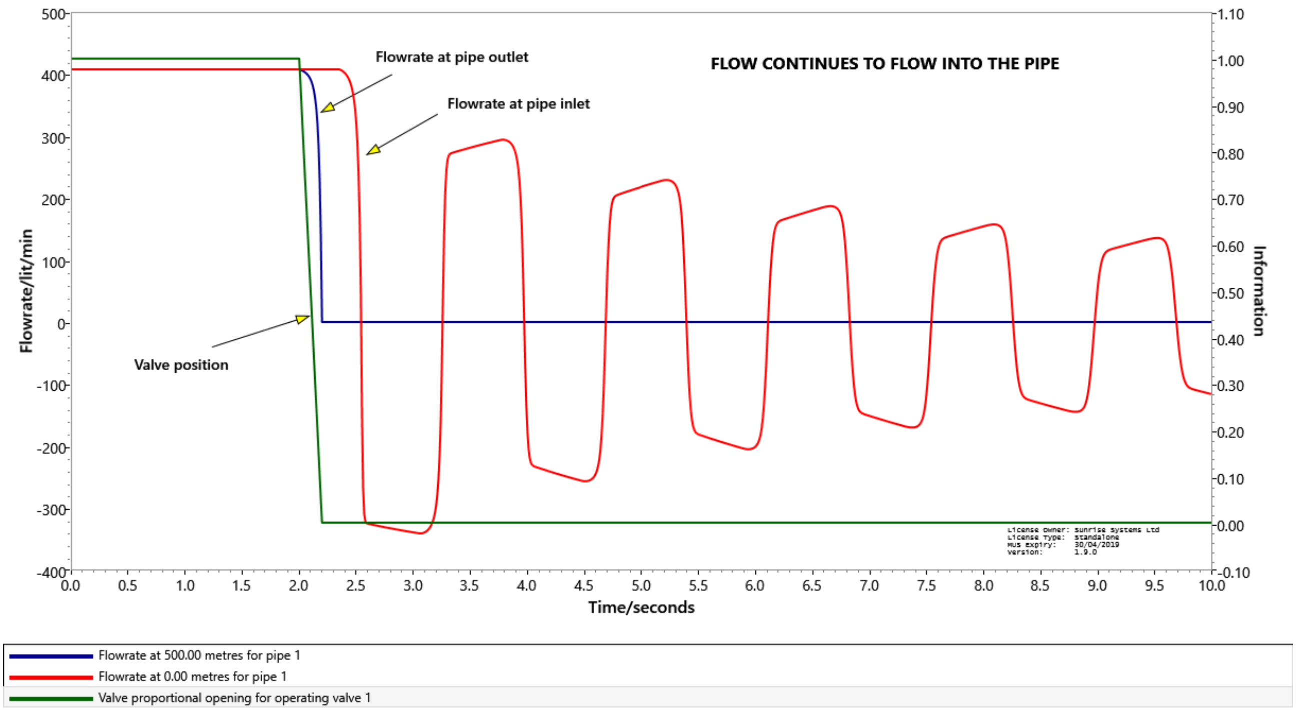
where ρ is the fluid mass density and c is the wave speed.

$$C = \sqrt{\frac{1}{\rho\left[\frac{1}{K} + \frac{D}{Ee}\right]}} \quad \text{m/s}$$

WHY CAN WE NOT USE THIS IN PRACTICAL ENGINEERING?

1. It does not consider valve closure time. It assumes valve closes instantaneously.
2. It does not consider line packing.
3. Several events cannot be considered.
4. It cannot be used for items other than pipes.





As can be seen from the graphs above, the flow continues to enter the pipe inlet even though the flowrate at the pipe outlet has reached zero. This clearly demonstrates line packing, which arises due to pipe elasticity and fluid compressibility.

WHAT EQUATIONS ARE USED IN PIPENET TRANSIENT MODULE FOR PIPES

Momentum Equation

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

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

Continuity Equation

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

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

where:

- p is the pressure in the pipe.
- u is the fluid velocity along the pipe.
- x is the distance along the pipe.
- t is the time.
- A is the cross-sectional area of the pipe.
- d is the diameter of the pipe.
- ρ the fluid density.
- α the angle the pipe makes to the horizontal.
- f is a friction factor.

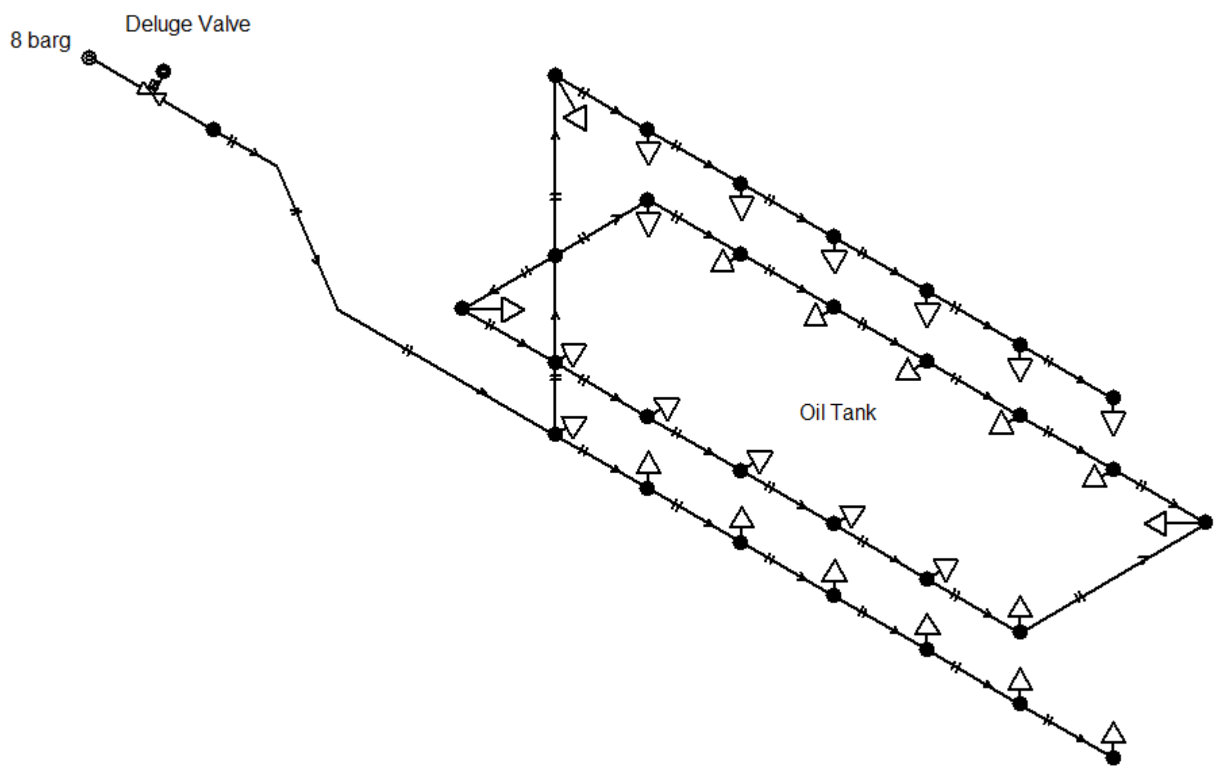
PIPENET Transient Module uses the Method of Characteristics to solve the above equations. For complete networks it solves large sets of partial differential equations, ordinary differential equations and algebraic equations.

How to Reduce Pressure Surge in Dry Deluge System

Pressure surge is a common phenomenon when a dry deluge system is primed. In the worst case the maximum pressure can reach high values when the remote nozzle is primed. This is because the flow resistance increases and the flow rate decreases instantly when the fluid in the nozzle changes from air to water. The following network has pressure surge problem based on the simulation results. In this document two cost-effective ways are discussed to solve the difficulties.

1 – using a short dry pipe near the most remote nozzle for creating a cushion effect
2 – decreasing pipe size to limit the flow rate when the network is primed.

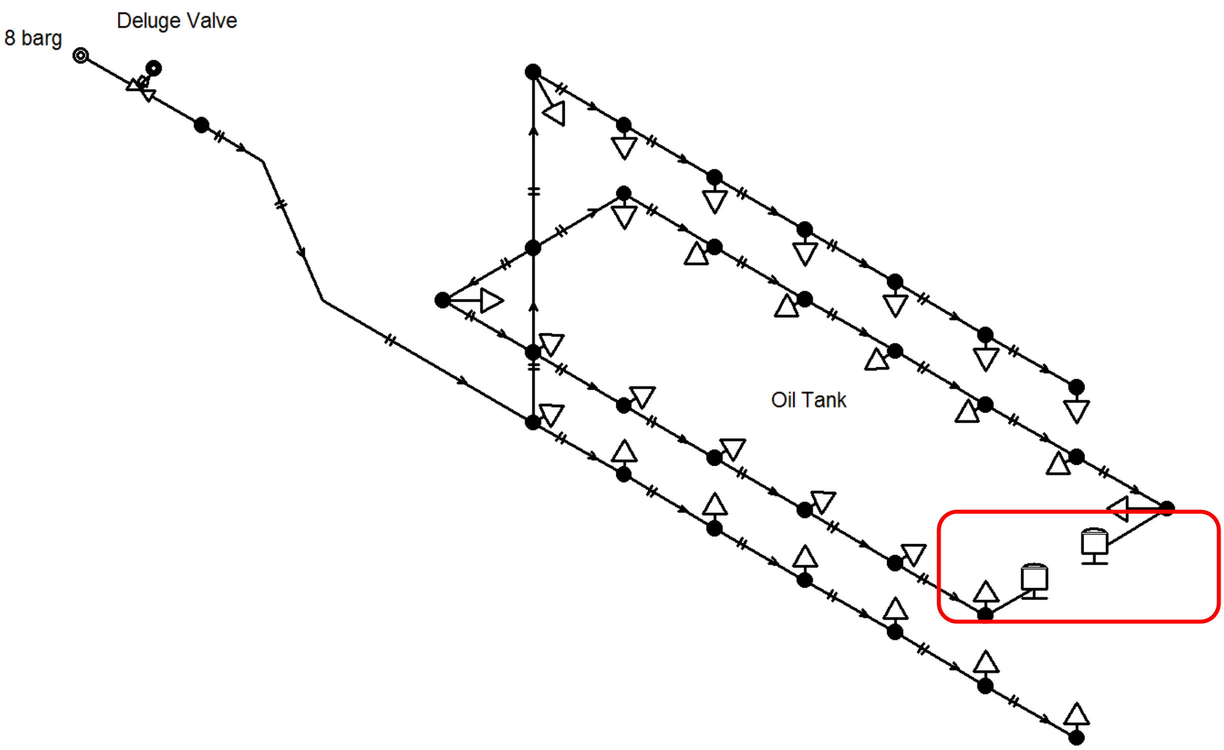
THE NETWORK



Inlet pressure:	8 barg (assume to be constant during simulation)
Deluge valve closure time:	5 seconds
Main pipe size:	Dn100
Riser size:	Dn80
Branch pipe size:	Dn50

BASE CASE

The air trapped in the pipe at the far end of the loop has the same effect as an accumulator when the remote nozzles are primed. Therefore, the above network can be simulated in PIPENET by modelling the mentioned pipe with the help of two accumulators. They are not real but 'virtual' accumulators representing the behaviour of the air trapped inside the pipe.

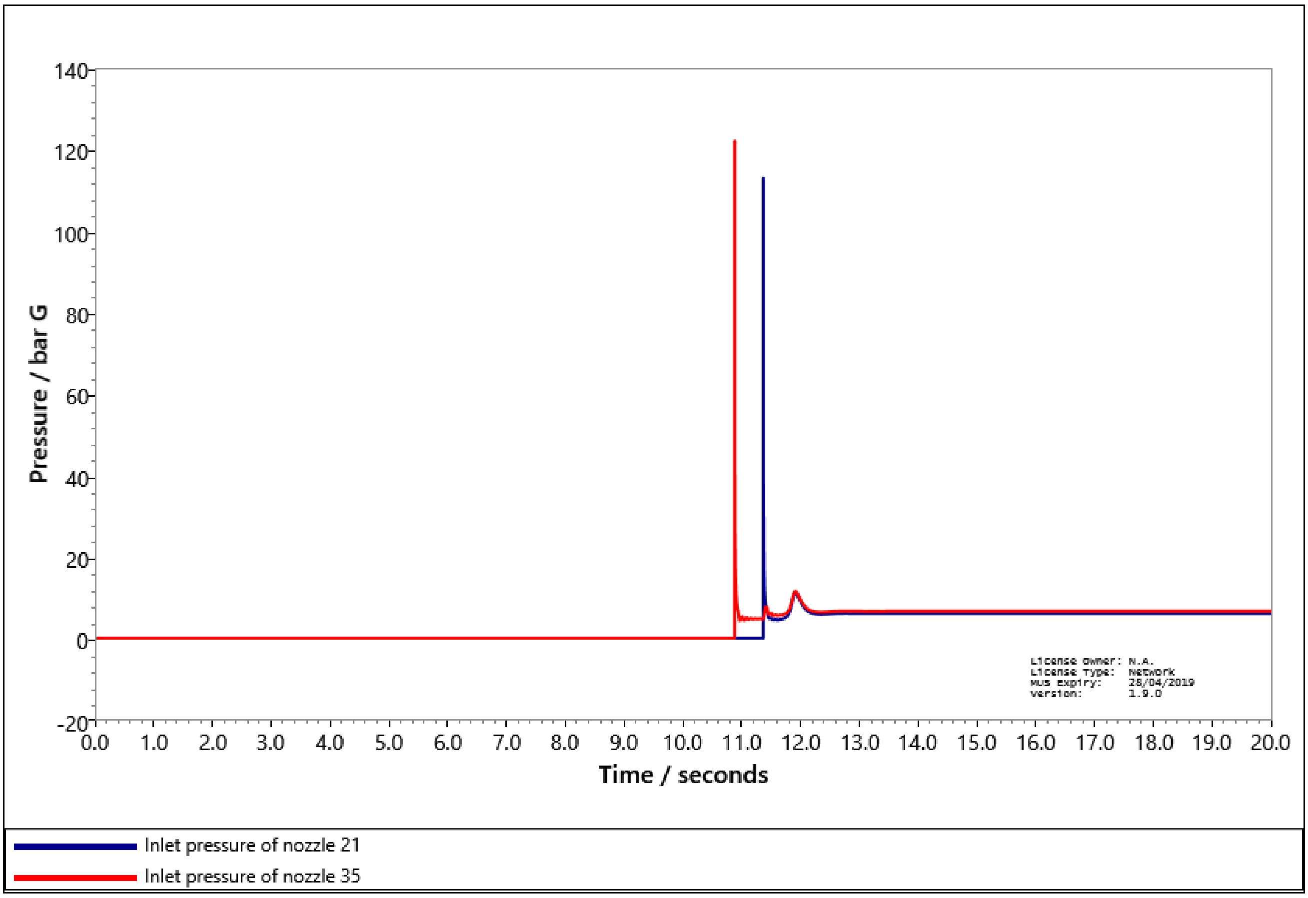


The dimensions of the virtual accumulators are the same as the modelled pipe. See the figure below.

Label	2	
Input node	210	
Type	Horizontal c...	
Diameter	50	mm
Length	2	m
Reference Fluid D...	1	mm
Reference gas	Diatomic gas	
Heat capacity ratio	1.4	
Reference Gas Te...	21	°C
Reference Gas Pr...	0	Bar G
Results selected?	NO	

The calculated maximum pressure occurs at the remote nozzle of the lower and upper branches when the nozzles are primed. See the summary and figure below:

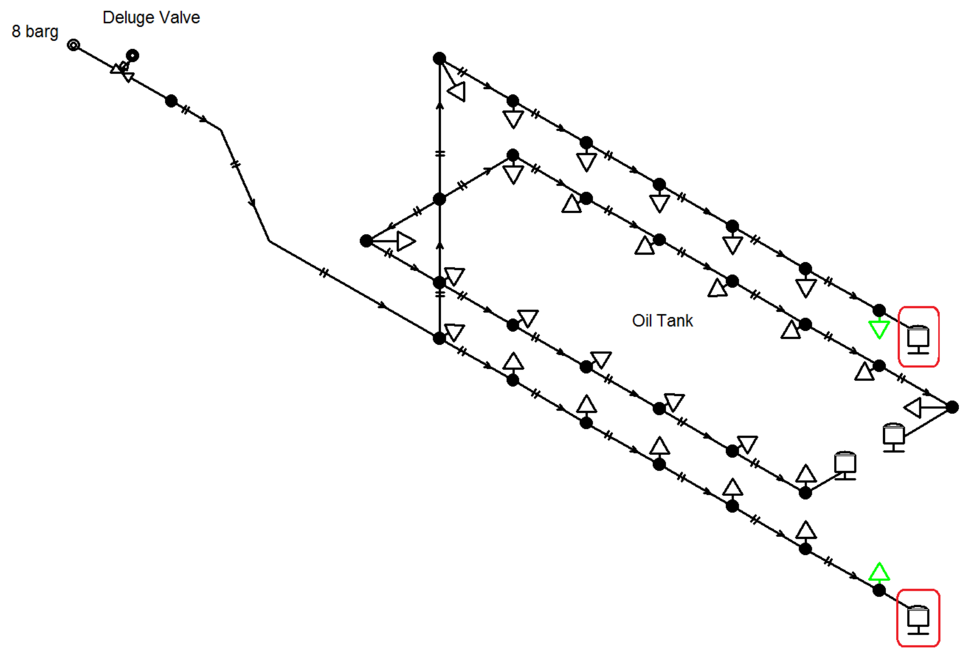
Maximum pressure is 122.052 bar G
on pipe 163 at the outlet
at time 10.88200 seconds



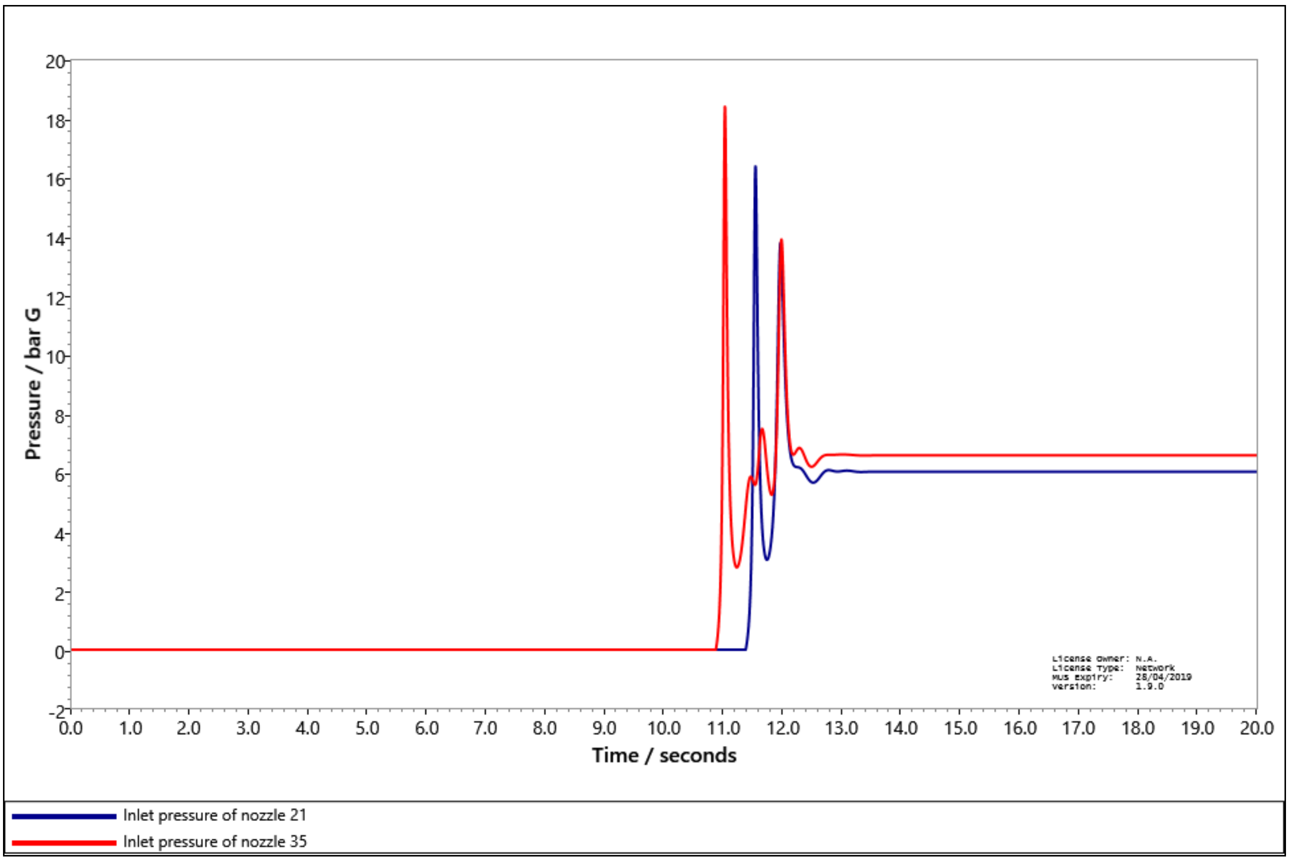
Solution 1 – extend pipes near the remote nozzle

This is the same as the base model, except that the lower and upper branches are extended 1 m for creating the cushion effect when the remote nozzles are primed. These pipe extensions are modelled in PIPENET as accumulators for convenience, although they do not represent real accumulators. They work as cushions to absorb the pressure surge from the nozzle priming.

Label	3	
Input node	7	
Type	Horizontal c...	
Diameter	50	mm
Length	1	m
Reference Fluid D...	1	mm
Reference gas	Diatomic gas	
Heat capacity ratio	1.4	
Reference Gas Te...	21	°C
Reference Gas Pr...	0	Bar G
Results selected?	NO	



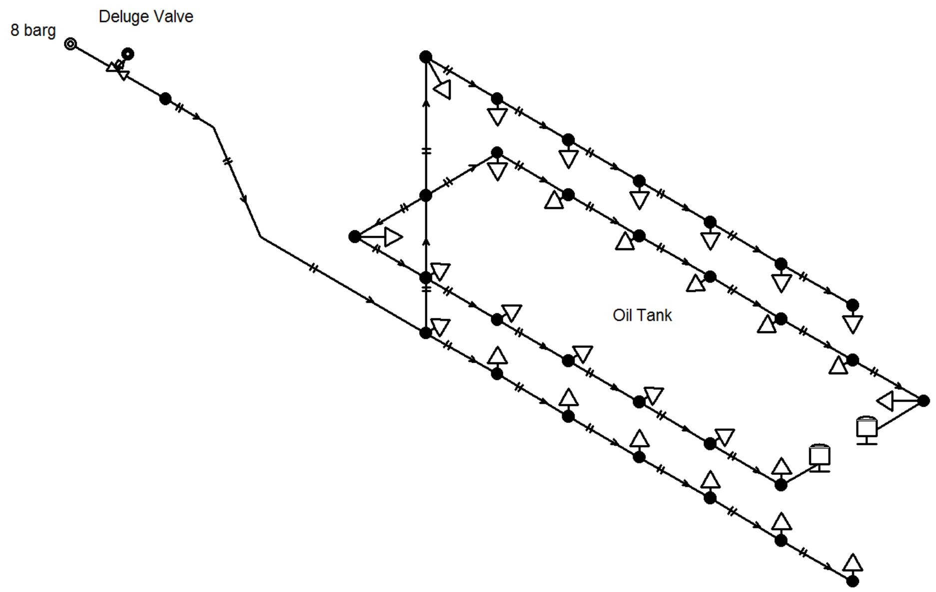
The maximum pressure drops to 18.4 barg. See the figure below:



Solution 2 – use smaller pipes to reduce the flow rate when the system is primed

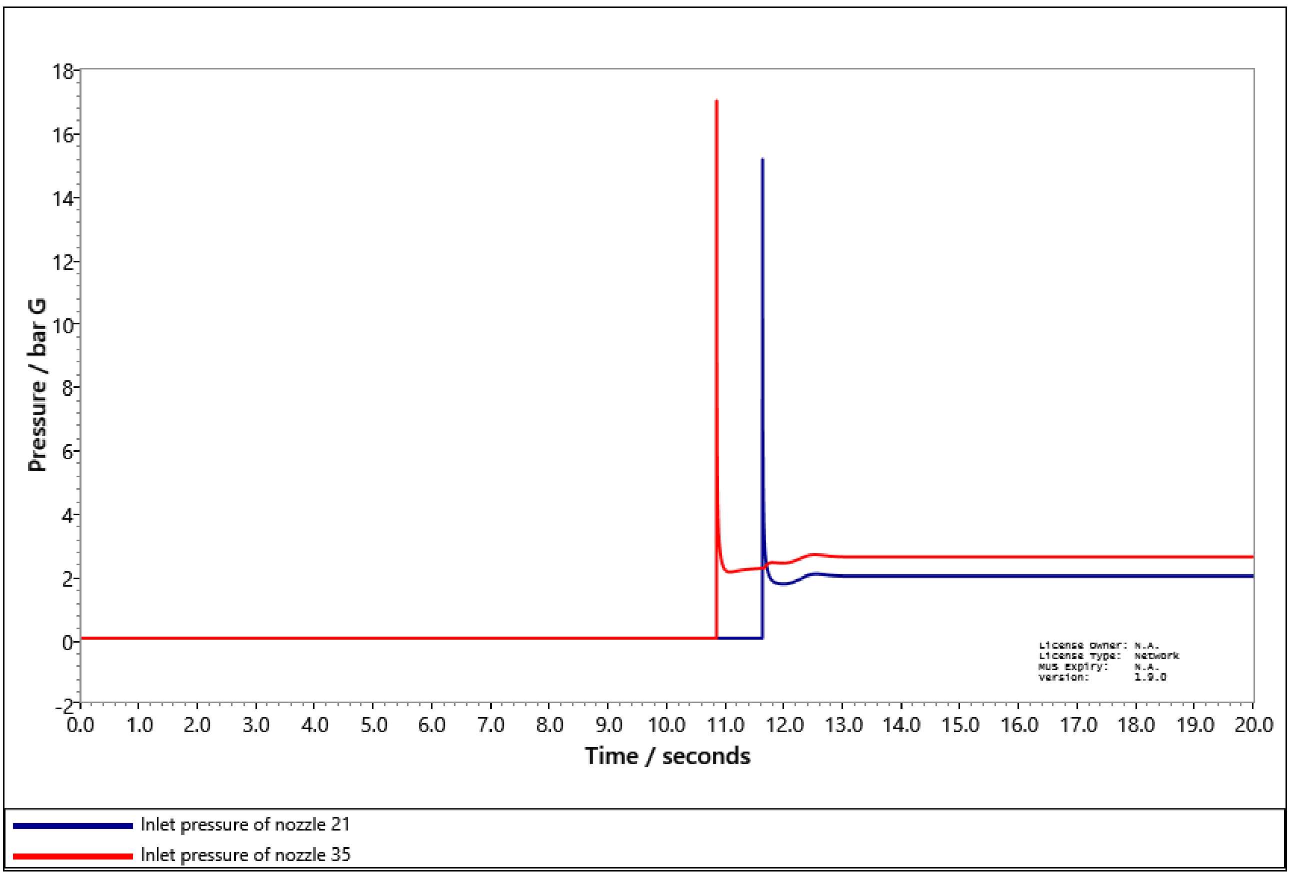
This is the same as the base model, except that the pipe size is decreased in order to create an additional pressure drop. As a result, the pressure loss of the pipes increases significantly and the flow rate when the system is primed decreases. This significantly reduces the pressure surge. However, the flow velocity in the pipes and the pressure at the nozzle inlet must be checked carefully to satisfy the project requirements. In this scenario, the pipe sizes are changed as follows:

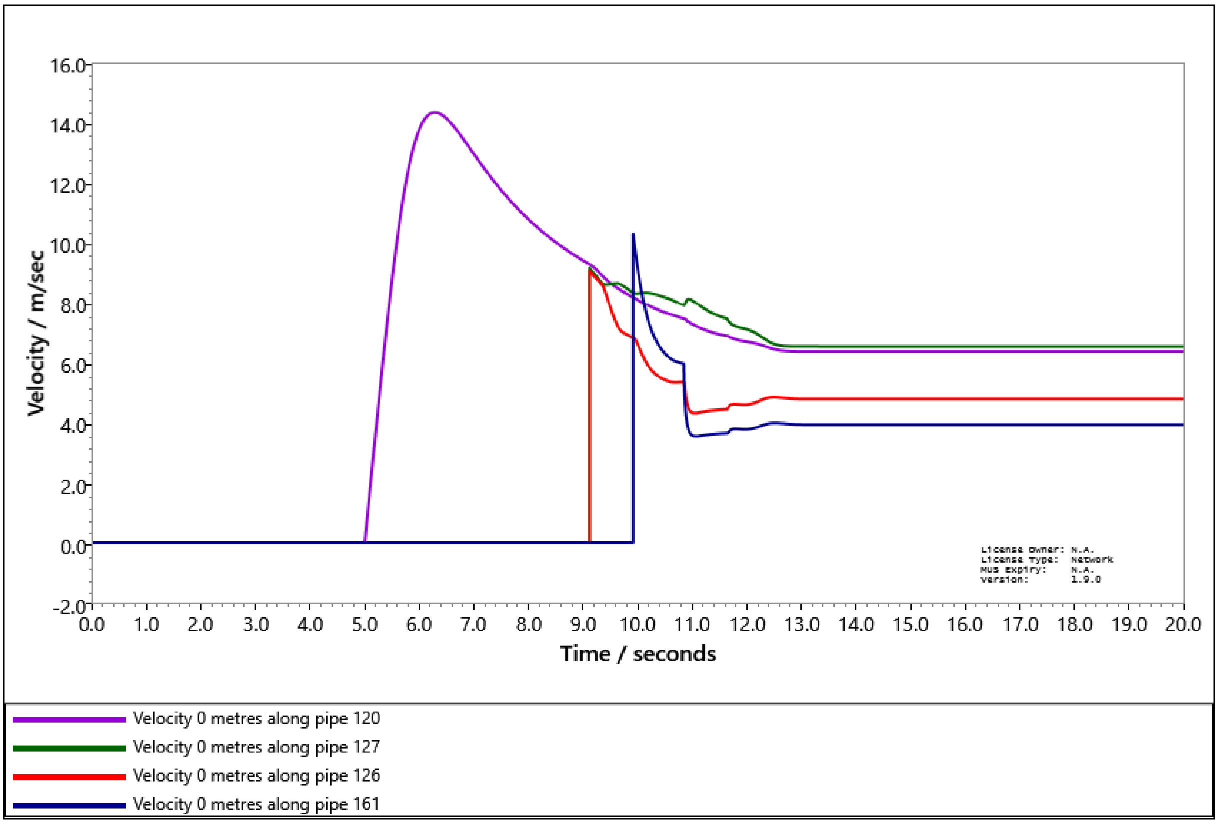
Main pipe size: Dn65
Riser size: Dn50
Branch pipe size: Dn32/Dn25



The maximum pressure drops to 17.0 barg in this scenario. The minimum pressure at the nozzle inlet is 1.95 barg and the maximum flow velocities in the pipes are between 3.9 to 6.5 m/s after the system is primed. See the summary and figures below:

Maximum pressure is 16.9769 bar G
on pipe 163 at the outlet
at time 10.84400 seconds





CONCLUSION

Dry deluge system and the caused problems are discussed in this article. Both pipe extension near the most remote nozzle and smaller pipe are the effective methods to reduce the pressure surge when the system is primed. The size of pipe extensions and pipes can also be optimized based on the simulation results.

Modelling cavitation in PIPENET

There are three options in PIPENET for modelling cavitation:

No cavitation
Vapour cavitation
Channel cavitation

No cavitation: with this option the calculation will be faster. No cavitation model can allow the pressure to go below the vapour pressure or even the absolute pressure.

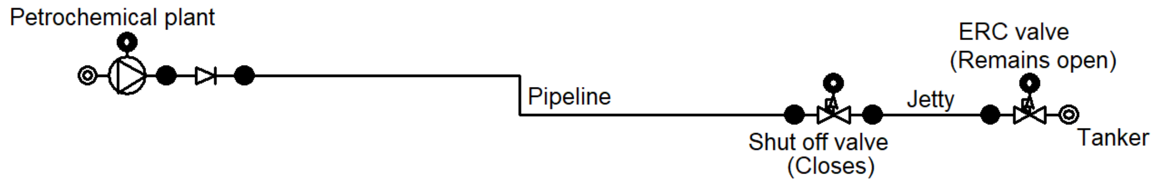
Vapour cavitation: this model is suitable for relatively small cavities where the volume occupied by the cavity does not affect the behaviour of the system.

Channel cavitation: this model considers the physical size of the cavity. This is suitable for bigger cavities and where the size of the cavity itself can affect the results significantly.

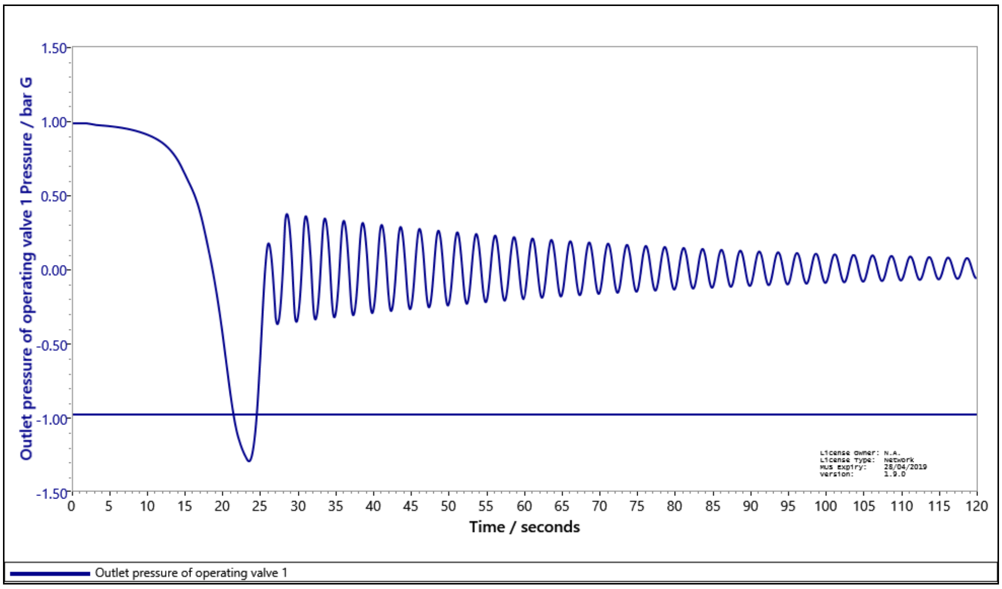
The choice of the cavitation model strongly depends on the system. We recommend the user to start with the Vapour cavitation option. If there is no vapour cavitation in the system, the results for Vapour cavitation model and No cavitation model will be the same. The only difference will be the running time: Vapour cavitation model will take longer than No cavitation model. If the system has vapour cavitation, check the cavitation volume. If the cavity size is relatively large or growing indefinitely, it is better to use the Channel cavitation model.

In order to illustrate the difference between No cavitation, Vapour cavitation and Channel cavitation models, let us consider the following examples.

No cavitation vs Vapour cavitation

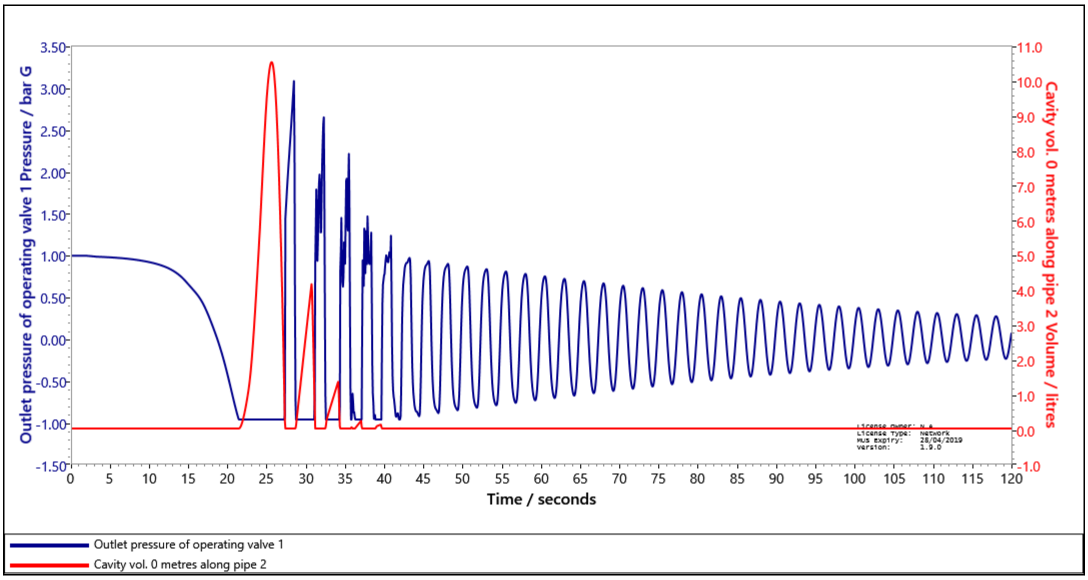


First, we assume that the jetty is completely horizontal. With No cavitation model, the results look as follows.



Pressure goes below the vapour pressure of -0.977 barg. This is an unrealistic situation, which suggests that there is cavitation in the system, and so the No cavitation option is not suitable.

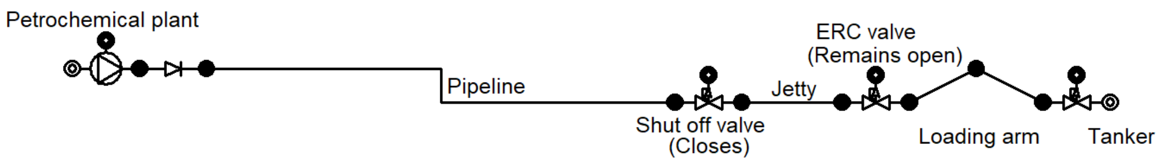
The same model run with Vapour cavitation option will produce the following results:



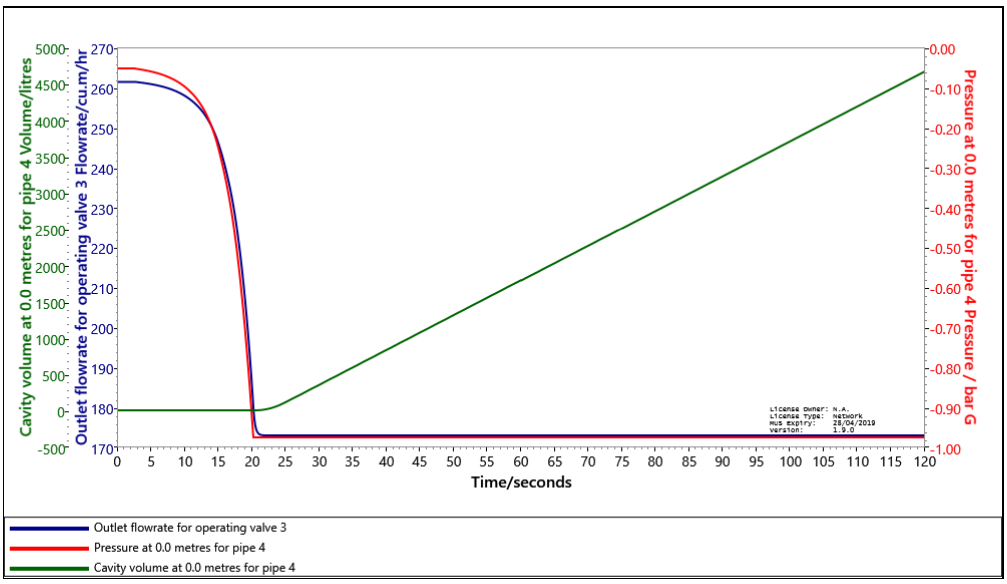
In this case the pressure does not go below the vapour pressure of -0.977 and the cavity volume can be traced. The cavity volume increases up to a maximum 10 litres. This is much smaller than the liquid volume in the pipe. Therefore, the vapour cavitation model is appropriate in this case.

Vapour cavitation vs Channel cavitation

Now, let us consider another case – with a loading arm at the end of the jetty. The loading arm has a 20 m elevation.

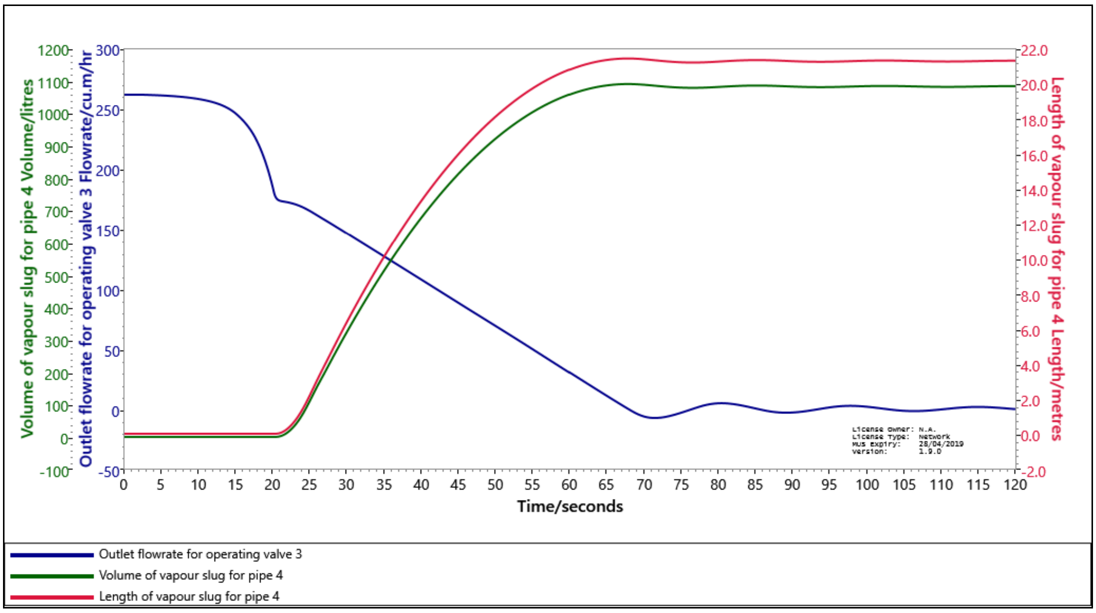


With the Vapour cavitation option, the graphs look as below:



The results suggest that the cavity size is growing indefinitely. Furthermore, the flowrate through valve 3 does not go to 0. That is unrealistic. This shows the inadequacy of using the vapour cavitation model under certain circumstances. So, it is necessary to use the channel cavitation model in this case.

The results of Channel cavitation model are shown below.



It can be seen that the cavity length reaches around 22 m and stays there. The flowrate through valve 3 goes to 0. So, we can conclude that the channel cavitation model predicts the cavity formation phenomenon well.

AUTOLAYOUT: Step-by-Step Guide

PIPENET Spray/Sprinkler Module includes the Autolayout feature that can be used to generate sprinkler systems for vessels. This document provides a brief guidance on how to use the Autolayout tool.

Before using the Autolayout tool

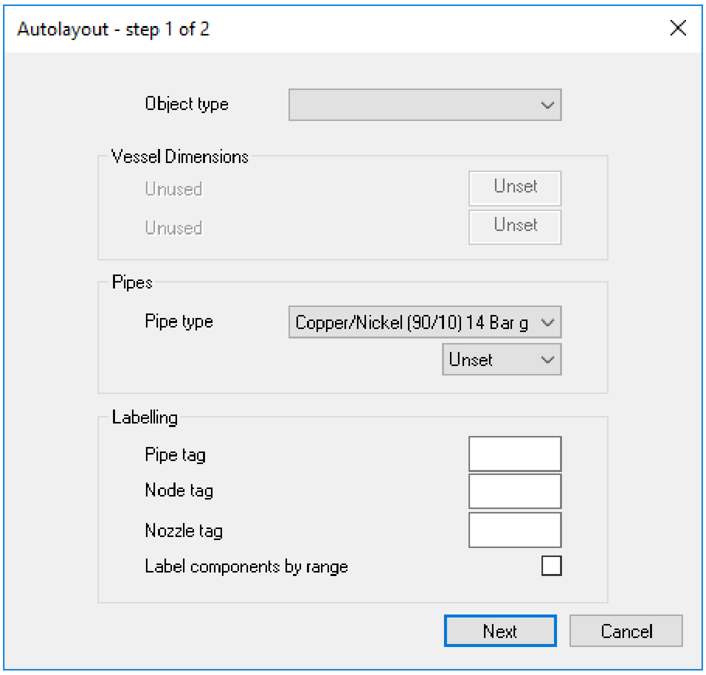
Before using the tool make sure that you have defined the nozzles you intend to use as library items and that you have provided, for each nozzle type, the spray divergence angle. Also ensure that you have at least one pipe type defined.

Launching the tool

Select Autolayout from the Tools menu bar item.

First dialog - defining the vessel and pipes

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



- Object type

  This defines the type of object that requires protection. The options are:

Tanks	Area Protection
Vertical Cylinder	Rectangle
Horizontal Cylinder	
Sphere	

- Vessel dimensions

  Specifies the dimensions of the object.

- Pipes

  This section defines the type and diameter of pipe you wish to use in the Autolayout network.

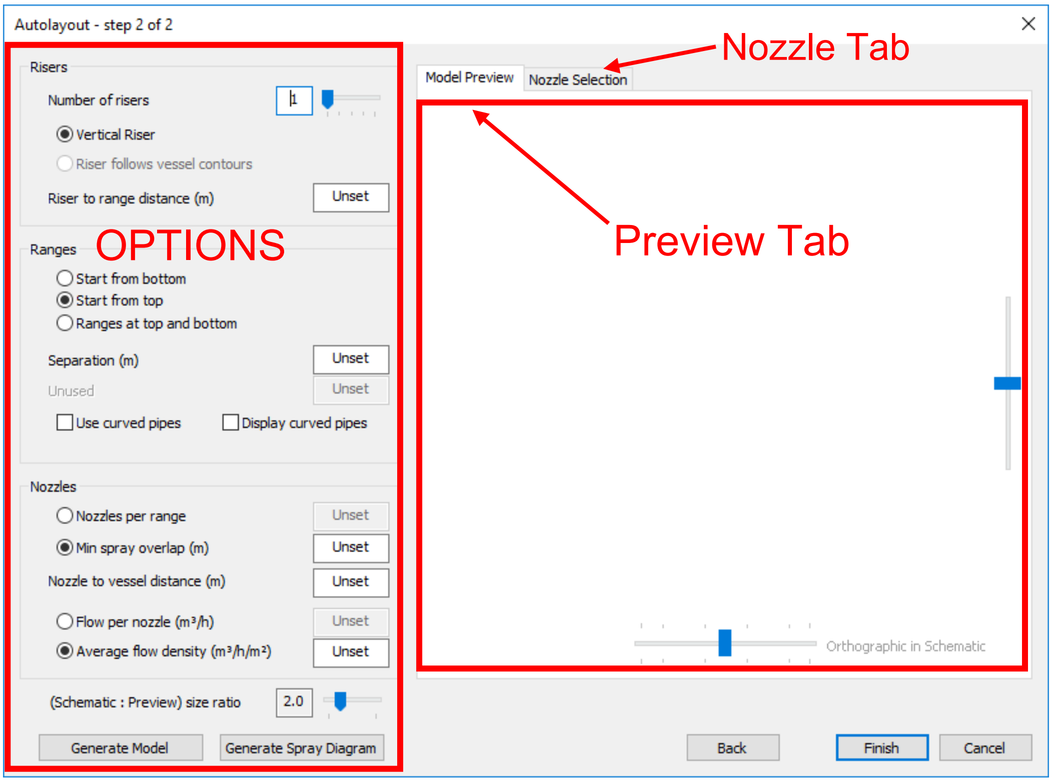
- Labelling

  If you want Autolayout to automatically label the generated components, you can define the tags to be used in this section. If the *Label components by range* check-box is checked, the labels of the generated components will be numbered according to the ranges with which they are associated, based on the specified tags. For example, if the nozzle tag is "NOZZ" and this option is enabled, the third nozzle on the second range will be labelled "NOZZ2/3".

Click the *Next* button to proceed to the second dialog.

Second dialog - network specification

The second dialog is where the network is specified, and depends upon the type of object being protected.



- Options

  This area displays the options for the vessel/area. The contents of this section depend on the selected object type.

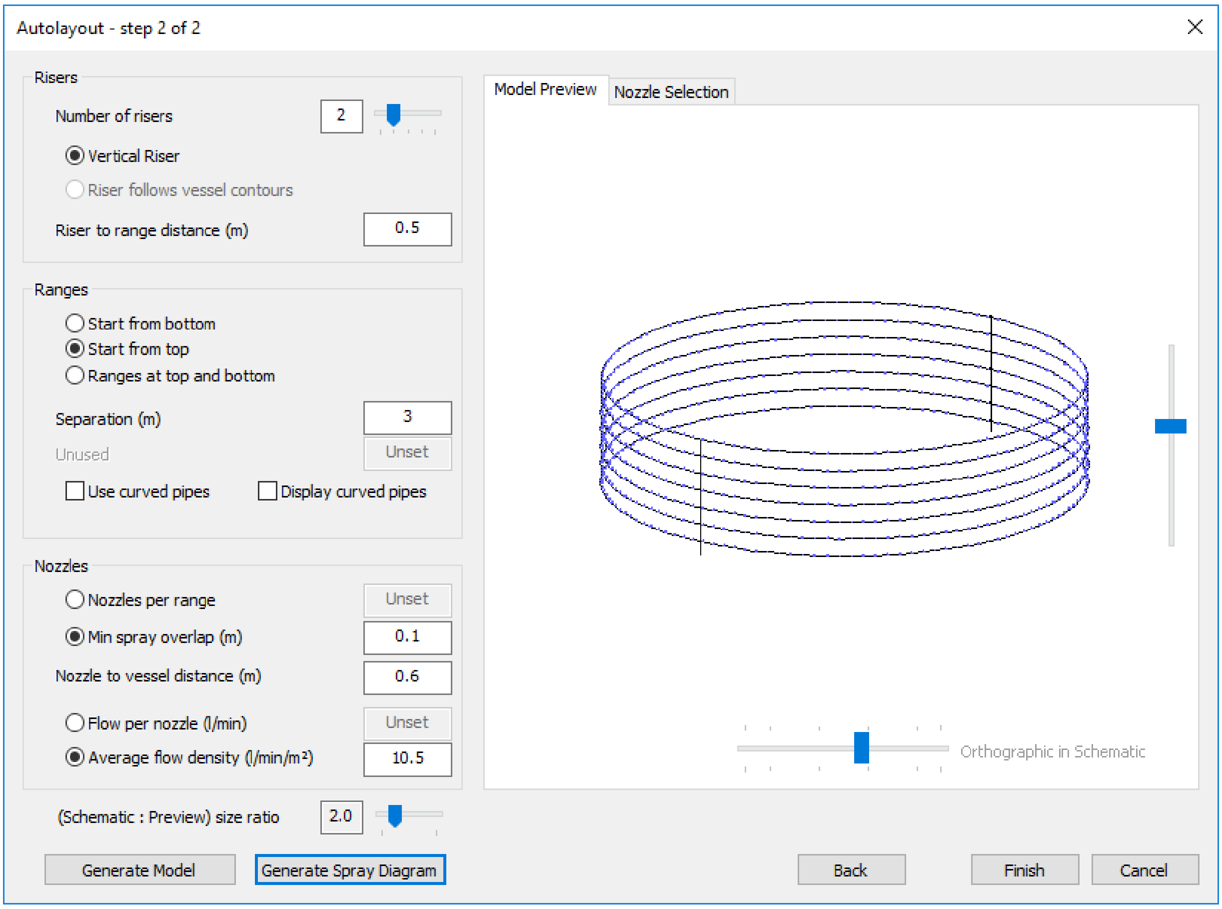
- Model Preview Tab

  This tab shows a 3D view of the generated model. The model can be viewed from different angles of rotation and elevation by moving the horizontal and vertical sliders respectively. In addition, rotation in the preview tab defines the rotation of the model when placed in the schematic. The notification to the right of the horizontal slider indicates when the model would appear orthographic in the schematic. If there is an error in generating a model, or the model is not completely specified then no model is displayed.

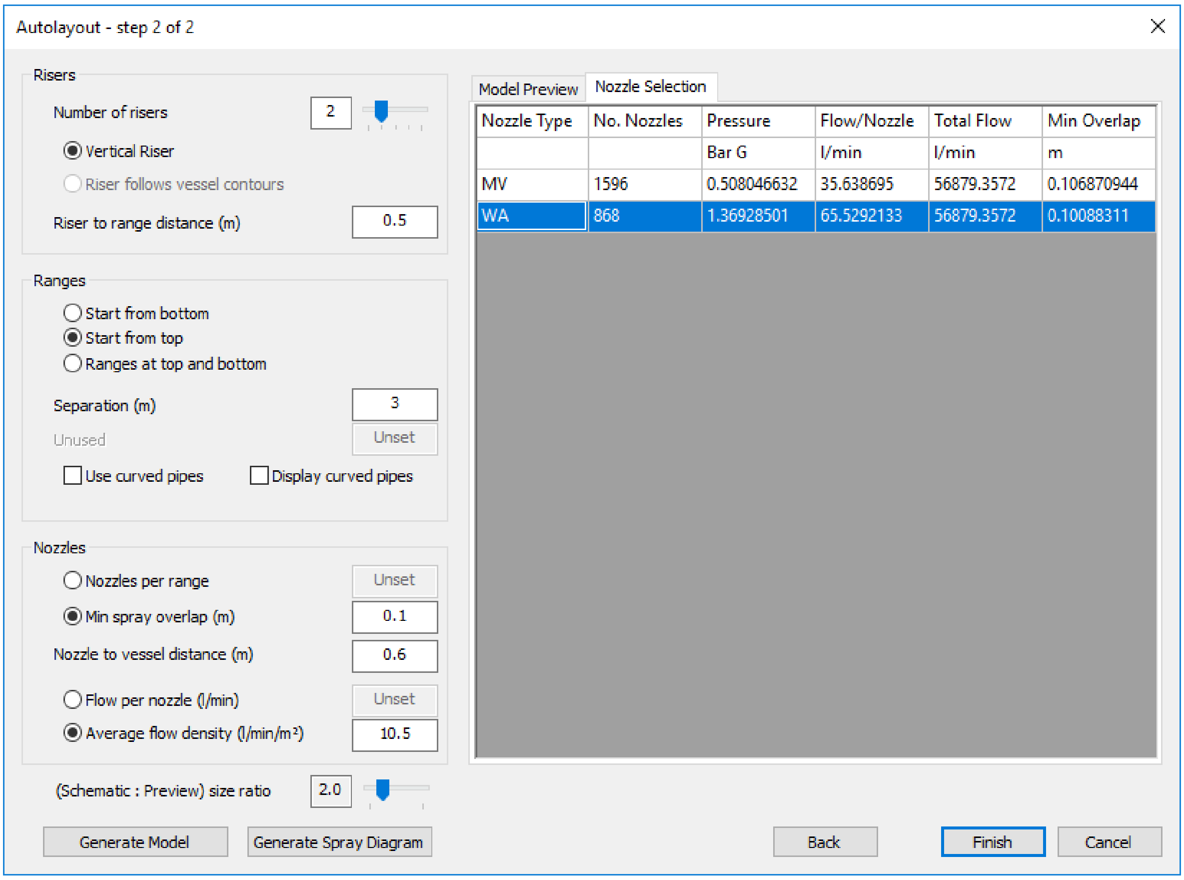
- Nozzle Tab

  This tab shows a table of all nozzles in the nozzle library, along with information about the model that would be produced if each nozzle were chosen. This is where nozzles are selected to be used in the model.

In order to generate a model, enter valid data as described in the options pages for the chosen object type. In the nozzles tab select the nozzle you wish to use. If a valid nozzle has been chosen then a model will now display in the preview tab. Changing any of the options will automatically update the model/nozzle table depending on the currently selected tab. To manually force an update, click the *Generate model* button.



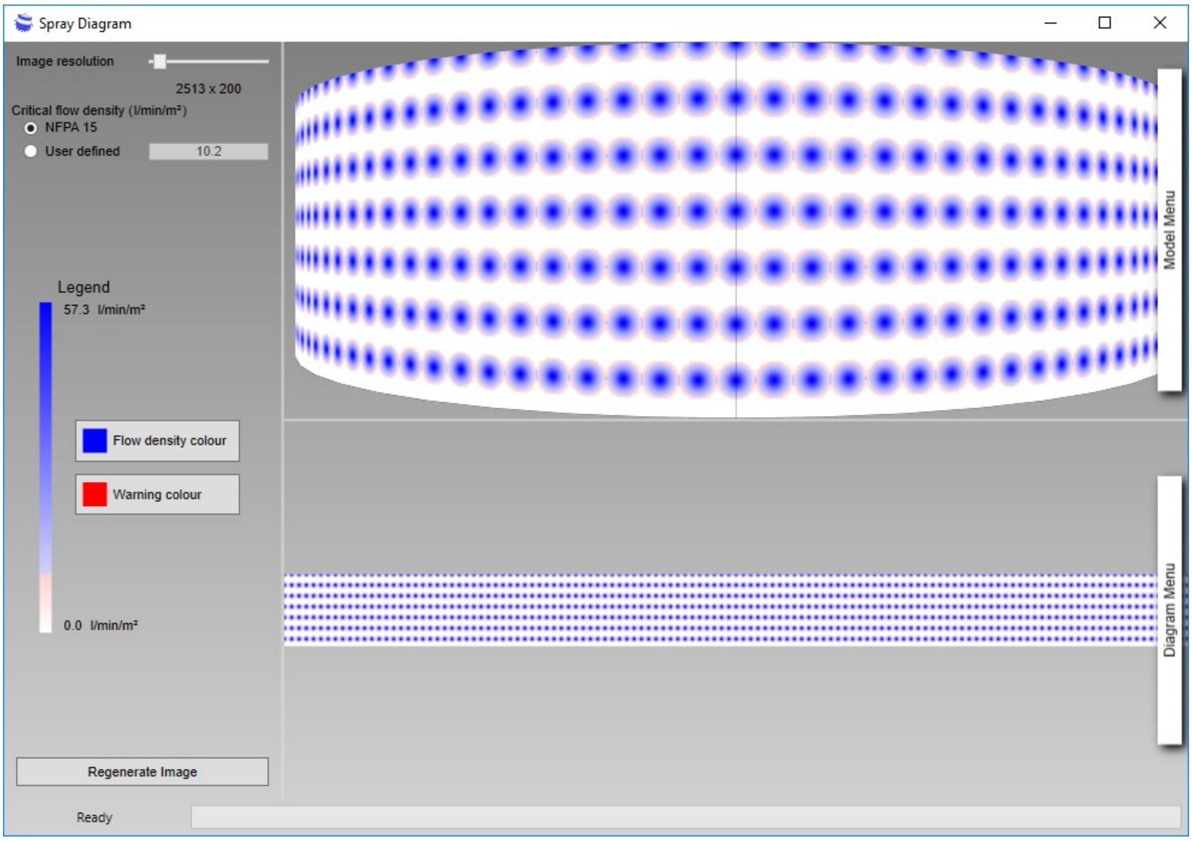
Once the model has been generated, you can check the corresponding calculated nozzle data in the Nozzle tab:



Spray Diagrams

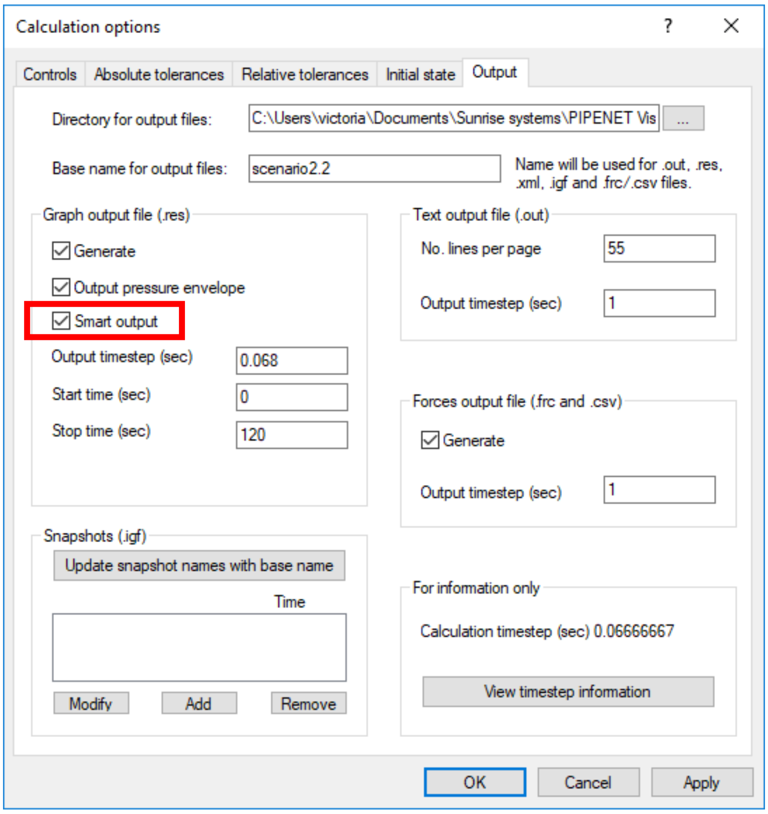
The *Generate Spray Diagram* button launches the Spray Diagrams tool, which can be used to review the effectiveness of the model in protecting the target.

The spray diagram can be copied and pasted into the report.



Smart Output: Why it is Important to Choose This Option

PIPENET Transient module users may sometimes notice discrepancies between the results shown in data and properties windows or generated reports on the one hand, and the plotted graphs on the other hand. Such discrepancies most probably arise due to mismatch between the calculation timestep and the output timestep. This issue can be easily solved by making sure that the Smart Output option is checked.



PIPENET allows the user to select the calculation timestep and the output timestep. Naturally, the output timestep must be equal or larger than the calculation timestep. When the output timestep is larger than the calculated time step, the graph results may miss the peak values if they are located between output time interval. For example, the calculation time step is 0.01 sec and the graph output timestep is 0.05 sec. If a peak occurs at 24.53 sec, the graph only shows the results at 24.50 sec and 24.55 sec.
However, if the Smart Output option is selected, the graph will still “capture” all peak results even if the output timestep is larger than the calculation timestep.

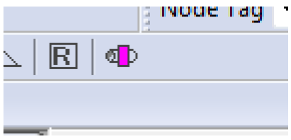
Therefore, users are recommended to select the Smart Output option if the output timestep is larger than the calculation timestep, to avoid discrepancies in the results.

How to…?

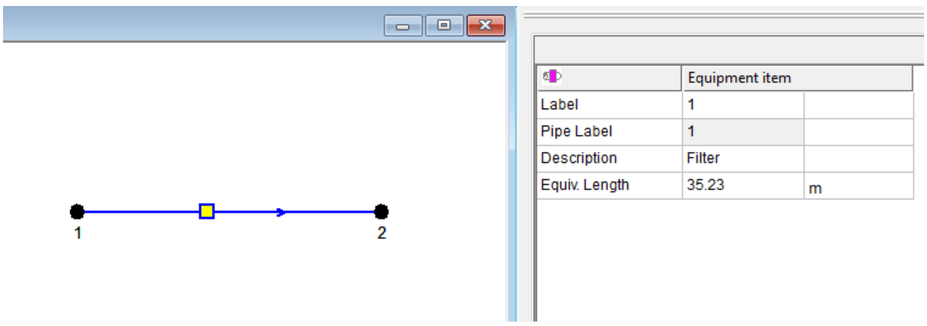
How to model pipe expanders and pipe reducers

Modelling pipe expanders or reducers in PIPENET depends on the particular module in which the network is built. The following rules should be kept in mind.

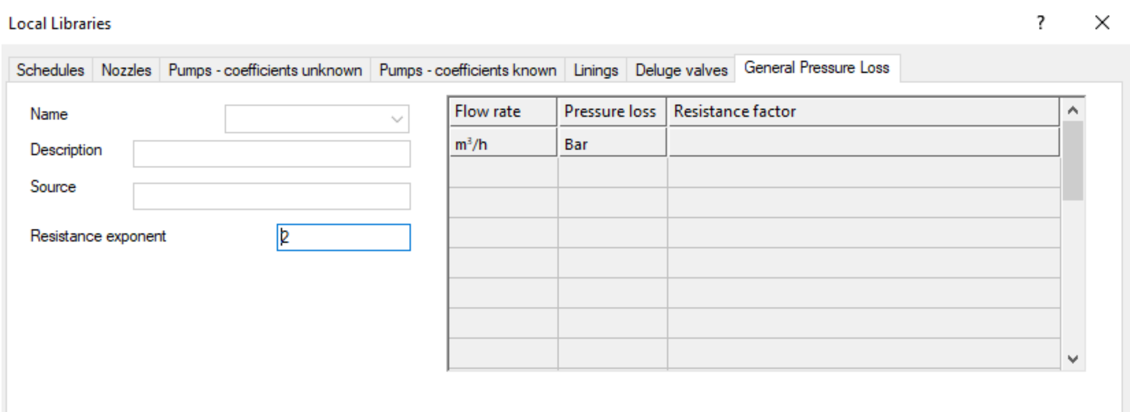
1. The Spray option in the Transient module and the Spray/Sprinkler module strictly comply with the NFPA rules. Reducers and expanders are absent from the data given in the NFPA rules. Therefore, they are not available as built-in fittings. However, they can be input as 'equipment items' if the equivalent length is known, or as a 'general pressure loss component' if the pressure loss and flowrate are known. Some relevant screen shots are shown below:

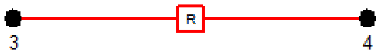


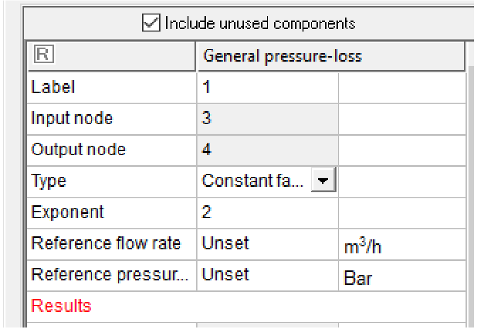
Properties window for equipment item:



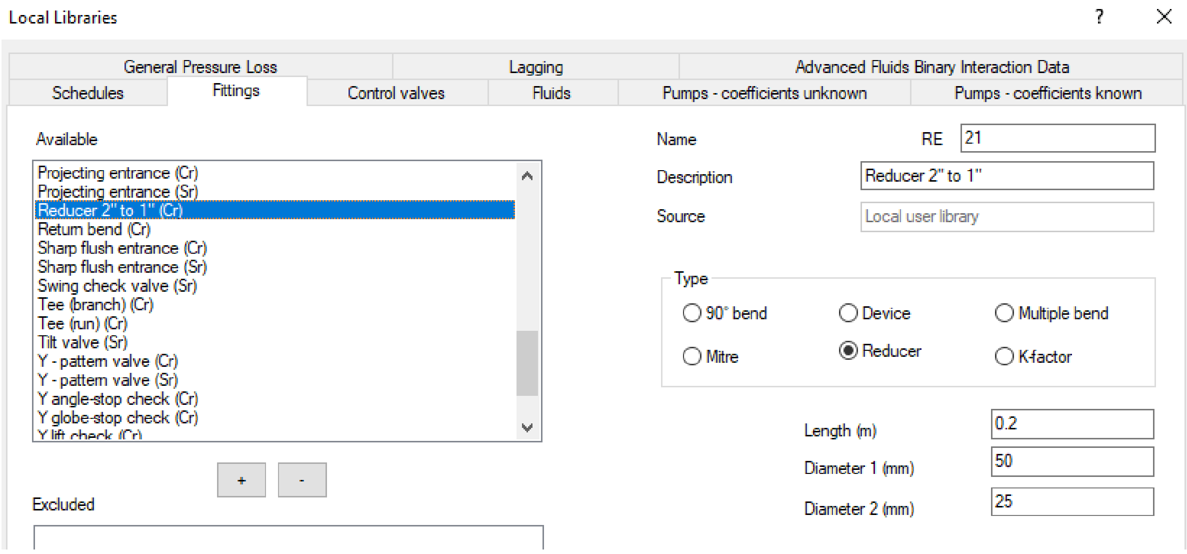
General pressure loss component can be set up in the library as a table or directly in the properties window:

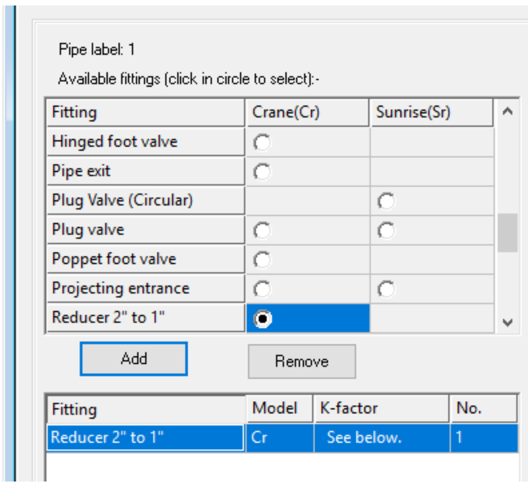






2. The Standard option in the Transient module and the Standard module use the Crane manual data. The user can add their own data to this. An example of adding a reducer is given below.

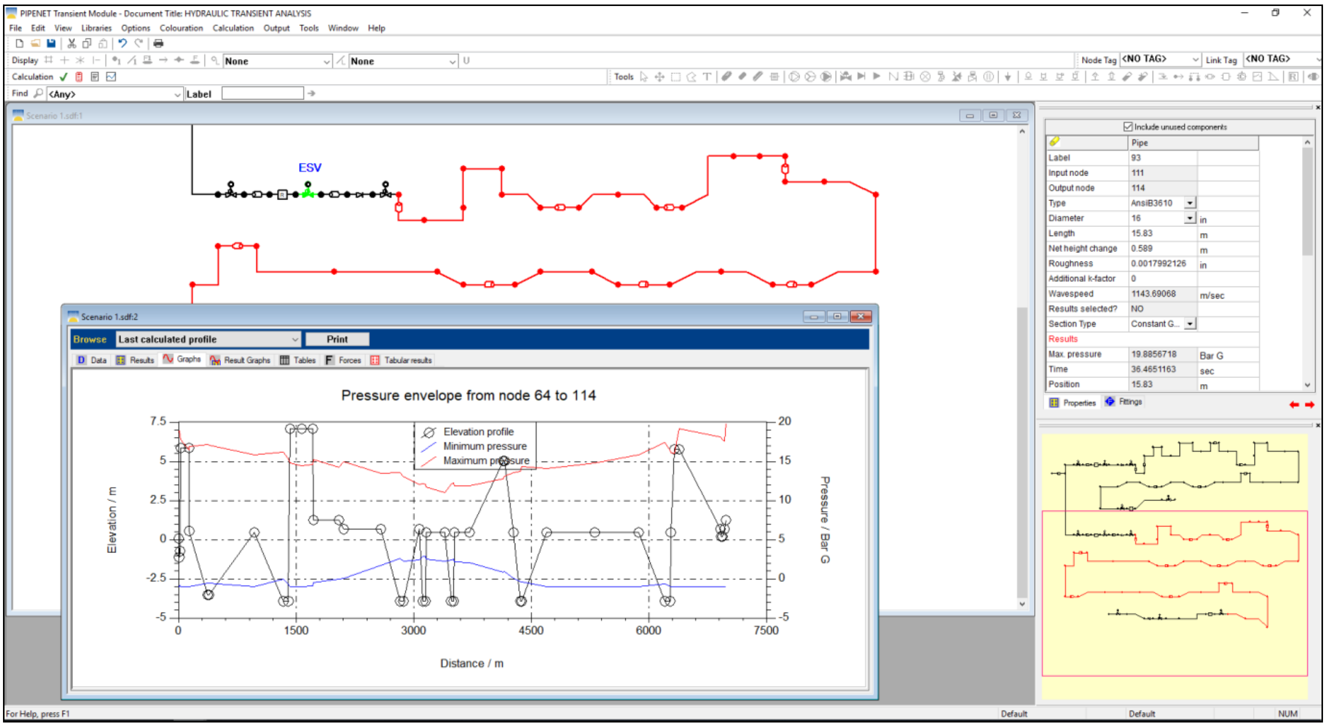


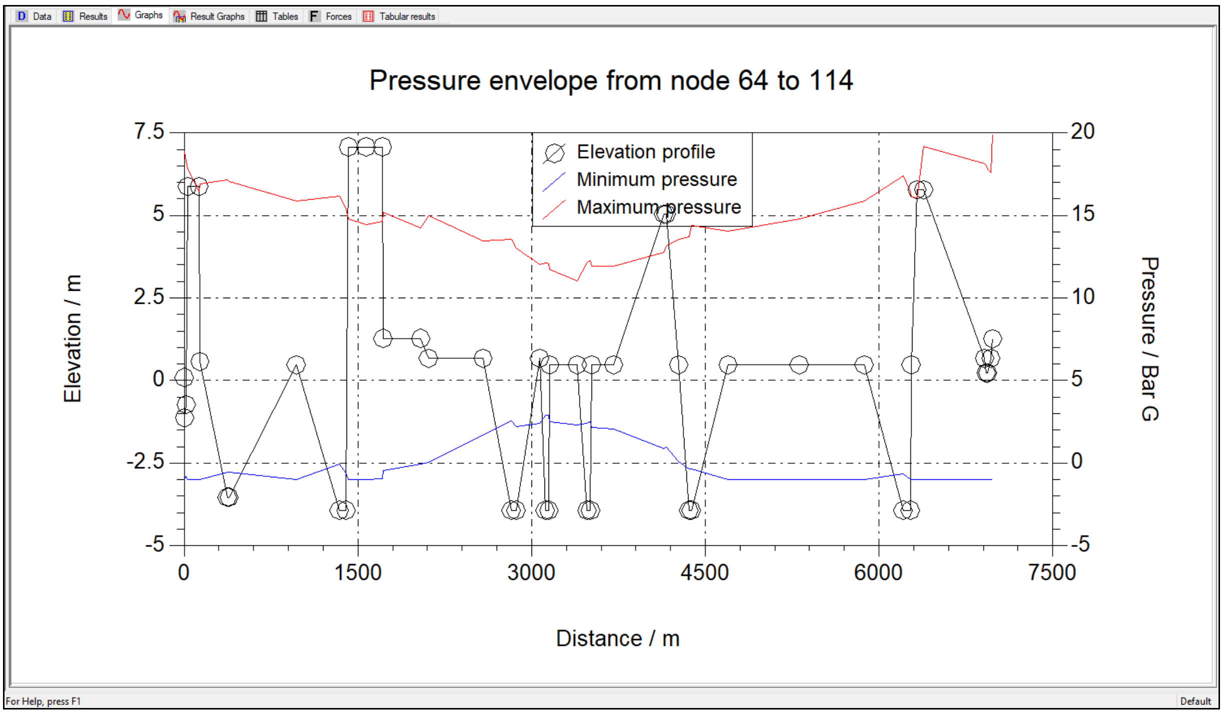


How to model pressure envelope for selected pipes

In order to model a pressure envelope, the user has to make a path between two nodes first. The following algorithm would be useful to remember.

1) Select a node
2) Hold the “Ctrl” button and select the other node
3) Go to Tools - Make a path. Then the path will be highlighted in red.
4) Results can be seen in the Data window > Graphs.





You can also export the graph data into an Excel spreadsheet:

1) First make a path.
2) Then, go to Tools, Click “Export envelope CSV for path”