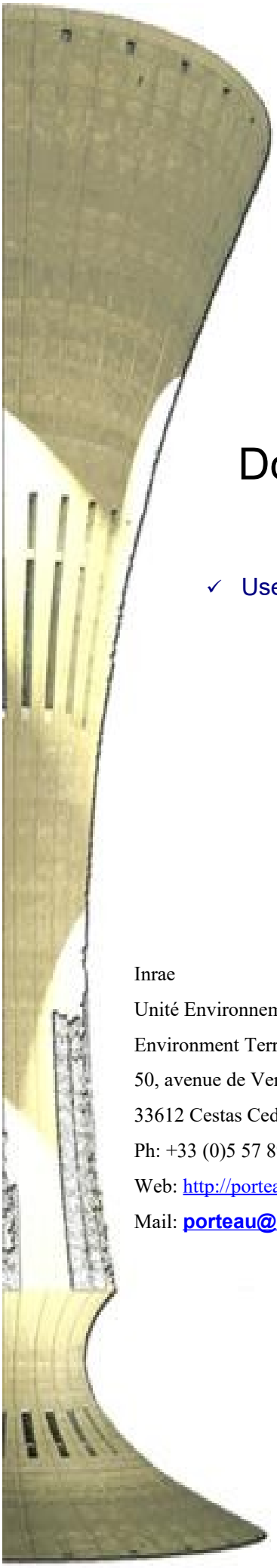


Documentation Porteau 4

✓ User Guide

Inrae
Unité Environnement Territoires en Transition Infrastructures Sociétés
Environment Territories in Transition Infrastructures Societies
50, avenue de Verdun
33612 Cestas Cedex
Ph: +33 (0)5 57 89 08 00 – Fax: 05 57 89 08 01
Web: http://porteau.irstea.fr
Mail: **porteau@inrae.fr**

Updated February 17th 2022

Summary

Graphic Interface

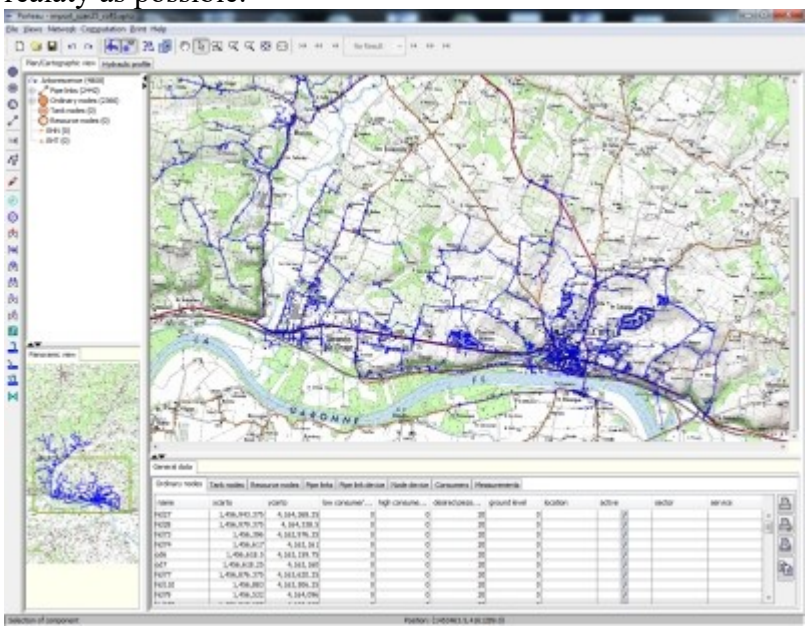
Some screenshots of this document are not updated when Porteau is updated.

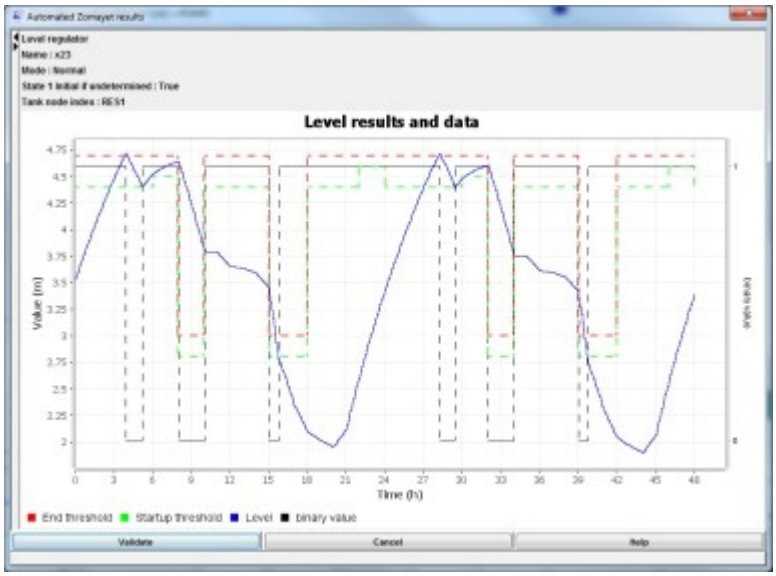
Presentation

Porteau is a tool to **model the behaviour of looped main networks distribution or transporting water under pressure**. It provides a decision-making tool for designing and managing drinking water supply networks.

Graphic interface

The graphic interface is easy tu use. It can draw the network studied with pipes et nodes for cross section. These elements are documented to represent all equipments and all hydraulics conditions to be like the realaty as possible.

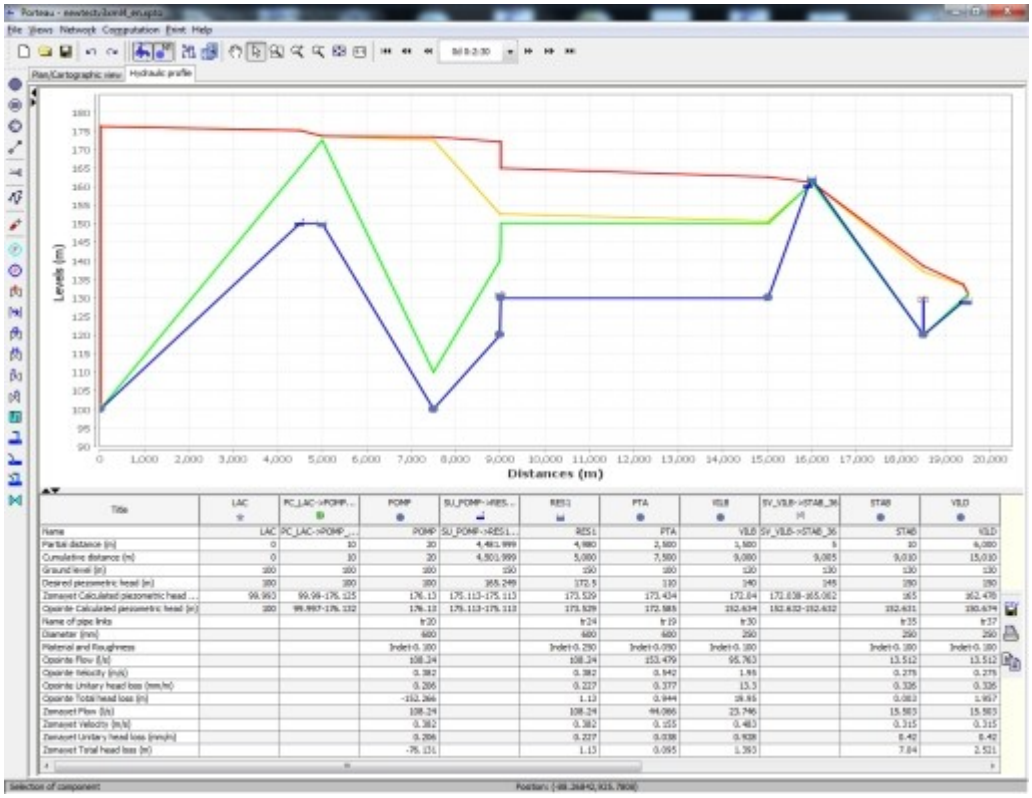




The main features are:

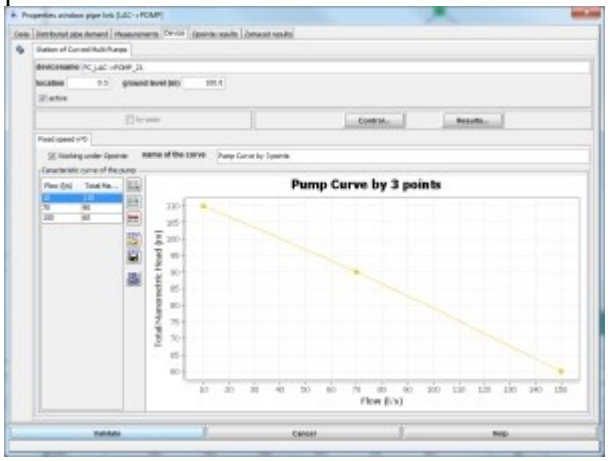
- network visualization of water distribution in basemap,
- viewing plans of distribution networks of drinking water,

- plotter printing sizes up to A0,
- viewing and printing the results in tabular form or on the network plan,
- viewing and printing results as hydraulic profiles,
- export data and results to a spreadsheet,
- export graphics in vector format results.

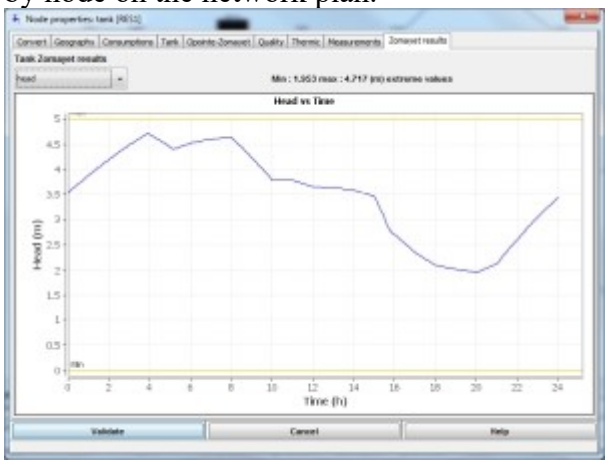


Module Zomayet

It produces a simulation over several hours (from 24 hours to 15 days) to study the hydraulic operation of a looped main network distributing or transporting water under pressure. It provides a view of the corresponding network plan.
The network may include tanks (with a choice of several filling/emptying modes), pumps, Control Valves Flow, Pressure Reducing Valve, motorised valves and pressure regulators, as well as comnsumption determined by models providing a breakdown of data over the course of the day or more.
The data required includes the complete topography of the network (length, diameter and roughness of the pipes, position in the relation to the datum level of fixed flow nodes, water level, invert and overflow, surface area at the invert and overflow for fixed-load nodes) as well as the most precise distribution possible of the consumers between the different nodes or along the pipes.

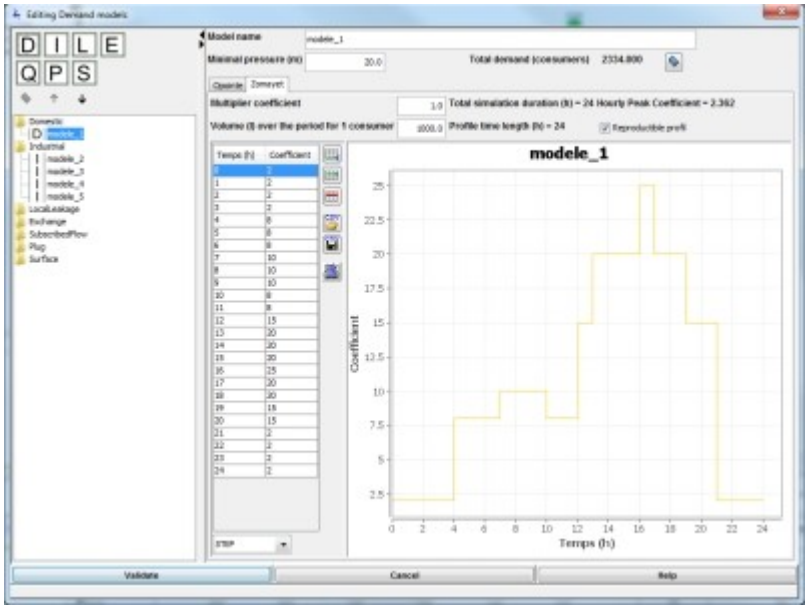


A deterministic model is used to process the data. The calculation results give the variations in the different values for each node and pipe over the day (or more): water level in the tanks, incoming and outgoing volumes, lead level of the consumption points, flows in the pipes, hours of operation of the pumps and point at which they start up and stop, working of the various components in the network etc. The results can be displayed either in the form of a table at each of the time steps (1 s to 1 hour) for the whole network or in the form curves of the different values over the course of the simulation by pipe and by node on the network plan.



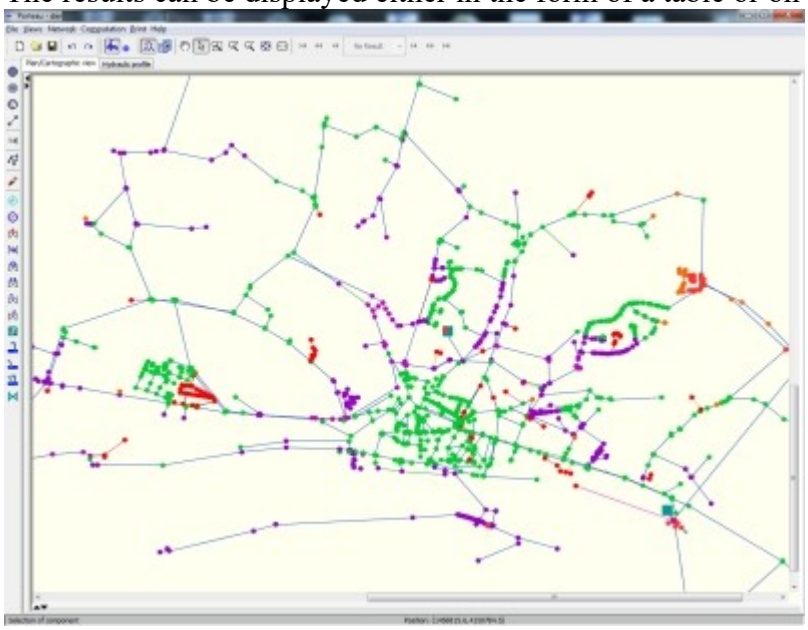
Module Opointe

It provides to simulate the working of a looped drinking water distribution network in a peak operating mode and to view results on the diagram.
The data used is the physical data concerning the nodes and pipes (position, water level, diameters, lengths), hydraulic data (roughness, household or industrial consumptions) and the data concerning the distribution of the subscribers. The probabilities of opening and satisfaction of the subscribers on the network must also be known.



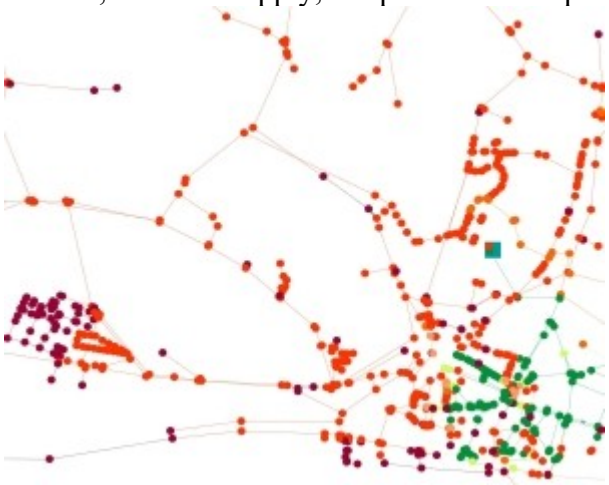
A probabilistic model is used to process the data and estimate the peak flow, as well as the pressure at each node.

The results can be displayed either in the form of a table or on the network diagram.



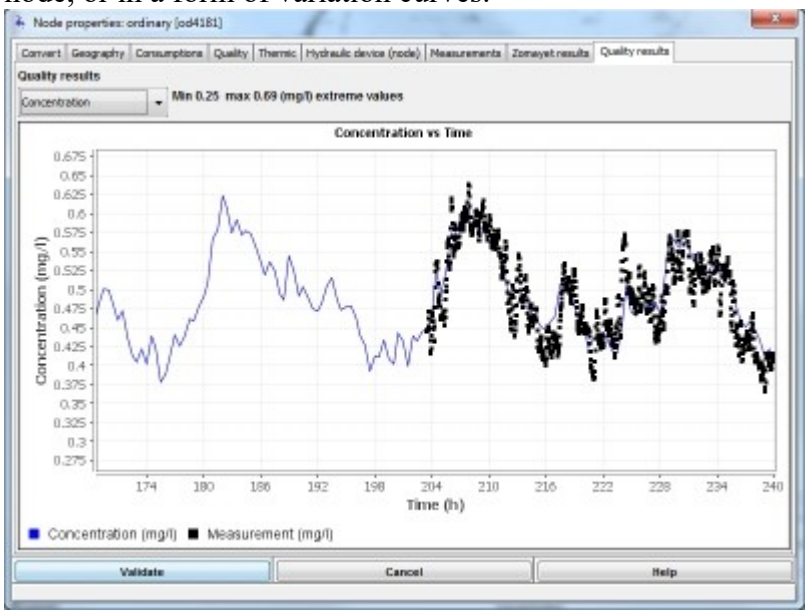
Module Quality

It provides to simulate changes in solute concentration throughout the network over time.
Chlorine is the most common solute. It is used to ensure the bacteriological quality of the water in the network. Excessively low concentrations of chlorine can render the water unfit for consumption while excessively high levels can be unpleasant for consumers. It is therefore essential to manage chlorine injections correctly in order to optimise concentations.
This module takes into consideration the legitimate expectations of consumers who now demand not only smooth, constant supply, but perfect water quality.



The quality calculation thus monitors the quality of the water in the network and optimises the quantities injected and/or the points where injections take place. The calculation is based on reaction kinetics and considers that mixtures at each node are perfect. Concentations are calculated on the basis of the hydraulic data obtained by the Zomayet module.
Three types of results are supplied: the concentration of a product (chlorine for example) the age of the water and the origin of the water. The results can be displayed either in the form of a table at each step for the whole network or on the network diagram, or in the form of a table over the whole simulation by

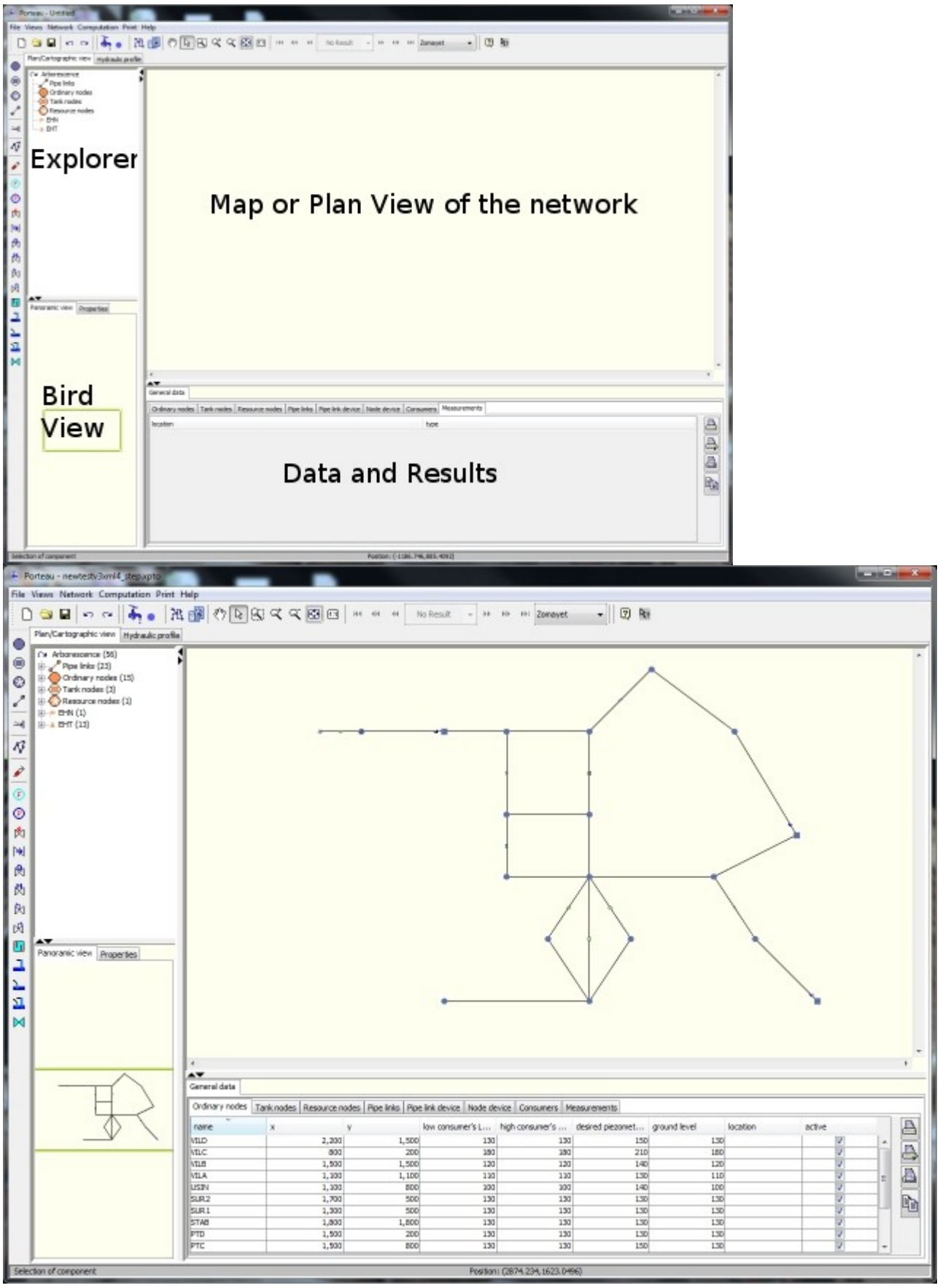
node, or in a form of variation curves.



Module Thermic

Latest of Porteau's modules, it allows to calculate the temperature of the water at every point in the distribution network. It is based, as module Quality, on the results of the module Zomayet and its speeds to simulate the transport of fluid and the evolution of the desired parameter. Its operation is similar to the module Quality, the main parameters are the thermal constants of exchange and the outside limit temperature for sections.

Windows


Explorer
Map or Plan View of the network
Bird View
Data and Results

Menus

File Menu

- New: to create a new project.
- Open: to open an existing project.
  The opening or the creation of a file automatically closes the current file opened.
- Concat: to concat the current network with an existing file.
- Save: to save the current project.
  Perform often to avoid losing any changes to the project.
- Save As: to save the changes into a project with a new name.
- Import a PTO file: to import an old project Porteau V2 into the new XPTO format.
- **Import a SHP file:** to import a Shape-coded file into the current network.
- **Export a SHP file:** to export a part of the data into the SHP format.
  - Export of Nodes : to export the nodes into the SHP format, if results are present, they are exported into the CSV format.
  - Export of Sections : to export the pipes into the SHP format, if results are present, they are exported into the CSV format.
- **Import an INP file:** to import an INP file into the current network.
- **Export in INP format:** to export a part of the data into INP format.
- Recents: list of 10 last opened file to reopen.
- Exit: to exit Porteau.

View Menu

- Map:
  - Map view: show the graphic view in map view. The coordinates of the nodes define the graph cartography; pipe segments are represented as polylines if they possess intermediate vector coordinates.
  - **Select background:** to select an image that will become the map background.
  - **Resize background:** to alter the scale of the background to that of the cartography.
  - Show/hide the background: to switch on or off the background image.
- Schematic:
  - Schematic View: to show the graphic view in schematic view. The coordinates of the nodes used for the graph are the "schematics"; pipe segments are represented as straight lines even if they contain intermediary points.
- **Current graphical attribute:** to set the attributes of the graphics views: arrow direction, choice of labels, choice of thickness and color attributes by object type (ordinary node, tank, resource, pipe).
- **Graphic styles:** to manage the sheets of graphic styles: add, delete, copy, print a graphic style.

Network Menu

- **Management of materials/pipe types:** to allows the management of libraries of materials and pipe types for the application and for the project, and the exchange of data between them.
  - Application library: to conserve the materials and pipe types to use them in other projects, it consists of a file stored under the user's profile: C:\Documents and Settings\User\ porteau\bib\catalogue.xml, thus by copying this file to another user's profile, it is possible to exchange lists of materials and pipe types. This can also be achieved simply by transferring a project file.
  - Project library: all materials and pipe types, these data are stored in the project file (xpto).
- **Management of Sectors-Services:** to allows the management of sectors and services of the network.

- **Consumption models:** To manage the consumption in the network with different patterns.
- **General:** To manage the data of each calculation module, the algorithm parameters, the warning levels, preferences related to the project. Certain data are saved in the project file, others in a file linked to the user profile in the file, C:\Documents and Settings\User\porteau\bib\contexte.xml.
- **Synthesis:**
  - **For the network:** report of the different objects met in the project.
  - **Pipe sections:** cumulative length per material/pipe type.
  - **Distribution:** report by consumer model.
  - **Supply:** establishes the global report of pumps, motorised valves, tanks et resources.
  - **Pipe devices:** report of hydraulic devices related to the pipes.
  - **Node devices:** report of hydraulic devices related to the nodes.
- **Pressure Indicators**: To calculate pressure indicators by sector or for all of the network with Zomayet's results.
- **Quality Section Class**: to manage quality classes for pipe segments.
- **Thermic Section Class**: to manage thermic classes for pipe segments.
- **Origins Management**: to manage the list of tracked nodes crossed by the path of water.
- Tools:
  - Concat link without demand: concat links with the same pipe, same diameter, same roughness, same Hazen Williams, if there is no consumption along the link, no equipment and the node between the two links is converted in vertex. A logfile of merger operations can be created, if Cancel is clicked at the request of the filename, no tracking will be made. The choice of tracking or not is permanent for the running time of Porteau.
    The logfile contains:
    "#CONCAT";LinkName1;BeginName1;EndName1;LinkName2;BeginName2;EndName2;ErasedNodeName;NewBegin;NewEnd
    CONCAT;od6->od7;od6;od7;od7->Nd74;od7;Nd74;od7;od6;Nd74
    The first line gives headers of column, it is repeated every time the merger menu is selected.
    CONCAT refers to concatenate two sections described by their respective names and extremities, by the name of the deleted node, and by those of the extremities of the section so created.
  - Concat link with demand: concat links with the same pipe, same diameter, same roughness, same Hazen Williams, no equipment, if there is a consumption along a link, it is affected to the new link, if the node between the two links has a consumption, it is shared between the two end nodes of the new link. The node between the two links is converted in a vertex of the new link.
  - **Translate coordinates**: provides a translation of nodes and vertex of the network in the diagram view or cartographic view, or both.
  - **Concat dead ends**: delete nodes and links of dead end without demand, one link by calling the function, or shorten dead end with a criterion of length and/or bigger diameter of the links and concat consumptions at the root node.
  - **Import demand by CSV**: import demand by CSV format file on nodes and/or links.
  - **Import demand by SHP**: import demand by SHP format file on the nodes by projecting position of consumers on links with model and quantity affected.
  - Connexity to service, connected link to sectors: calculate the number of connex subnetwork. A subnetwork is a part of the network with no connexion pipes to the rest of the network, or isolated by closed valves. If there is a tank and/or a resource node in the subnetwork, porteau can calculated it, otherwise no module can be calculated. Each subnetwork is affected to a new service. The number of adjacency links for each node is calculated and affected in a new sector. There is the same number of sectors as the maximum adjaceny of a node in the network. **Be careful**: All services and sectors existing before the operation are deleted.

- **DEM to ground level**: the ground level is interpolated with the DEM data (Digital Elevation Model). The format of DEM file read with the Geotools library may be Ascii Grid ERSI, GeoTiff, Erdas Imagine. The interpolation method of the value of the grid is calculated by Sextante library, by BicubicSpline. If a selection of nodes is done, only selected nodes may be affected by the DEM import, otherwise all the network is affected.
- **Reproject the network**: give a new projection to the network, editing EPSG codes (origin, destination) and change the projection system of the network, to merge with another network which one has a different projection.

Calculate Menu

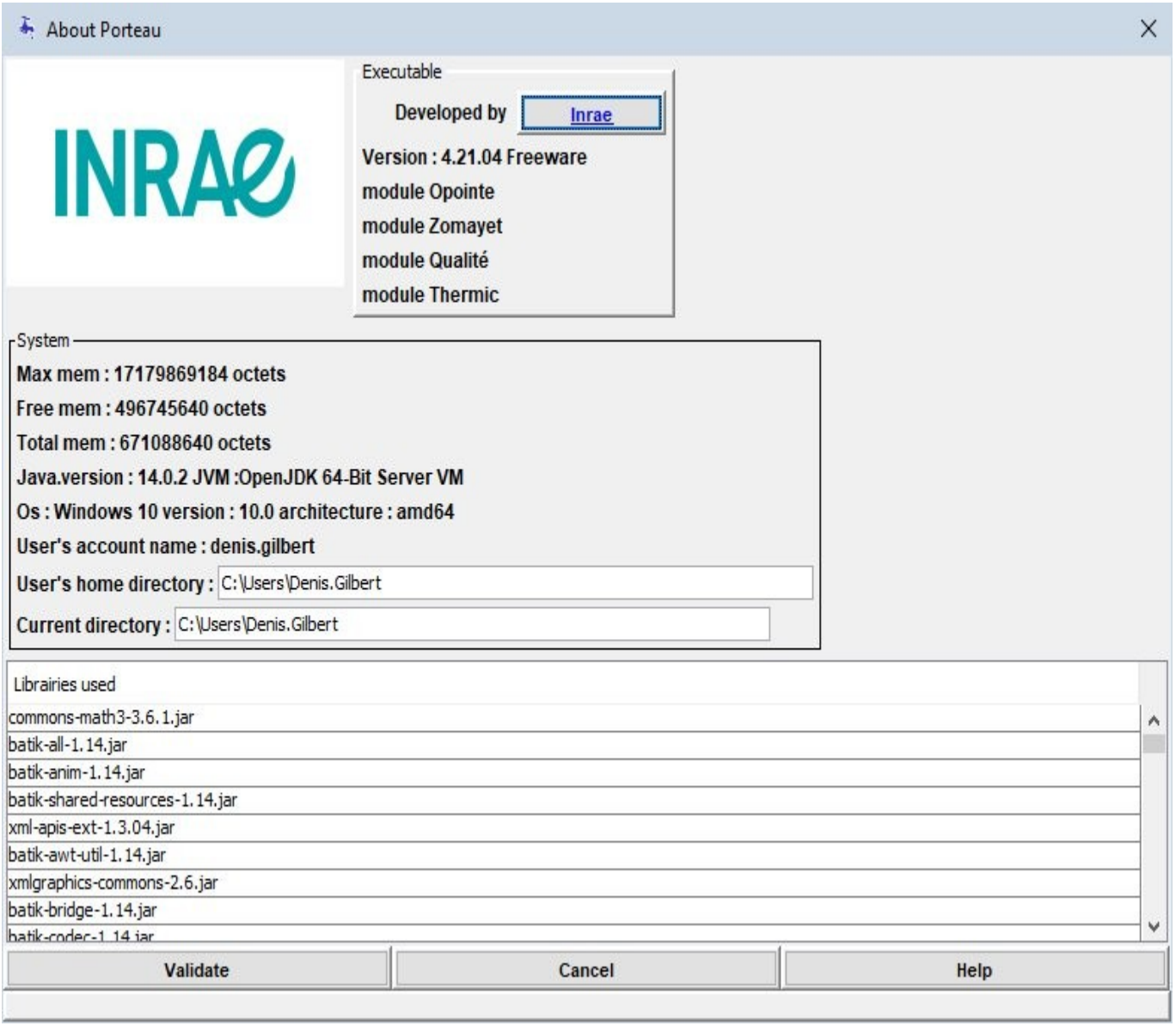
- Opointe: launch calculations with Opointe module.
- Zomayet: launch calculations with Zomayet module.
- Quality: launch calculations with Quality module.
- Thermic: launch calculations with Thermic module.

Print Menu

- Print…: launch the dialog box for the choice of printing options and print the current graphics view.
- Print preview: show the print preview for the current graphic view according to the chosen options.
- Page layout: launch the dialog box for the choice of parameters for the page layout to print the current graphics view.
- Set print zone: allows the tracing of the printing zone on the current graphics view.
- Delete print zone: delete the frame of the print zone; printing will include all the space occupied by the project.

Help Menu

- Help: opens the online help window.
- About: version of Porteau and data about system.



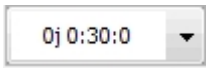
The "About" dialog box allows to verify:

- the current version of Porteau
- the available memory corresponding to that set in the launch shell of Porteau by the Xmx JVM parameter
- the version of the JVM: Java
- the operating system and its 32-bit or 64-bit architecture
- the settings of the user, here the configuration files and log files are stored directly in the root of C drive in a folder named Porteau; default this folder is created in the user profile "HOME"
- A table lists the libraries used in Porteau

The standard button toolbar

The standard button toolbar contains icons and popup lists. These buttons allow a command to be executed; whenever you point at an icon, an explanation of the associated command is displayed in an information bubble.

1. File -> New: Create a new project
2. File -> Open: Open an existing project
3. File -> Save: Save current project
4. Undo

5. Redo
6. Appy/Remove graphic style
7. Show names of nodes
8. Hide names of nodes
9. View -> Map -> Map view: Show Map View
10. View -> Schematic -> Schematic View: Show Schematic View
11. Show/Hide objects legend
12. Search a Node or a Pipe
13. Panorama
14. Selection
15. Zoom window
16. Zoom in
17. Zoom out
18. Show all
19. Result of first time step
20. Result 1 hour before current time
21. Result 1 step before current time
22. Result 1 step after current time
23. Result 1 hour after current time
24. Result of last time step
25. 

List of choice of results show

The tool palette

It proposes the different objects to model in the modules.
To select a tool, click on the icon of the palette corresponding to the tool of your choice. The cursor changes appearance as a function of the selected tool.

- Create an ordinary node.
- Create a tank node.
- Create a resource node.
- Create a pipe section.
- Divide a pipe section.
- Enter a path for the hydraulic profile.
- Create a disinfectant injector on an ordinary node.
- Create a velocity pump on a pipe.
- Create a power pump on a pipe.
- Create a motorised valve on a pipe.
- Create a check valve on a pipe.
- Create a local headloss on a pipe.
- Create a flow control valve on a pipe.
- Create a pressure reducing valve on a pipe.
- Create a pressure sustaining valve on a pipe.
- Create a pressure reducing valve on a pipe.
- Create an overflow inlet of a tank node.
- Create a float valve of a tank node.
- Create a combined float valve-overflow inlet of a tank node.
- Create a valve on a pipe.

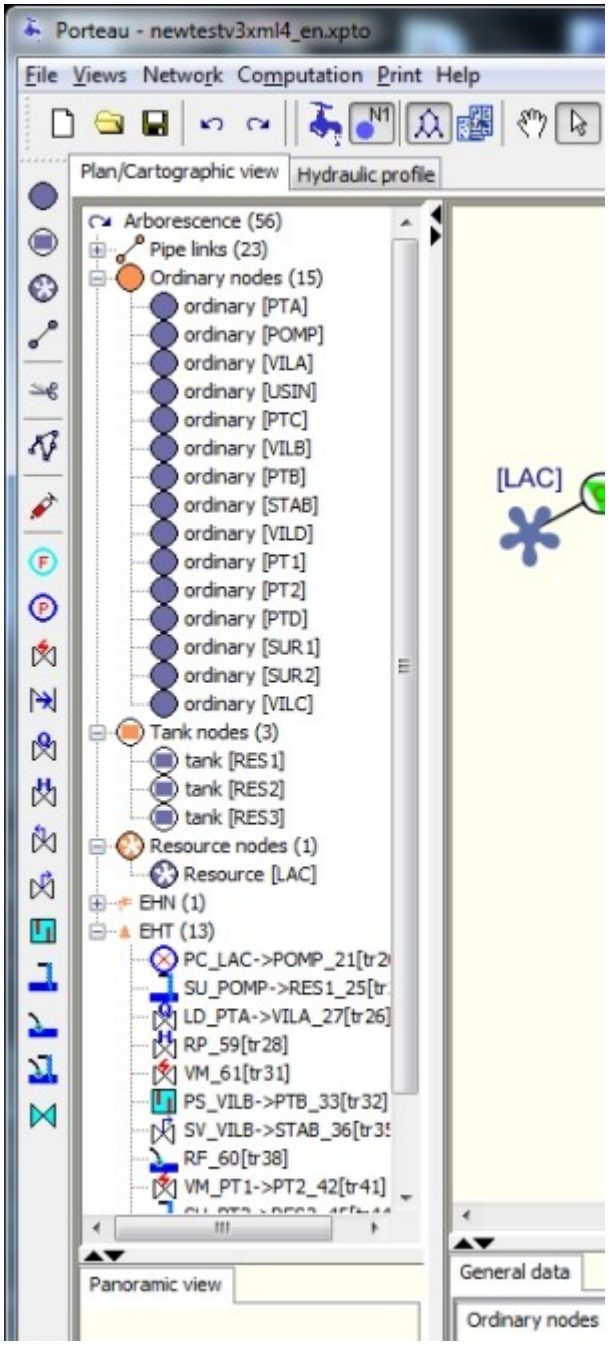
Tables

All the tables are linked with the graphic view by selection. They are printable and copiable to the clipboard in tabulation separated format by a simple click on the button  on the right side of the table. Usually they are not editable for control data reason, only dialog boxes do control.

- Data
  - Ordinaries Nodes.
  - Tank Nodes.
  - Resource Nodes.
  - Pipe Links.
  - Pipe Link Devices.
  - Node Devices.
  - Consumers: data for all consumers (node or pipe link, model, value)
  - Measurements
- Opointe Results: Same table as data except consumers and mesureaments.
- Zomayet Results: Same table as data except consumers and mesureaments.
- Quality Results: Only Ordinary and Reserve Nodes.
- Thermic Results: Only Ordinary Nodes.

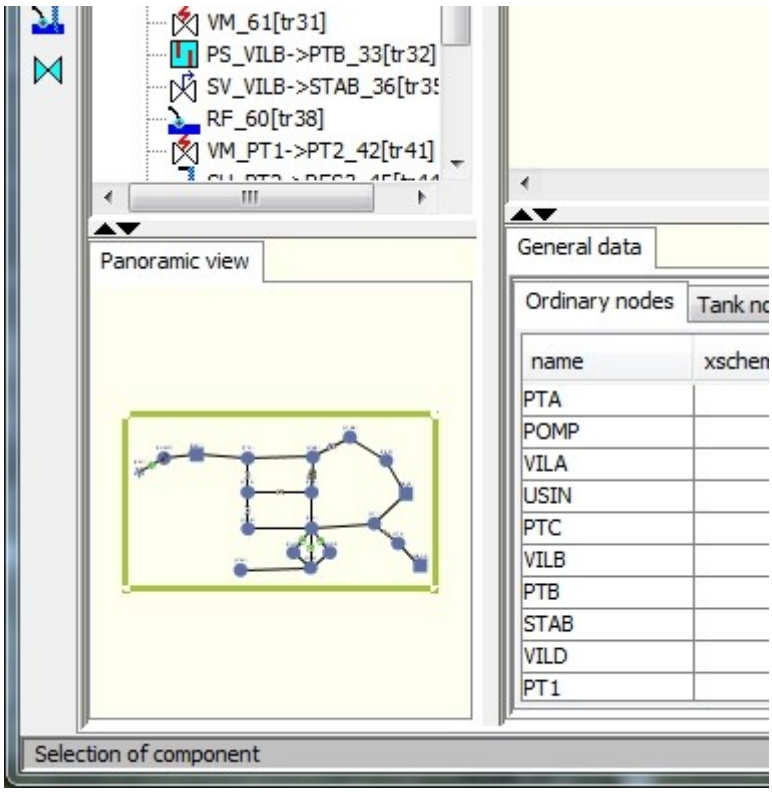
View object in objects Explorer

Explorer uses a tree for group informations in directories. To develop a directory, click on the plus (+) caracter or double click, this action show more detailled informations. Double click on an object (node, pipe) to center the view on the object (if it's not visible) and open his dialog box.



The selected objects in the tree are also selected in all of the network views : graphic and tables if present.

Panorama View



It is a small view of the entire network. You can use it to choose the visible part of the network or move the rectangle showing this part. You can change the rectangle dimensions.

Mouse

Graphic View

- Right click of the mouse in graphic view:
  Show an information bubble on the data of the object pointed at.
- Double left click of the mouse in graphic view:
  Switches to panoramic when no object selected.
  On an object: Shows the properties dialog box for the selected object .
- Double right click of the mouse in graphic view:
  Show a contextual menu based on the selected object.
- Wheel click: deplacement view action.
- Use of mouse wheel in graphic view:
  Forwards: zoom in on current view.
  Backwards: zoom in on current view.

Tabular View

- Click left: selection of rows.
- Click right: choice of columns to display
- Click column header: increasing, then decreasing sorting column.

Keyboard shortcuts

Creation of node/pipe segment

In creation mode:

- Node: Shift + left click, new node and pipe section with last node of project.
- Pipe: Shift + left click for the end of the pipe section, new pipe leaves from an existing node and creates a new node serving the end of the pipe at the position of (Shift Click).
- Pipe: Ctrl + left click to edit vertex point if present. One more ctrl+left click add a new vertex, delete a selected vertex if it is at the same position as the mouse.

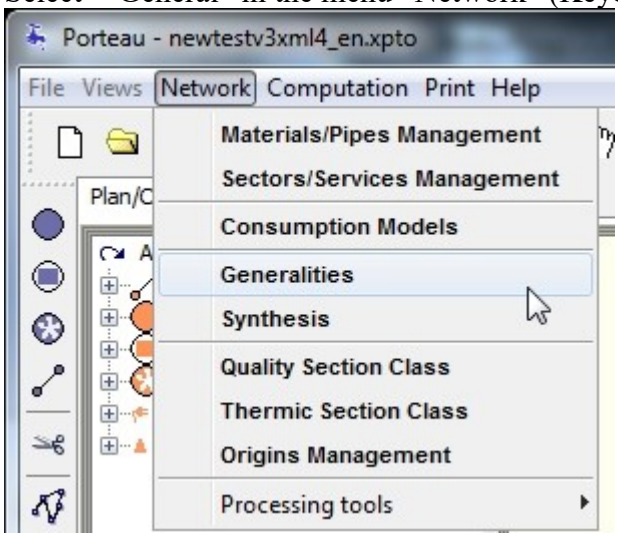
Modifying hydraulic profile

In the editing mode of hydraulic profile:

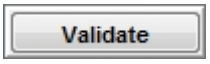
- In the tab Profile in detail: right click on the menu of profiles: menu contextual.
- In the graphics view: tool creation of profile  ,click on the first node of the path, then from pipe section to pipe section, to go back click on the last pipe section.

General Data

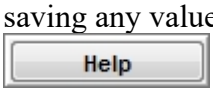
Select ""“General” in the menu “Network” (Keyboard shortcut: <Alt>+R+G)



A dialog box containing six tabs entitled “General” is shown... The default tab is “General data.” At the bottom are 3 buttons:

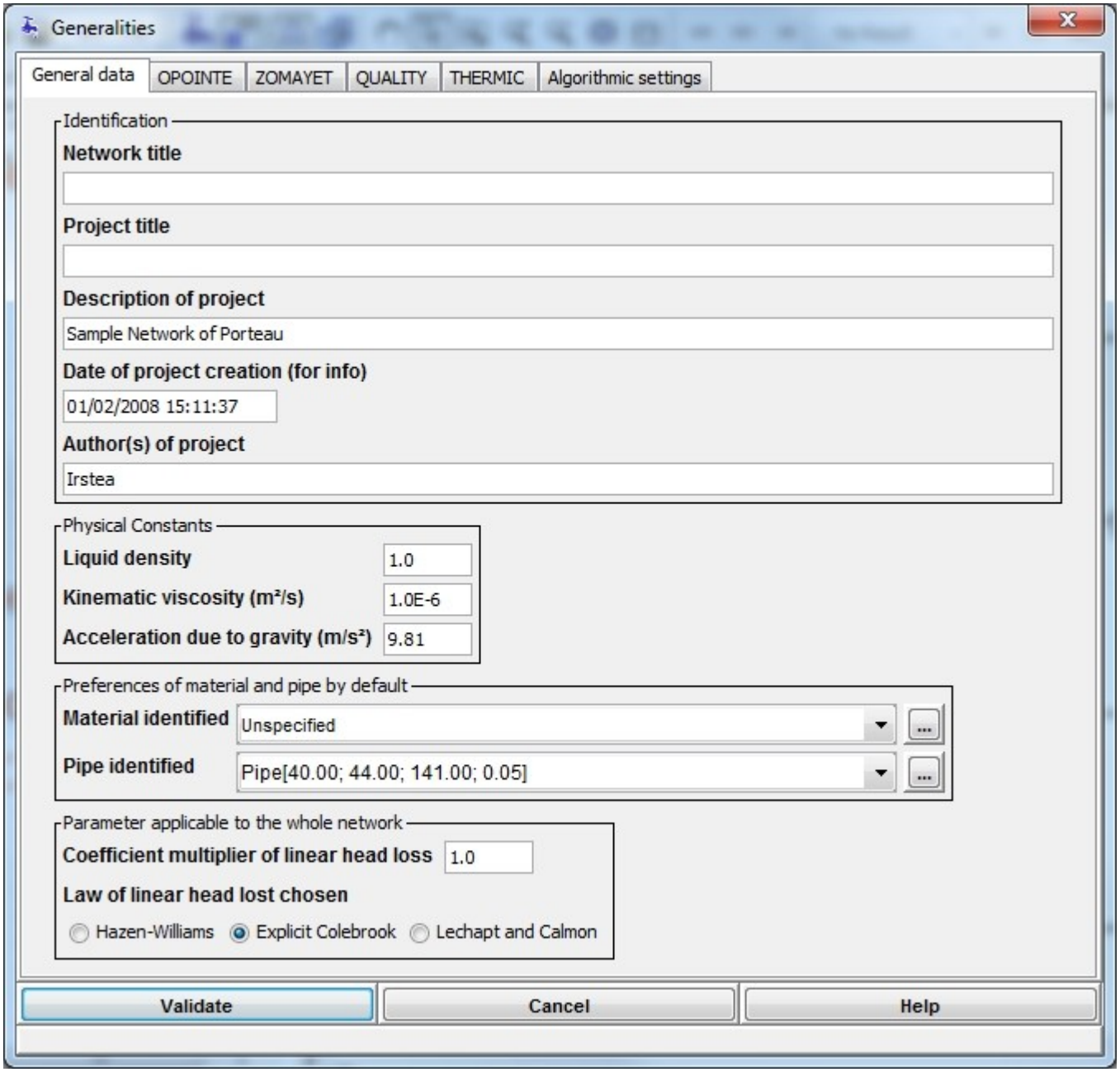
- 

validates the data entered in the 6 tabs and exits “General” dialog box
- 

cancels the data entered in the 6 tabs and exits “General” dialog box without saving any values entered or modified
- 

accesses to contextual help depending on the active tab.

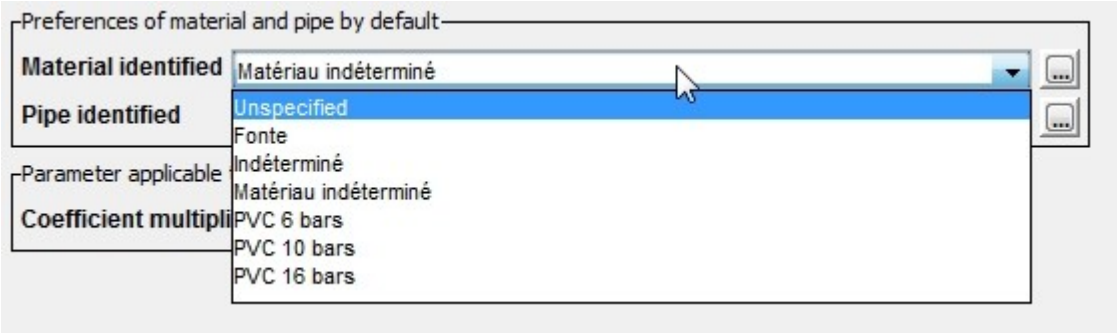
General data tab



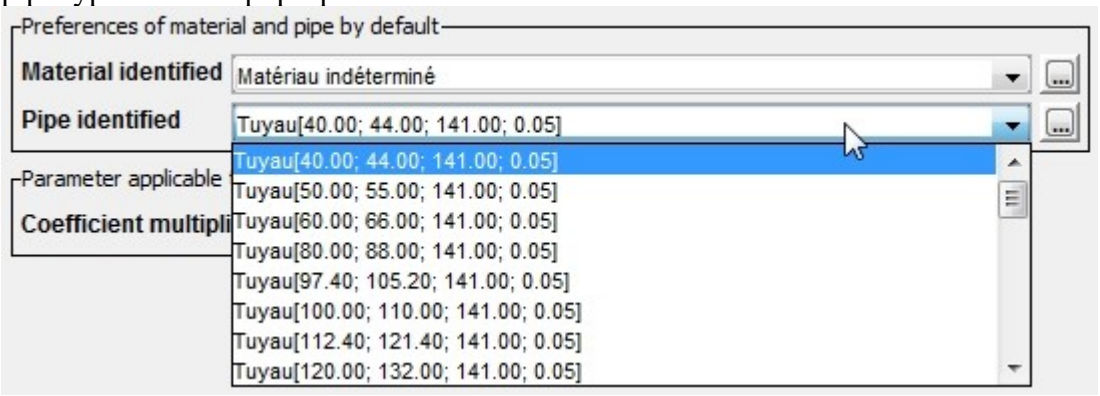
Identification : **Title of Network:** enter the name of the network.**Title of Project:** enter the name of the project.**Project description:** enter the description of the project.**Project creation date:** the file creation date is shown and can be modified. **Project authors:** enter the name of the project author(s).

Physical Constants : **Density of liquid:** Also called the specific gravity of water. This field allows the density of water to be changed in particular cases, as a general rule the density should equal to 1.00. This value is not used in the calculations.**Kinematic viscosity (m²/s):** default value: 10-6**Acceleration due to gravity (m/s²):** default value of g: 9.81

Preference of default material and pipe type : **Identified material:** This is the choice of material that will be used by default when creating a new pipe section. The choice is made from a pop-up list:



from the materials already created. The material **"unspecified"** is proposed at start up to avoid total blockage. Note, the name of the material has no effect on the calculations, it is only for information. The button to the right allows access to the **"management of materials"** dialog box to add one.**Pipe type identified:** From the list of pipe types associated with the material chosen in the previous step, choose a pipe type from the pop-up list:

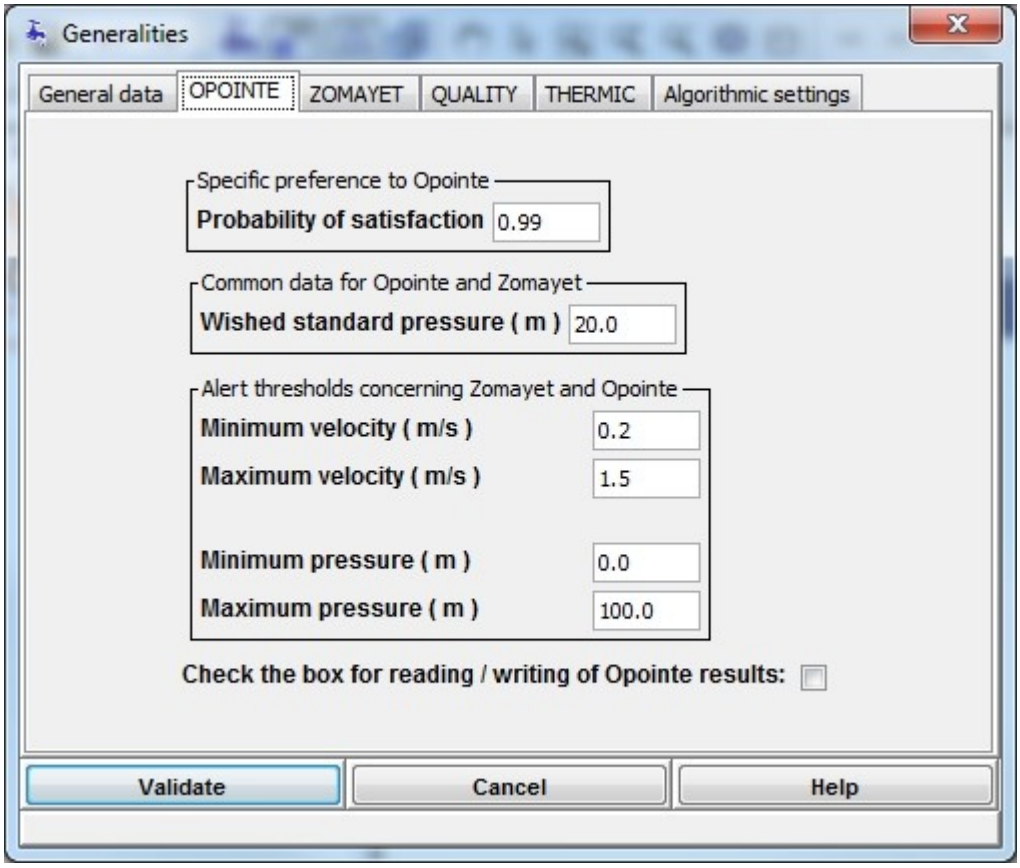


that will be proposed by default during the creation of a new pipe section. The button to the right allows access to the **"modification of a material"** dialog box to add one.

Parameter for the entire network : **Multiplier for linear headloss:** value applied uniformly to all pipes of the network multiplying the linear headloss of the pipe.
The law of the headloss used for Opointe and Zomayet calculations is chosen among three possibilities: Hazen-Williams, Colebrook and Lechapt-Calmon. **Law of linear head loss chosen:**

- Hazen-Williams
- Colebrook explicite
- Lechapt-Calmon

Opointe tab



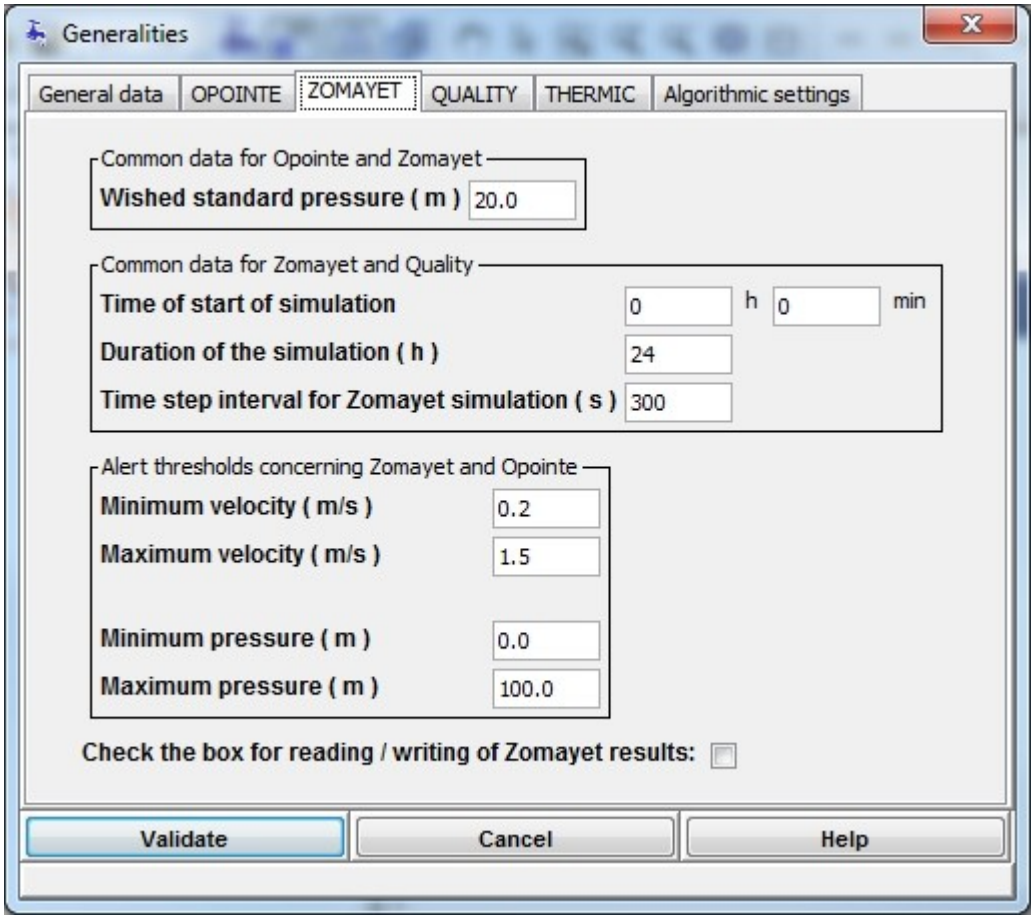
Preference specific to Opointe : **Probability of satisfaction:** The probability of satisfaction of consumers: enter a number between 0.95 and 0.99 ; advised value: 0.99

Data common to Opointe and Zomayet : **Desired piezometric head (m):** By default it is the minimum pressure to be present at the consumer nodes. If inferior pressures are encountered, an error message is displayed at the end of the calculation. It corresponds to the desired pressure to supply all consumers; a value of 20m is generally considered acceptable. This value will be added to the elevation, during the specification of a node, to give the desired piezometric head. This value must be between 1 and 250m.

Threshold alarms common to Opointe and Zomayet : **Minimum velocity (m/s):** The velocity of water circulating in the pipes must not be less than this minimum velocity. If this occurs, an error message will appear at the end of the calculations for the Opointe and Zomayet modules.**Maximum velocity(m/s):** The velocity of water circulating in the pipes must not be greater than this maximum velocity. If this occurs, an error message will appear at the end of the calculations. It is advised to use a maximum value corresponding to the limits of the network, generally between 1.5 and 3 m/s.**Minimum Pressure (m):** The value of pressure at an ordinary node for which the calculated pressure should be superior. An error message is shown if this minimum is not achieved.**Maximum Pressure (m):** The value of pressure at an ordinary node for which the calculated pressure should be inferior. An error message is shown if this maximum is exceeded.

A **check box** allows results from Opointe to be saved and reread if checked.

Zomayet tab



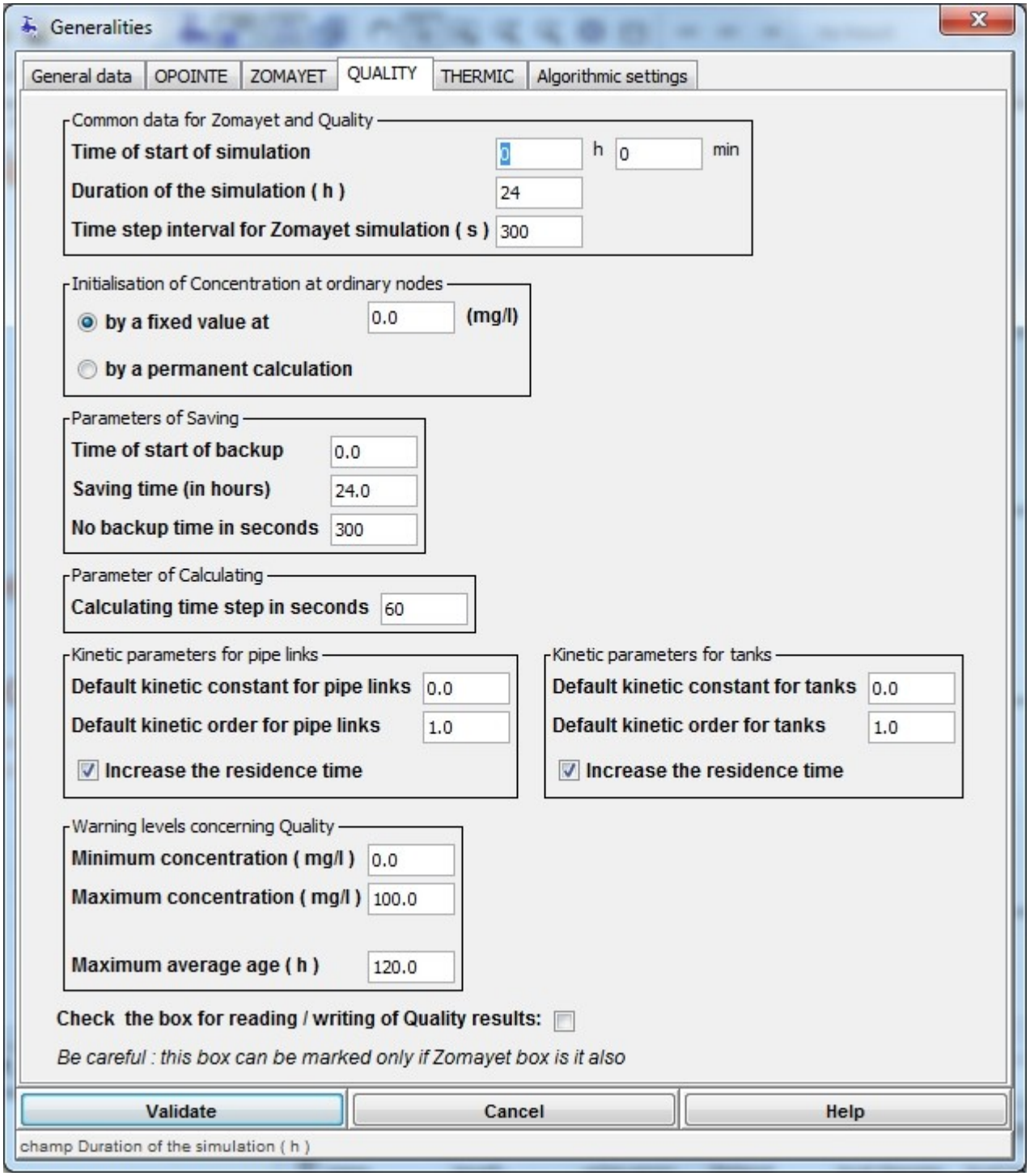
Data common to Opointe and Zomayet : **"Desired piezometric head (m)":** By default it is the minimum pressure to be present at the consumer nodes. If inferior pressures are encountered, an error message is displayed at the end of the calculation. It corresponds to the desired pressure to supply all consumers; a value of 20m is generally considered acceptable. This value will be added to the elevation, during the specification of a node, to give the desired piezometric head. This value must be between 1 and 250m.

Data common to Zomayet and Quality : **"Simulation duration (h)":** enter a value > 0 in number of hours (maximum 500 h).**"Time of beginning of simulation":** enter two values, the hour and if there is place the minutes.**"Time step for Zomayet simulation (min)":** enter value between 1 and 60. In order to limit the derivatives during the calculation, notably for the tanks, it is recommended to use 5 minutes. However, in the case of a complex network, 5 minutes may lead to quite long calculation times. Usually 12 or 15 minutes provides a good compromise during initial calculations.

Threshold alarms common to Opointe and Zomayet: cf Opointe Tab :

A **check box** allows results from Zomayet to be saved and reread if checked.

Quality tab



Concentration at ordinary nodes : **"Initialisation of the concentration at ordinary nodes":** Choose the initialisation mode. If the first button is selected it is necessary to enter **"Initial ordinary node concentration"**

Backup Parameters : **Duration of simulation in hours:** duration of backup quality results, it's not interesting to save the beginning of the simulation which just intialize the values in the network ; but they can use a large memory space.**Time interval for simulation in seconds:** if quality values have small variation, it can save memory to save results with a larger time step than hydraulics.**Time of beginning of simulation:** link to the duration to save only interesting time steps.

Calculating Parameters : **Time interval of calculation in seconds:** a time interval used to calculate Quality parameters - it must be compatible with the time interval of simulation in backup (inferior or equal) and with the time interval of Zomayet (inferior or equal).

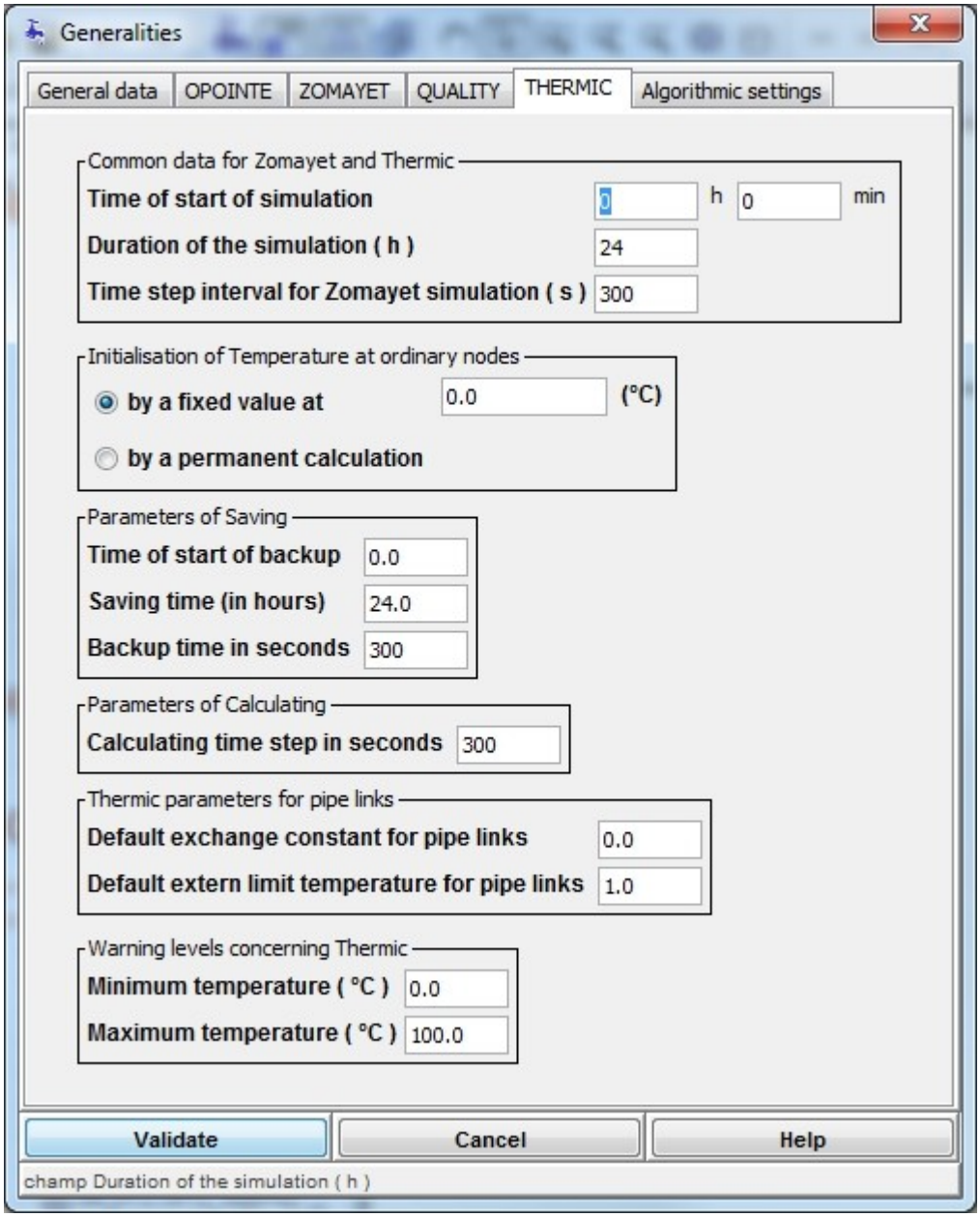
Kinetic Parameters for pipes : **Default kinetic constant for pipes:** default value of constant for pipes at creation. **Default kinetic order for pipes:** default value of order for pipes at creation.

Kinetic Parameters for tank nodes : **Default kinetic constant for tank nodes:** default value of constant for tank at creation. **Default kinetic order for tank nodes:** default value of order for tank at creation.

Threshold alarms for quality : **Minimum Concentration (mg/l):** lower value below which the concentration must not fall. If this appear, a fault message is generated by PostTreatment of Quality Module. **Maximum Concentration (mg/l):** upper value above which the concentration must not climb. If this appear, a fault message is generated by PostTreatment of Quality Module. **Maximum residence time (h):** upper value above which the residence time must not climb. If this appear, a fault message is generated by PostTreatment of Quality Module.

A **check box** allows results from Quality to be saved and reread if checked.

Thermic tab



Concentration at ordinary nodes : **Choose the initialisation mode:** if the first button is checked, indicate the "Ordinary initial temperature".

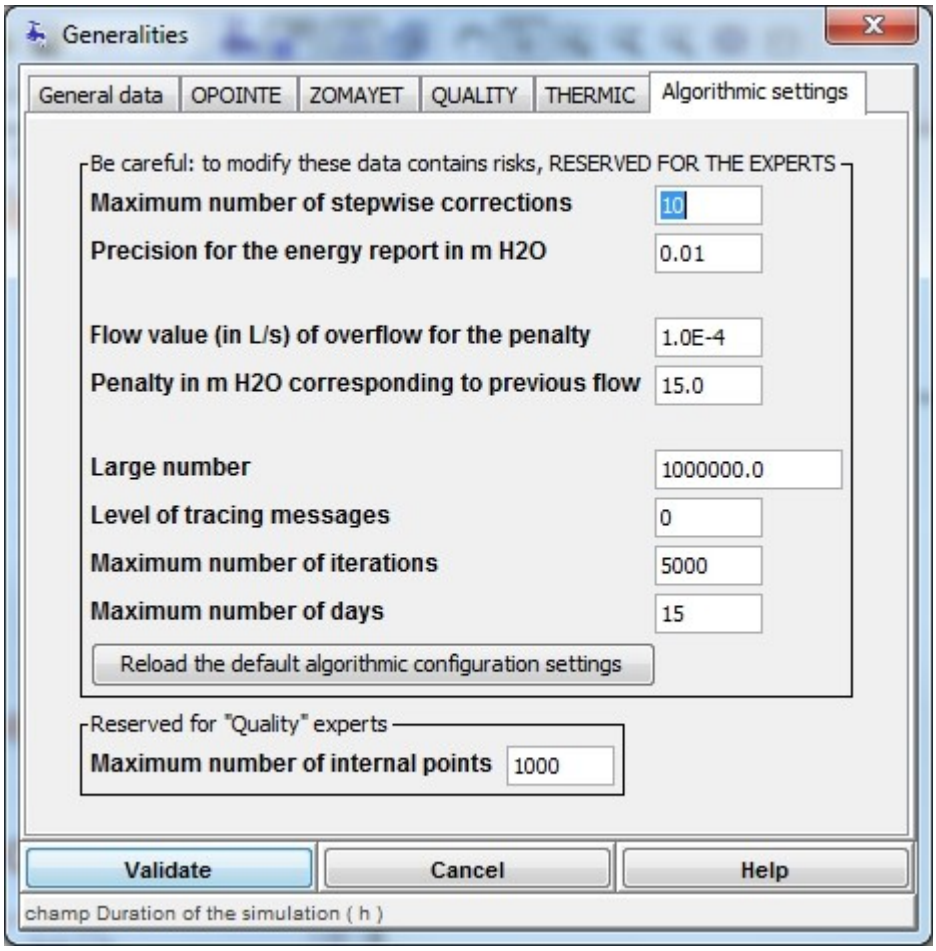
Backup Parameters : **Duration of simulation in hours:** duration of backup thermic results - it is not worth saving the first time intervals of the simulation used to initialize the calculation, because they can take a large amount of memory. **Time interval for simulation in seconds:** if values have small variation, it can save memory to retain a part of results stored in constant time. **Time of beginning of simulation:** to connect with the time interval for simulation, only interesting time intervals.

Calculating Parameters : **Time interval of calculation in seconds:** a time interval used to calculate Thermic parameters - it must be compatible with the time interval of simulation in backup (inferior or equal) and with the time interval of Zomayet (inferior or equal).

Thermic Parameters for pipes : **Default exchange constant for pipes:** default value of thermic exchange constant with the outside for pipes. **Default limit exchange temperature for pipes:** default value of the outside temperature used by limit allocated to the pipe.

Threshold alarms for Thermic : **Minimal temperature (°C):** value below which the temperature should not go down. If it occurs, a message of anomaly is registered in the Post-treatment of the calculation for the Thermic module. **Maximal temperature (°C):** value above which the temperature should not go up. If it occurs, a message of anomaly is registered in the Post-treatment of the calculation for the Thermic module.

Algorithm Parameters tab



Remember : **Be careful:** modification of these data is reserved for experts. For normal use, conserve the data proposed by default.

Data for expert only : **Maximum number of descending stepwise corrections:** default value 10. Between 0 and 1000. **Energy balance precision in mH2O:** default value 0.01. Between 0.01 and 0.0001. **Penalty threshold flow in l/s:** default value 1.0E-4. Between 0.01 and 0.00001. **Penalty in mH2O corresponding to previous flow:** default value 15.0. Between 15.0 and 35.0. **Infinity:** default value 1 000 000. **Level of message tracking:** by default 0, a higher value (maximum 99) allows more information to be available in the tracking files on the iterations of the calculation. This is to be used when there is a doubt in the development of the results. **Maximum number of iterations:** default value 5 000. Between 1 and 30 000. **Maximum number of days:** default value 15. Between 1 and 100.

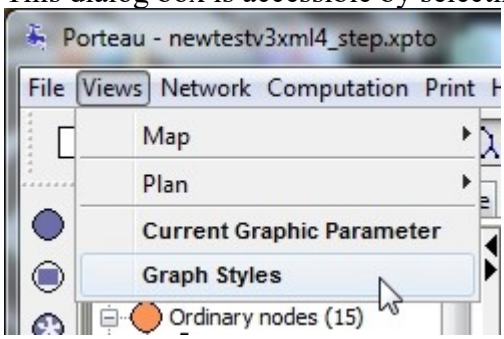
At the bottom of the screen, the button 

allows the default values to be restored if required.

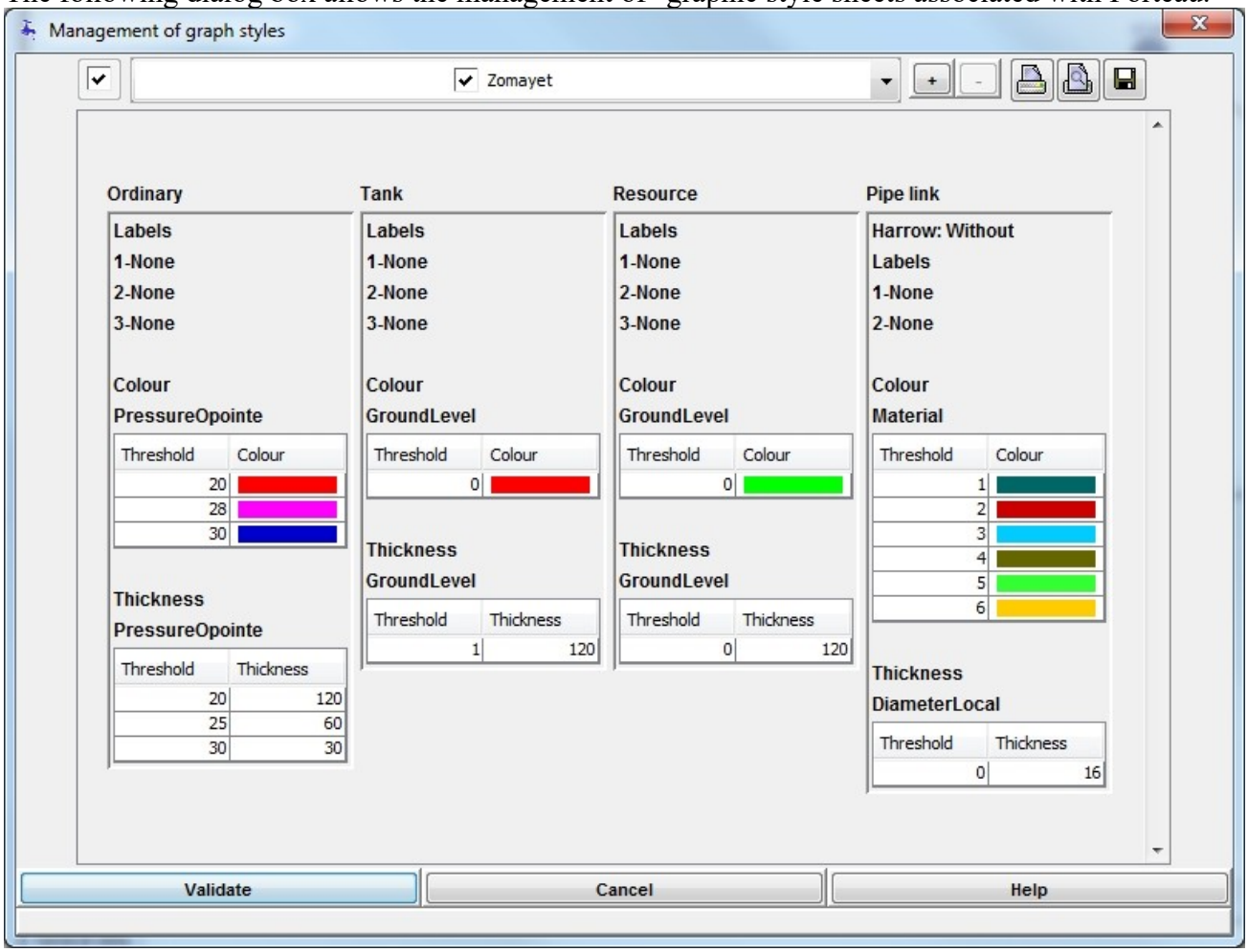
Data reserved for Quality experts : **Maximum number of internal points:** number of calculation points on a pipe through discretized for the calculation. Default 1000, between 1 and 100 000.

Graphic Styles

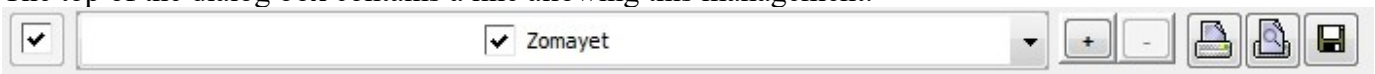
This dialog box is accessible by selecting the menu "Views", sub-menu "Graphic styles".



The following dialog box allows the management of "graphic style sheets associated with Porteau.

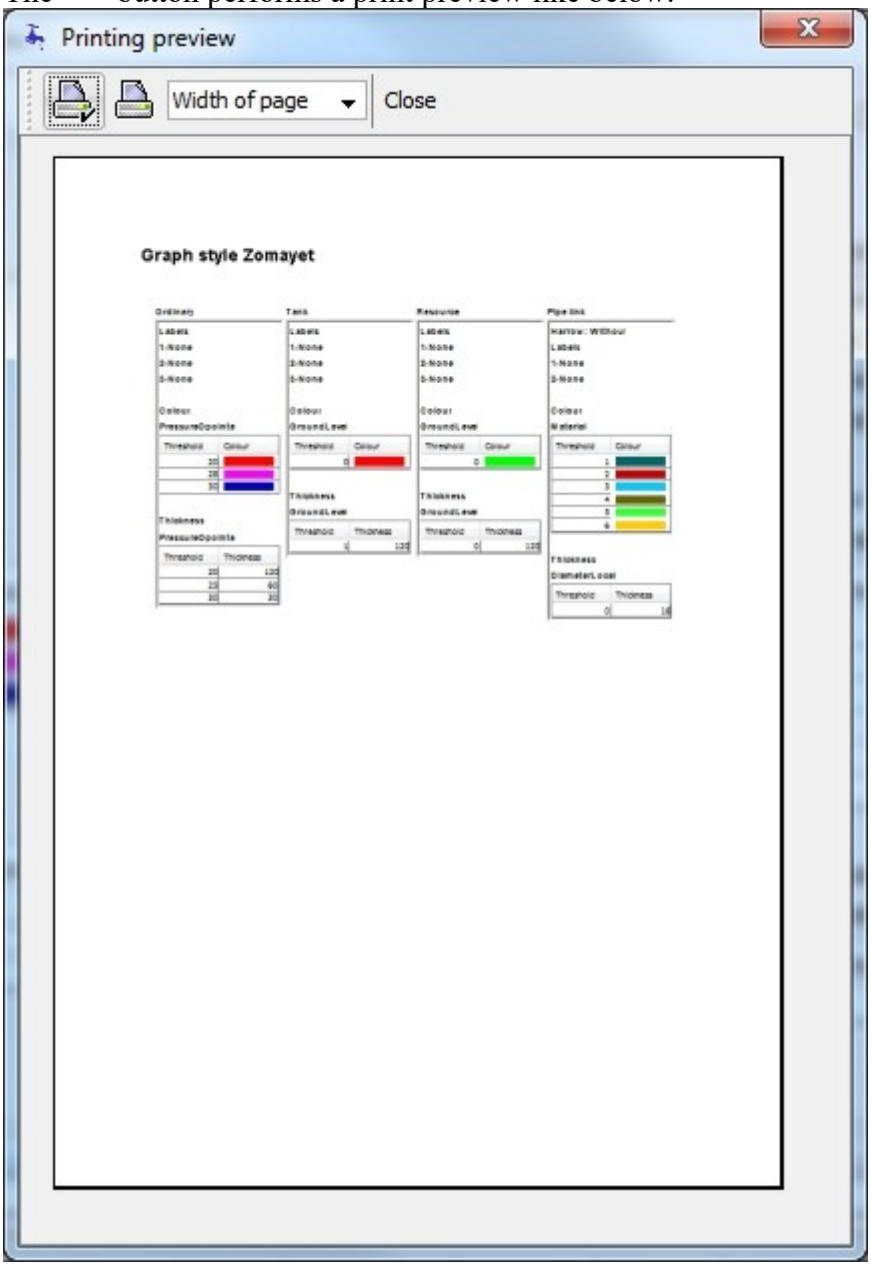


The top of the dialog box contains a line allowing this management.



- The first check box  on the left allows the style chosen in the list to the right to become "active".
- The scroll box allows the present styles to be chosen for manipulation (printing, visualisation in the space below, choice of active style etc.).
- The  button allows new styles to be added to the list by copying the chosen style in the list.
- To create a new style and make it active, click the button  then check the box situated before the scroll box.

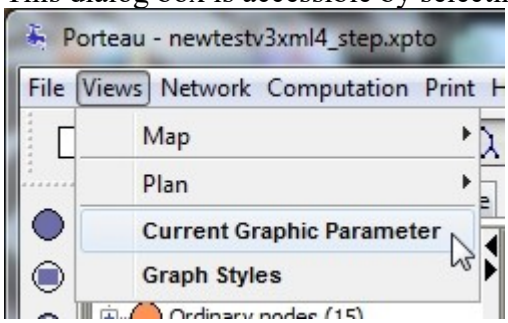
- The button allows the selected style to be deleted, as long as it is not the active style (the one with the checked box before the scroll list).
- The button allows the style to be printed on the system default printer with a presentation similar to the dialog box.
- The button performs a print preview like below.



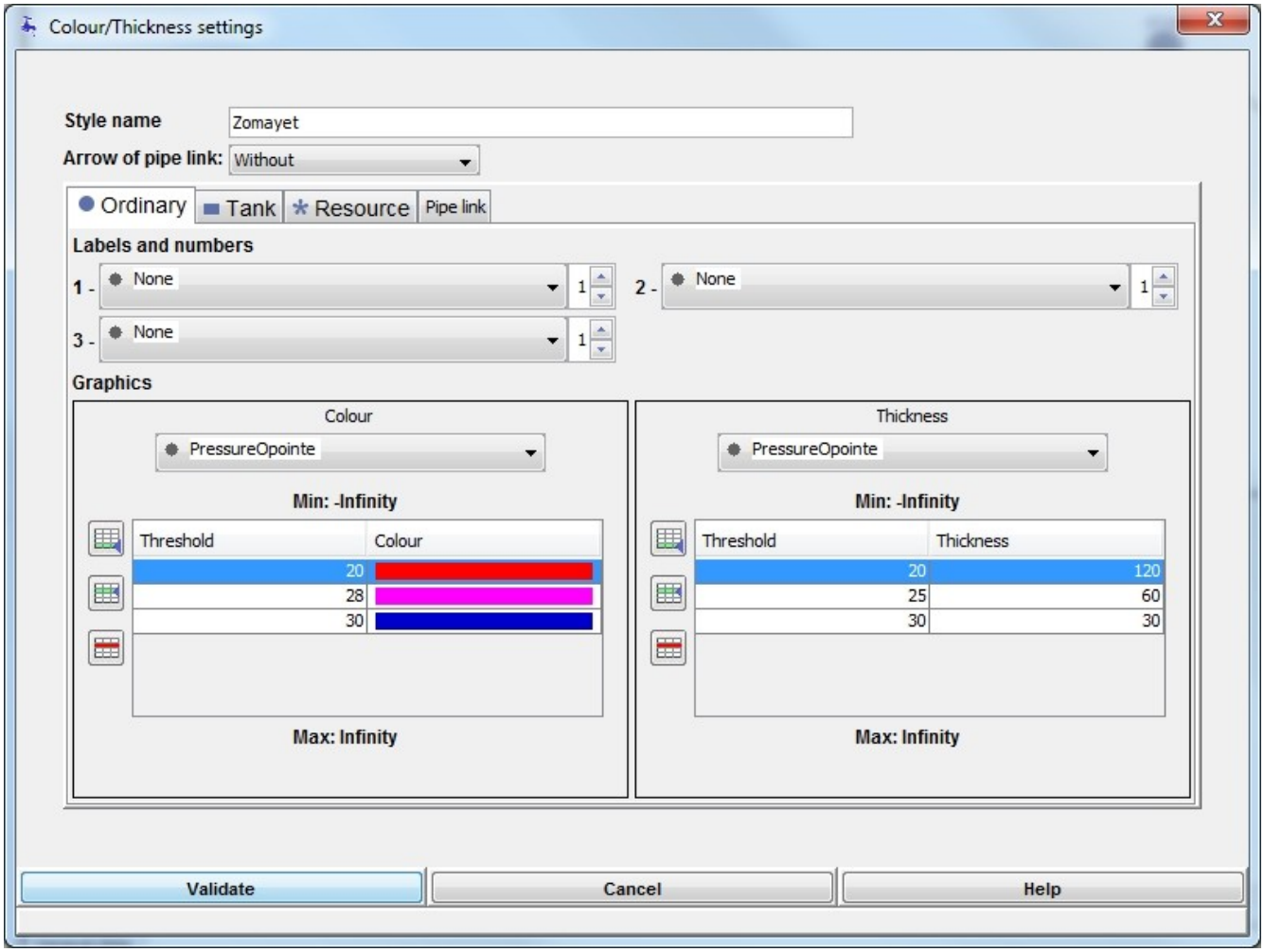
- This allows visualisation at different zoom levels and to modify the printer parameters if necessary.
- The button allows to save the printable style sheet in bitmap or vector formats.

Active Graphic Style

This dialog box is accessible by selecting the menu "Views", sub-menu "Active graphic attribute".

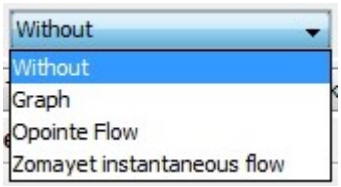


The following dialog box allows the management of all the attributes of the graphical representation of the network.



The field "Style name" allows a style to be identified by a chain of characters.
The field "Arrow for pipe" allows the selection of the type of arrow design for the pipes.

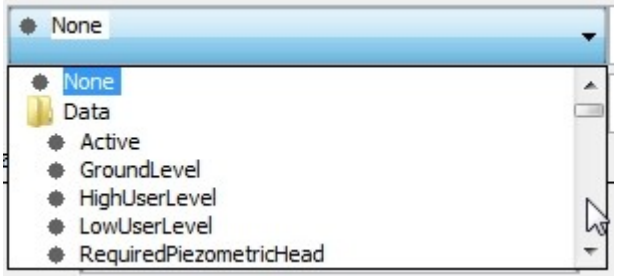
The scroll box gives the list of possibilities :



Without: no arrow is drawn. Graph: the arrow is in the direction the pipe was created (start node towards end node). Opointe flow: the arrow is in the flow direction calculated by Opointe if such a result is present for the pipe, otherwise no arrow is drawn. Instant Zomayet flow: the arrow is in the flow direction calculated by Zomayet for the time step (Instant) selected in the list of time steps des pas de temps if such a result is present for the pipe, otherwise no arrow is drawn.

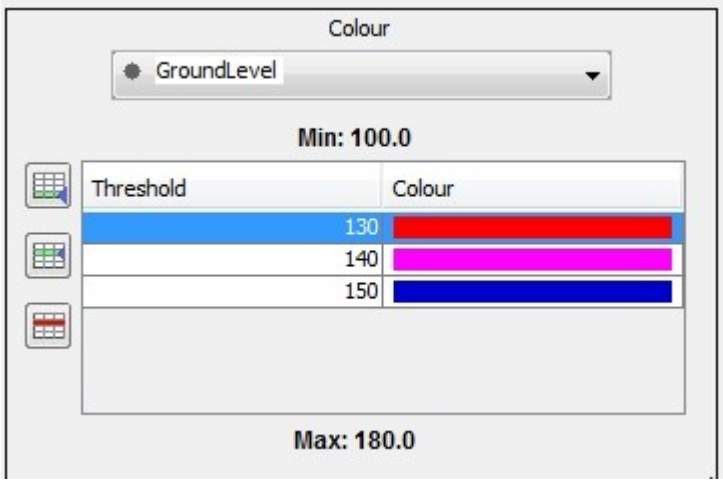
A tab allows the choice of objects to be parameterised: be it ordinary nodes, tank nodes, reservoir nodes, resource nodes or pipe segments. The parameterisation functions identically whichever tab is selected. On the graphic view, it is possible to write labels next to objects; three labels in addition to the name are possible for the nodes and two for the pipe segments. Both Schematic and Map views are illustrated identically. The choice of attribute written on the label is made by selecting from a scroll box for the type of object illustrated (pressure for node, flowrate for a pipe etc.).

This list shows the first choices for an ordinary node. The choices are organized by category :

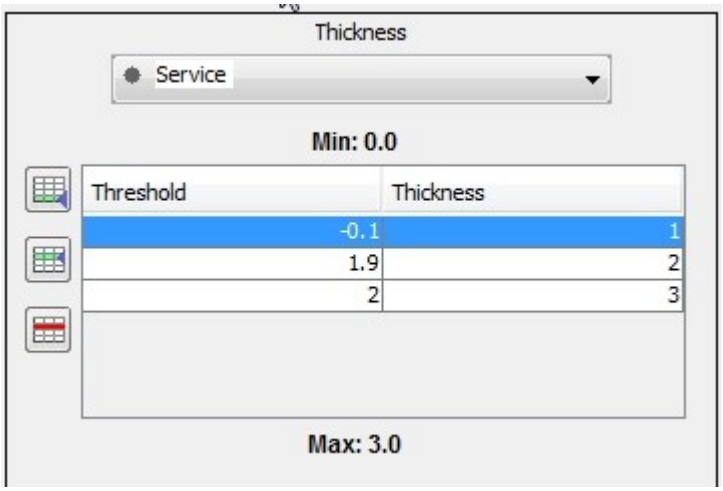


data, calculated data, Opointe results, Instant Zomayet results, Results report for the Zomayet simulation, Instant Quality results, Results report for the Quality simulation.

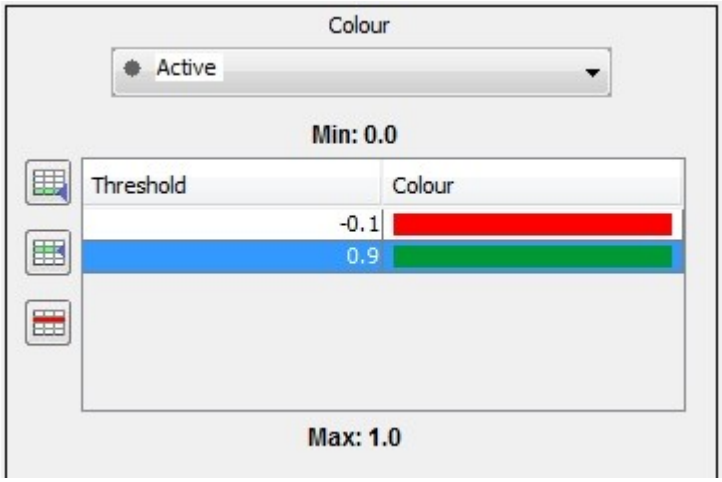
For all the labels, the choice of number of decimal places  allows values to be shown with the desired precision. These fields are present for all the labels but only effective for numerical values. To increase the number of decimal places, click on the up arrow; to decrease click on the down arrow.
The choice of colors for the attribute is also achieved through a scroll box.



When the attribute is chosen, its extreme values across the entire network are shown - the minimum (Min) value above and the maximum (Max) value below the table of parameters. Three buttons allow a line to be added to the bottom of the table , a line to be inserted above the current selected line , and the deletion of the selected line .
Each line becomes a limit with a corresponding colour such that if the attribute is greater than (>) the limit (and less than or equal to the next limit) the object is the colour of the line. If no colour is imposed by the values lower than tht of the first line, the objects will have the default colour (navy blue). The principal behind the thickness parameterisation is the same as for the colors.



This illustrates the choice of thickness attributes for node services. As the service is chosen from a scroll list of nodes (or pipes) , the parameterisation is made by the correspondence between the service and its rank in the list of services. Here, the smallest size (1) is applied to the service node with rank 0, the first in the list of services. If a node does not have a service, it keeps the default size (30).



The choice of colors above shows how active (green) and inactive (red) attributes can be displayed.

To apply current style, click the button in the standard button bar. If the button is on when style is modified, put it off and on by clicking **twice** on .

Lists of attributes for styling

Tables list attributes which can be used as label for a node or a link or be used as variable for color and/or thickness.
If the variable is boolean, for calculate the color or thickness true = 1, false = 0.
If the variable is a link in a table (like material, sector), the rank in the table is used, the first object of a table has the rank 0.

Attributes Ordinary Node

Data	
Active	True if active / false if inactive
GroundLevel	Value of ground level
HighUserLevel	Value of high user level
LowUserLevel	Value of low user level
RequiredPiezometricHead	Value of required piezometric head
Sector	Name of sector
Service	Name of service

Calculated	
SumDomesticUsers	Total of all users with type Domestic
SumIndustrialFlows	Total of flows with type Industrial
PeakFlowDuration	Peak Flow calculated on node for the duration of Zomayet's simulation
DomesticVolumeDuration	Volume consumed by domestic users for the duration of Zomayet's simulation
IndustrialVolumeDuration	Volume consumed by industrial consumers for the duration of Zomayet's simulation
VolumeDuration	Volume consumed for the duration of Zomayet's simulation
DifferenceHeadDesiredCalculated	Difference between the minimum of the simulated head and the local desired head

Opointe	
PiezometricHeadOpointe	Piezometric head calculated by Opointe
PressureOpointe	Pressure calculated by Opointe
PressureOpointeLow	Pressure for lower user calculated by Opointe
PressureOpointeHigh	Pressure for higher user calculated by Opointe
ExcessPressureOpointe	Excess of piezometric head between desired and calculated by Opointe

Zomayet Time Step	
PiezometricHeadTimeStep	Piezometric head at Zomayet Time Step
PressureTimeStep	Pressure at Zomayet Time Step
PressureTimeStepLow	Lower user Pressure at Zomayet Time Step
PressureTimeStepHigh	Higher user Pressure at Zomayet Time Step
ExcessPressureTimeStep	Difference between desired piezometric and calculated at Zomayet Time Step

Zomayet Duration	
PiezometricHeadMinDuration	Minimal Piezometric head reached for duration of Zomayet
PiezometricHeadMaxDuration	Maximal Piezometric head reached for duration of Zomayet
ExcessPressureMinDuration	Minimal Difference reached between desired piezometric head and calculated for duration of Zomayet
PressureMinDuration	Minimal Pressure reached for duration of Zomayet
PressureMaxDuration	Maximal Pressure reached for duration of Zomayet
PressureMinDurationUserHigh	Minimal Pressure reached on higher user for duration of Zomayet
PressureMaxDurationUserLow	Maximal Pression reached on lower user for duration of Zomayet

Quality Time Step	
Concentration	Concentration at Quality Time Step observed
AgeAverage	Average Age at Quality Time Step
AgeMinimum	Minimum Age at Quality Time Step
AgeMaximum	Maximum Age at Quality Time Step

Quality Duration	
ConcentrationMinimal	Minimal Concentration reached for duration of Quality saved
ConcentrationMaximal	Maximal Concentration reached for duration of Quality saved
AgeAverageMininimal	Minimal Average Age for duration of Quality saved
AgeAverageMaximal	Maximal Average Age reached for duration of Quality saved
AgeMinimumMinimal	Minimal Age Minimum reached for duration of Quality saved
AgeMinimumMaximal	Maximal Age Minimum reached for duration of Quality saved
AgeMaximumMinimal	Minimal Age Maximum reached for duration of Quality saved
AgeMaximumMaximal	Maximal Age Maximum reached for duration of Quality saved

Thermic Time Step	
Temperature	Temperature at Thermic Time Step observed

Thermic Duration	
TemperatureMinimal	Minimal temperature reached for duration of Thermic saved
TemperatureMaximal	Maximal Temperature reached for duration of Thermic saved

Attributes Tank Node

Data	
Active	True if ative / False if inactive
GroundLevel	Value of ground level
AchievementDate	Achievement Date
BaseLevel	Value of Base Level
EscapeLevel	Value of Escape Level
MinimumHead	Minimum Level of water in the tank
MaximumHead	Maximum Level of water in the tank
Volume	Volume calculated of the tank
Sector	Name of the sector
Service	Name of the service

Calculated	
SumDomesticUsers	Total of all users with type Domestic
SumIndustrialFlows	Total of flows with type Industrial
PeakFlowDuration	Peak Flow calculated on node for the duration of Zomayet's simulation
DomesticVolumeDuration	Volume consumed by domestic users for the duration of Zomayet's simulation
IndustrialVolumeDuration	Volume consumed by industrial consumers for the duration of Zomayet's simulation
VolumeDuration	Volume consumed for the duration of Zomayet's simulation

Opointe	
HeadOpointe	Water level for Opointe

Zomayet Time Step	
HeadTimeStep	Water level at the beginning of the observed time step of Zomayet
PiezometricTimeStep	Piezometric head at the beginning of the observed time step of Zomayet

Zomayet Duration	
HeadMinDuration	Minimum level reached for duration of Zomayet
HeadMaxDuration	Maximum level reached for duration of Zomayet
VolumeUsefulDuration	Useful Volume calculated on the duration of Zomayet
VolumeEnteringDuration	Total entering Volume calculated on the duration of Zomayet
VolumeLeavingDuration	Total leaving Volume calculated on the duration of Zomayet
VolumeOverflowDuration	Total overflowed Volume on the duration of Zomayet

Quality Time Step	
Concentration	Concentration at Quality time step
AgeAverage	Average Age at Quality time step
AgeMinimum	Minimum Age at Quality time step
AgeMaximum	Maximum Age at Quality time step

Thermic Time Step	
Temperature	Temperature imposed at Thermic time step

Attributes Resource Node

Data	
Active	True if active / false if inactive
GroundLevel	Value of ground level
AchievementDate	Achievement Date
MaximumAuthorisedFlow	Maximum Instantaneous Authorised Flow
DailyVolumeMaximumAuthorised	Maximum Authorised Daily Volume
Sector	Name of the sector
Service	Name of the service

Opointe	
HeadOpointe	Head of water for Opointe

Zomayet Time Step	
HeadTimeStep	Head of water at observed time step for Zomayet

Zomayet Duration	
VolumeEnteringDuration	Total entering Volume calculated on the duration of Zomayet
VolumeLeavingDuration	Total leaving Volume calculated on the duration of Zomayet

Quality Time Step	
Concentration	Average of Ending nodes Concentration at Quality Time Step
AgeAverage	Average of Ending nodes Average Age at Quality Time Step
AgeMinimum	Average of Ending nodes Minimum Age at Quality Time Step
AgeMaximum	Average of Ending nodes Maximum Age at Quality Time Step

Quality Time Step	
Concentration	Concentration at observed Quality time step

Thermic Time Step	
Temperature	Temperature imposed at Thermic time step

Attributes Pipe Link

Data	
Material	Material of the pipe
Length	Length
DiameterLocal	Local Diameter of the pipe link
HazenWilliamsLocal	Local Hazen Williams Coefficient of the pipe link
RoughnessLocal	Local Roughness of the pipe link
DiameterIntPipe	Interior Diameter of the pipe
DiameterExtPipe	Exterior Diameter of the pipe
HazenPipe	Hazen Coefficient of the pipe
RoughnessPipe	Roughness of the pipe
KineticClass	Kinetic Class of the pipe link
OrderKinetic	Ordre of the kinetic class
ConstantKinetic	Constant of the kinetic class
AchievementDate	Achievement Date
Sector	Name of the sector
Service	Name of the service

Calculated	
SumDomesticUsers	Total of all users with type Domestic
DomesticVolumeDuration	Volume consumed by domestic users for the duration of Zomayet's simulation
DifferenceLocalHazenPipe	Difference between local and pipe Hazen-Williams coefficients
DifferenceRoughnessLocalPipe	Difference between local and pipe Roughness coefficients
DifferenceDiameterLocalPipe	Difference between local and pipe interior diameter

Opointe	
FlowOpointe	Flow calculated by Opointe
VelocityOpointe	Velocity calculated by Opointe
UnitaryHeadLossOpointe	Unitary Head Loss calculated by Opointe
TotalHeadLossOpointe	Sum of Head Loss calculated by Opointe
Abs(FlowOpointe)	Absolute value of Flow calculated by Opointe
Abs(VelocityOpointe)	Absolute value of Velocity calculated by Opointe
Abs(UnitaryHeadLossOpointe)	Absolute value of Unitary Head Loss calculated by Opointe
Abs(TotalHeadLossOpointe)	Absolute value of Sum of Head Loss calculated by Opointe

Zomayet Time Step	
FlowTimeStep	Flow at observed Zomayet time step
VelocityTimeStep	Velocity at observed Zomayet time step
UnitaryHeadLossTimeStep	Unitary Head Loss at observed Zomayet time step
TotalHeadLossTimeStep	Sum of Head Loss at observed Zomayet time step
Abs(FlowTimeStep)	Absolute value of Flow at observed Zomayet time step
Abs(VelocityTimeStep)	Absolute value of Velocity at observed Zomayet time step
Abs(UnitaryHeadLossTimeStep)	Absolute value of Unitary Head Loss at observed Zomayet time step
Abs(TotalHeadLossTimeStep)	Absolute value of Sum of Head Loss at observed Zomayet time step

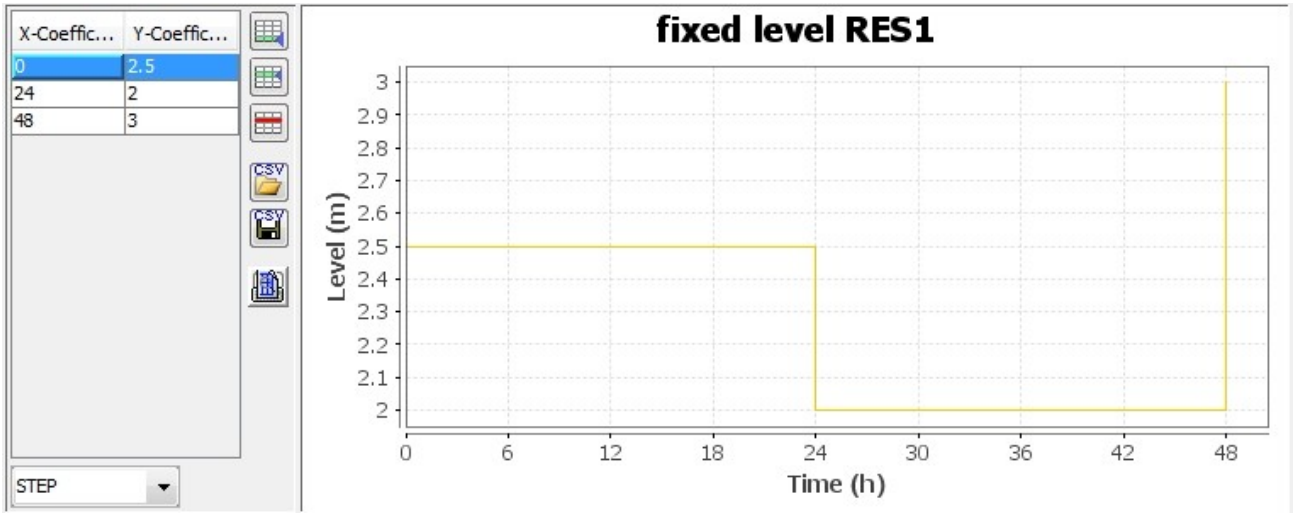
Zomayet Duration	
PeakFlowDuration	Peak Flow reached for the duration of Zomayet simulation
PeakVelocityDuration	Peak Velocity reached for the duration of Zomayet simulation
UnitaryHeadLossMaxDuration	Maximum Unitaru head Loss reached for the duration of Zomayet simulation
TotalHeadLossMaxDuration	Sum of Head Loss for the duration of Zomayet simulation
Abs(PeakFlowDuration)	Absolute of Peak Flow reached for the duration of Zomayet simulation
Abs(PeakVelocityDuration)	Absolute value of Peak Velocity reached for the duration of Zomayet simulation
Abs(UnitaryHeadLossMaxDuration)	Absolute value of Unitary Head Loss reached for the duration of Zomayet simulation
Abs(TotalHeadLossMaxDuration)	Absolute value of Sum of Head Loss for the duration of Zomayet simulation
SumPositiveFlowDuration	Sum of volume transported in the same direction as the pipe link the duration of Zomayet simulation
SumNegativeFlowDuration	Sum of Volume transported in the opposite direction of the pipe link for the duration of Zomayet simulation
BalanceFlowDuration	Balance of the two volumes above

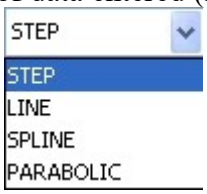
Special	
AverageNodesValues	Average Value choose for color parameter of ordinary ending nodes

Profil with tables and graphs

A number of dialog boxes require the entering of graphs that are functions of time or other abscissa. A “panel” allows curves to be constructed, either point-by-point or by importing a csv file. It is composed of 2 data entry fields: one text and one a scrolling list. Below the space is split into four vertical sections, from left to right:

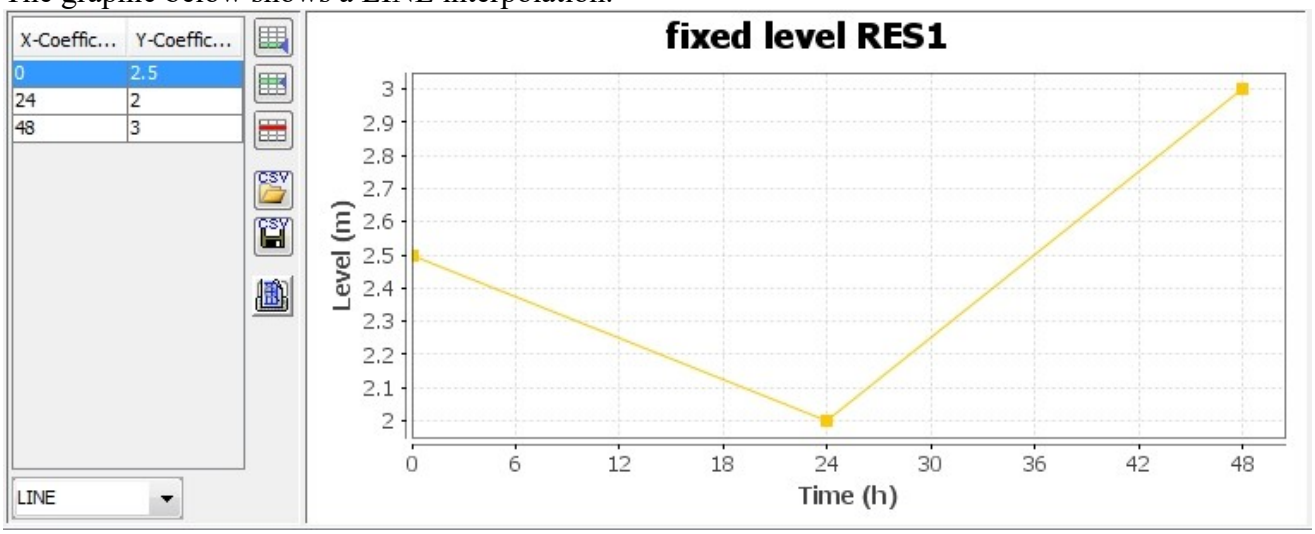
- The table of values
- A button bar for the management of the table
- The graph representing the entered values and their interpolation



- The field here named “Imposed level” allows a name to be given to the profile, its name varies with the type of data entered (level, head, concentration etc.).
- The scroll list  allows the type of interpolation between the entered points to be chosen.

- STEP – the value remains constant between two points.
- LINE – the value is interpolated linearly between two points.
- SPLINE – a smoothing function passing by all points is applied.
- PARABOLIC – a parabola is calculated from three points, this function is **required** for the graph of pumps with three points.

The graphic below shows a LINE interpolation.



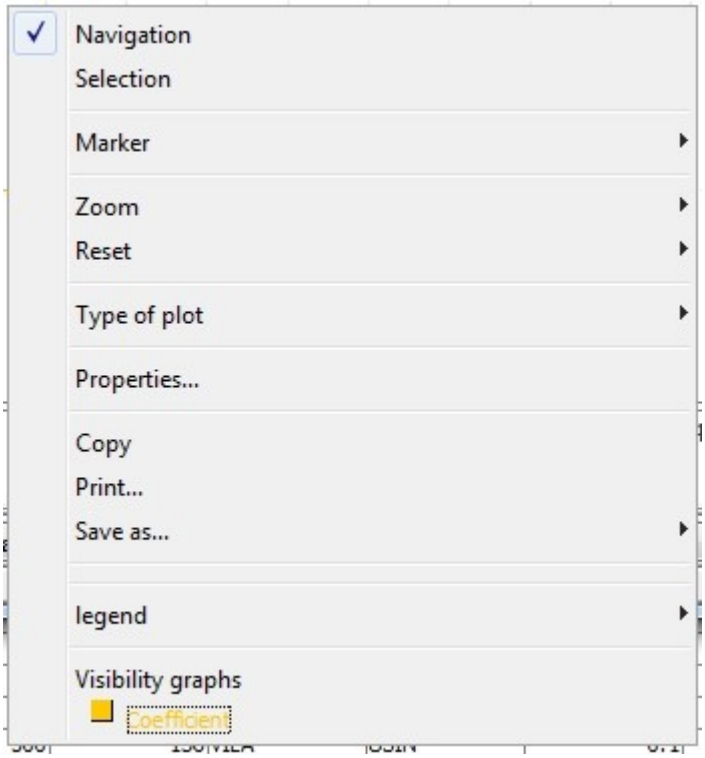
Description of the Table

Moving within the cells can either be done by the mouse or the keyboard. <TAB> and <SHIFT + TAB> moves from cell to cell. The up and down arrows allow moving from line to line.

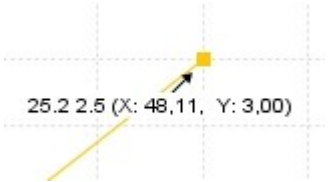
- adds a line at the end of the table
- adds a line above the selected line in the table
- deletes the selected line in the table
- imports a table in CSV format
- exports the entered table in CSV format
- allows print preview of table

Chart

The graph represents the interpolated data from the table. A right-click on the chart area shows the pop-up menu.



A checkbox allows to determine the current function, here **Navigation** allows to move the area of points in the screen in all the directions by clicking and holding the left mouse button.

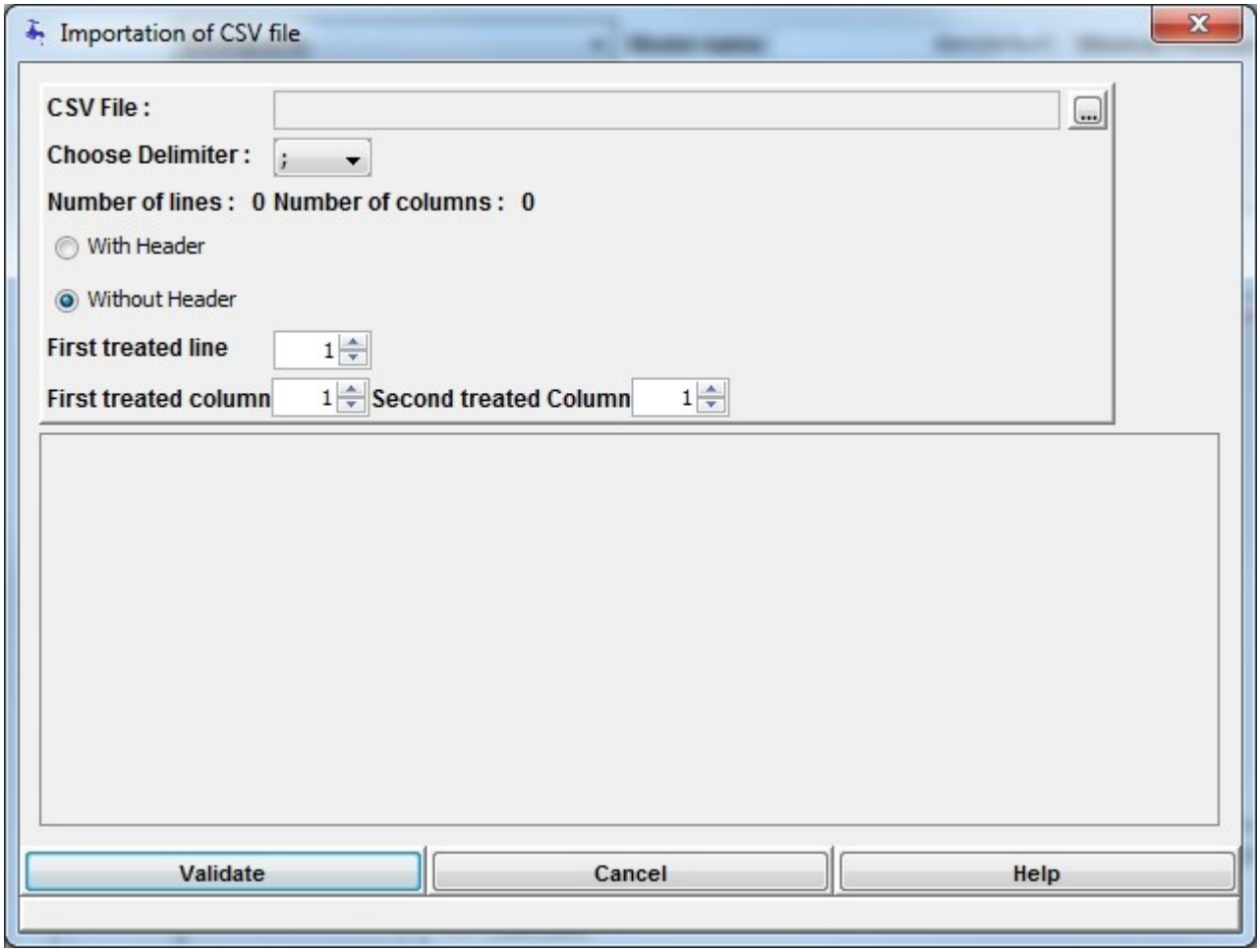


The function **Selection** allows to know by left-click at a point its coordinates.
The function **Marker** is used to draw a horizontal and/or vertical guideline following the mouse.
The function **Zoom** by clicking and holding the left mouse button allows to zoom either along the axis of abscissa, or along the axis of ordinates, or by drawing a rectangle defining the area to be zoomed.
The function **Reset** allows to return to the original zoom, either on an axis, or on both.
The function **Plot type** allows to change the plot shape of the curve, it is advised to leave the selected mode corresponding to the type of curve (Step, Line, ...).
The function **Properties...** opens the Properties dialog of the plot area properties to change the background or the color of the axes, for example.
The function **Copy** allows to copy the graph to the clipboard in bitmap mode.
The function **Print...** starts the Print dialog of the graphic area with choice of paper and margin size.
The function **Save as...** allows to save the graphic area to a file in PNG, SVG, PDF, EPS and JPG format.
The function **Legend** allows to show or hide the legend. The function **Graphics visibility** allows, for left-click on the color representing a series, to show or hide it. This function is not active in all graphics.

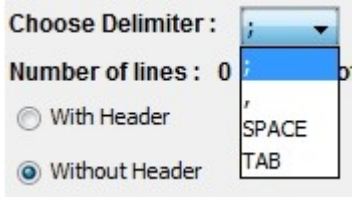
Import of a table in CSV format

- import a table in CSV format

Pressing the button for CSV file importation launches a procedure for selecting a file and choosing the data to import.



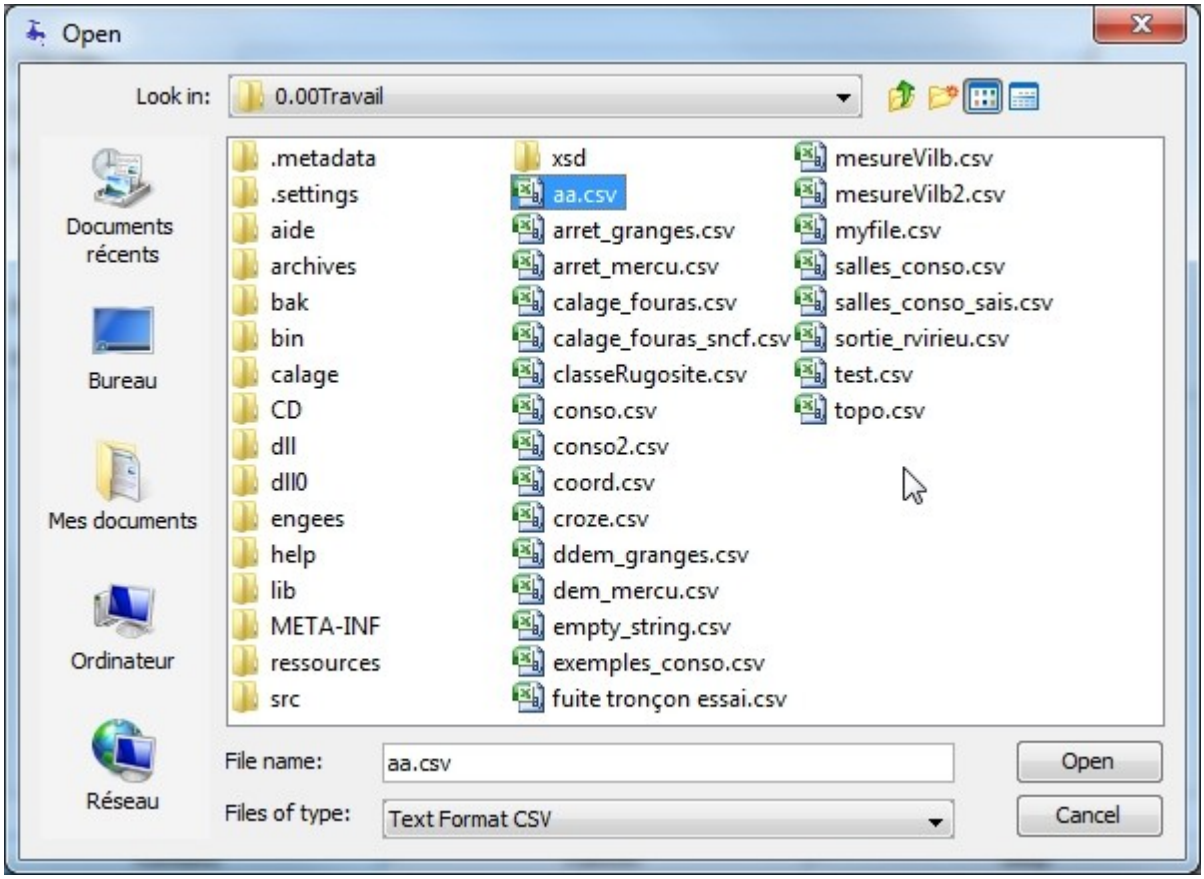
The field separator used can be changed from the scrolling list “delimiter choice”. If a file has already been opened and treated after changing the delimiter, it must be reopened to be treated with the new



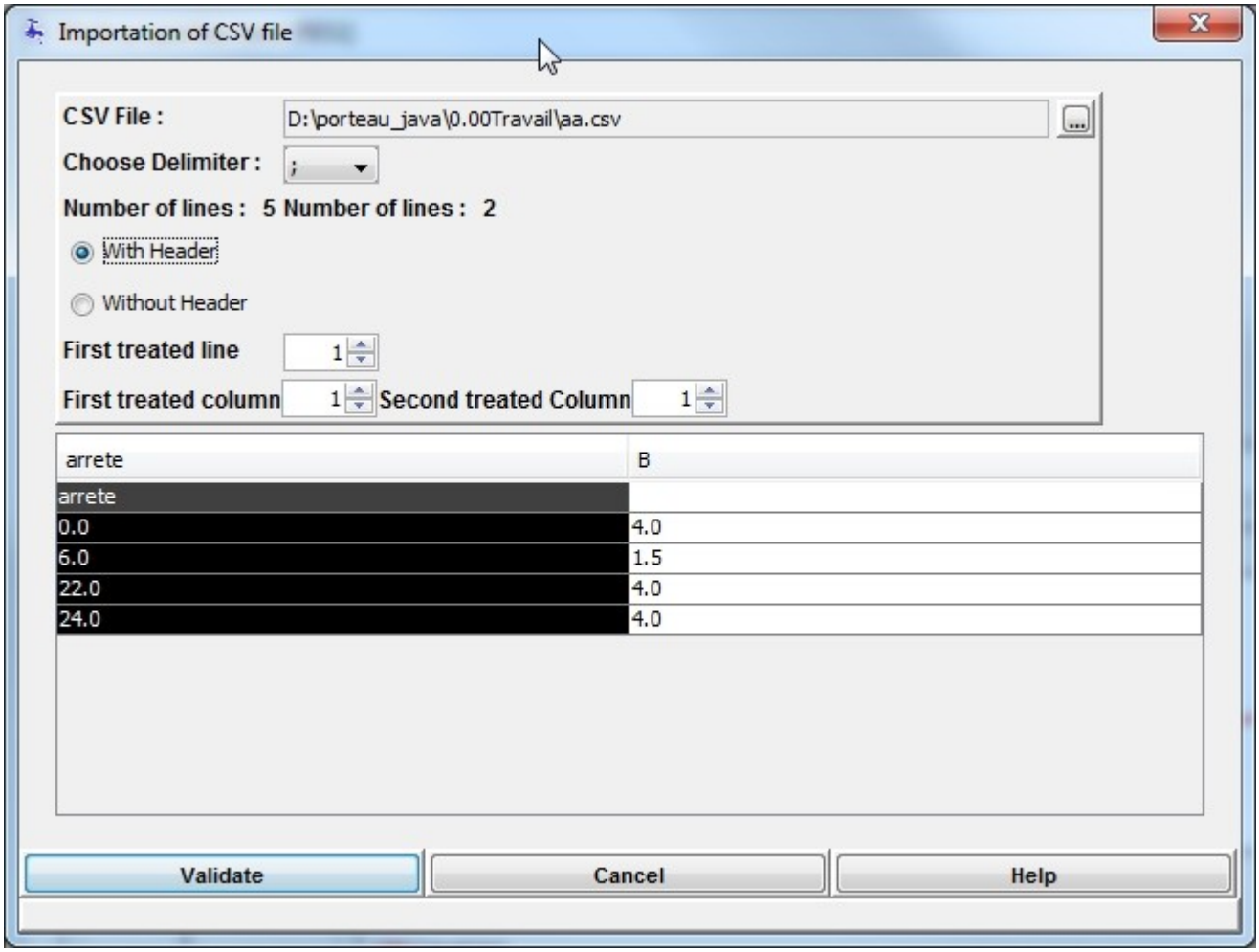
choice. The possible separators are:

- <;> the semi-colon
- <,> the comma
- <SPACE> the space
- <TAB> the tab.

To select the file to treat, click on the button  .

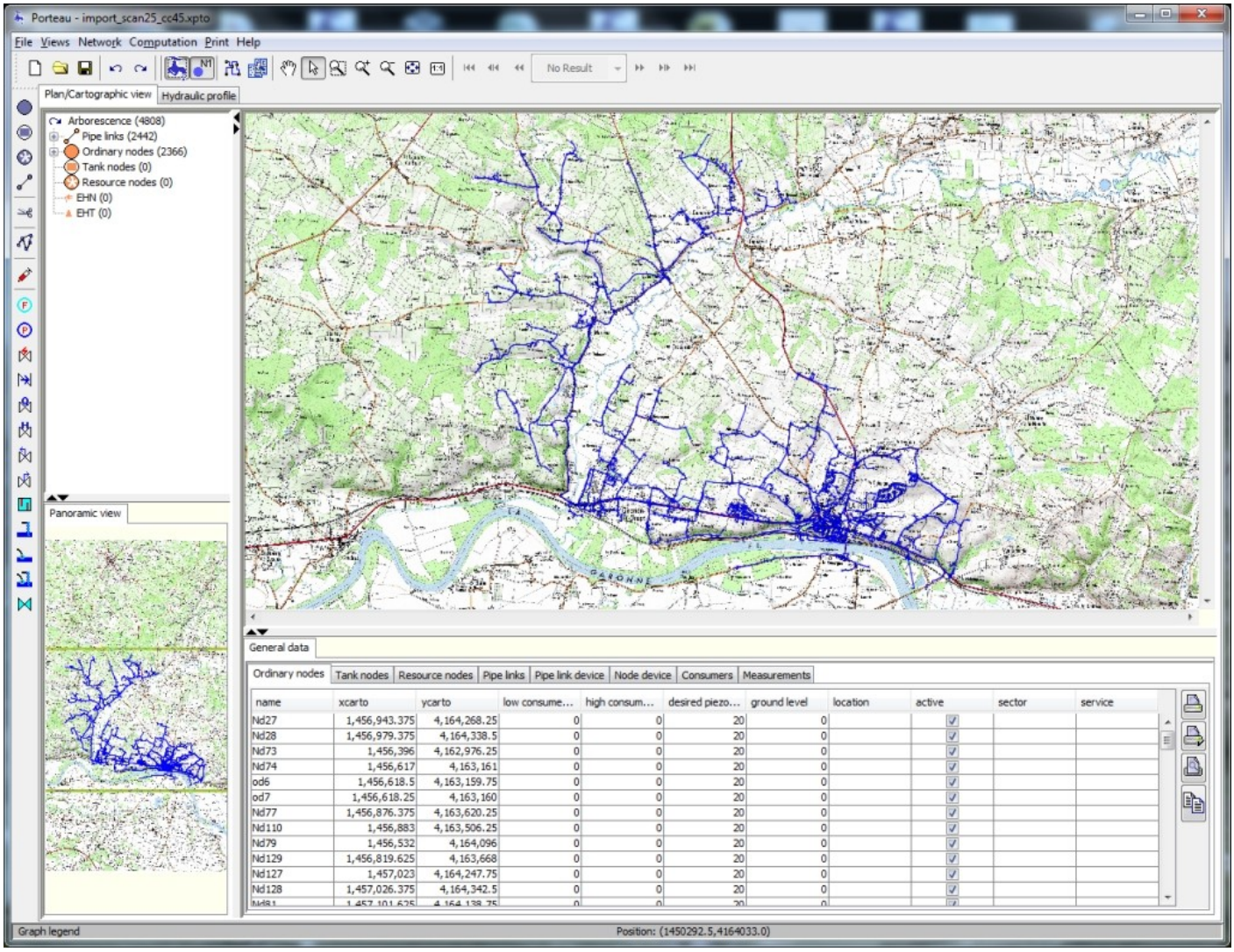


After validating the choice, the file is read and treated with the chosen delimiter.
It is possible to indicate whether the file contains any header lines by clicking on one of the buttons “With Header” or “Without Header”.
With header, the first line appears as headers for the columns selected.



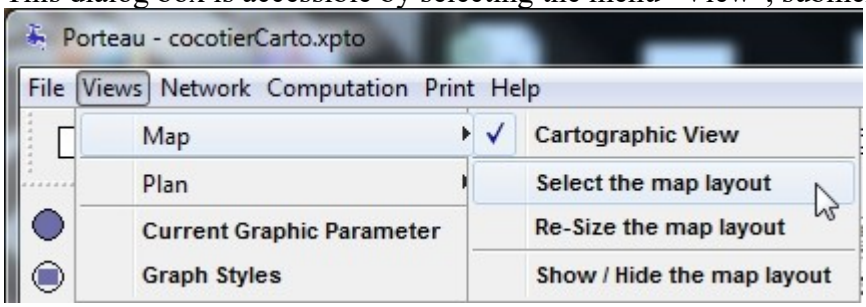
Next, it is necessary to choose the first line of data which will act as the starting point for its treatment. This is entered in the field “first line treated”. The treatment is performed until the end of the file.
The field “first column treated” designates the data to be used for the abscissa.
The field “second column treated” designates the data to be used for the ordinate.

Background Map

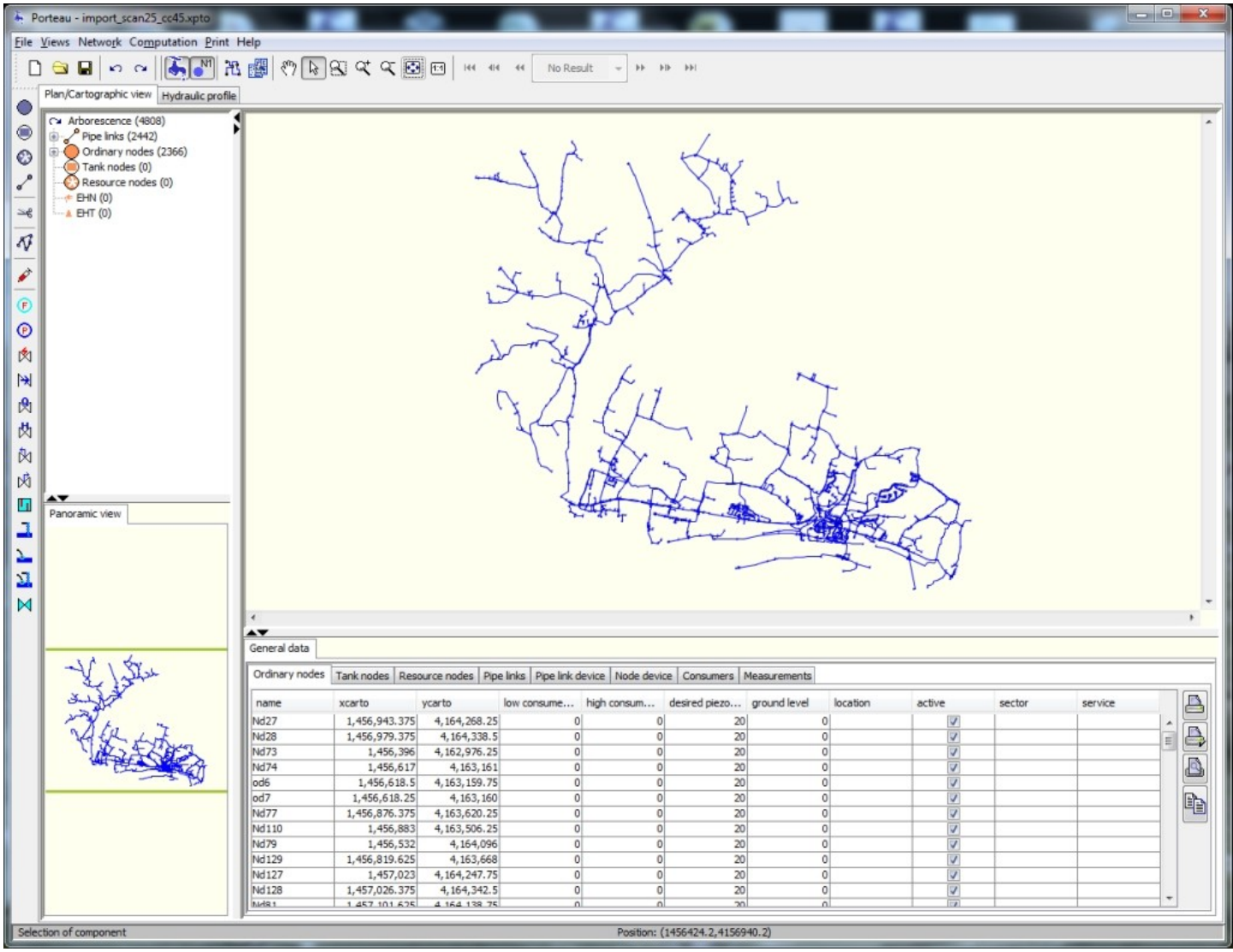


Select the background map

This dialog box is accessible by selecting the menu “View”, submenu “Select the background map”.



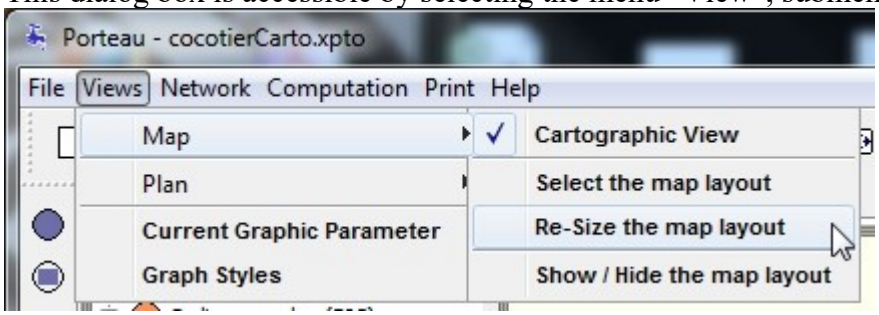
Warning, the map view must be active for this to work.
A dialog box for opening image files (JPG, GIF, PNG) allows the choice of image to be used as the background.



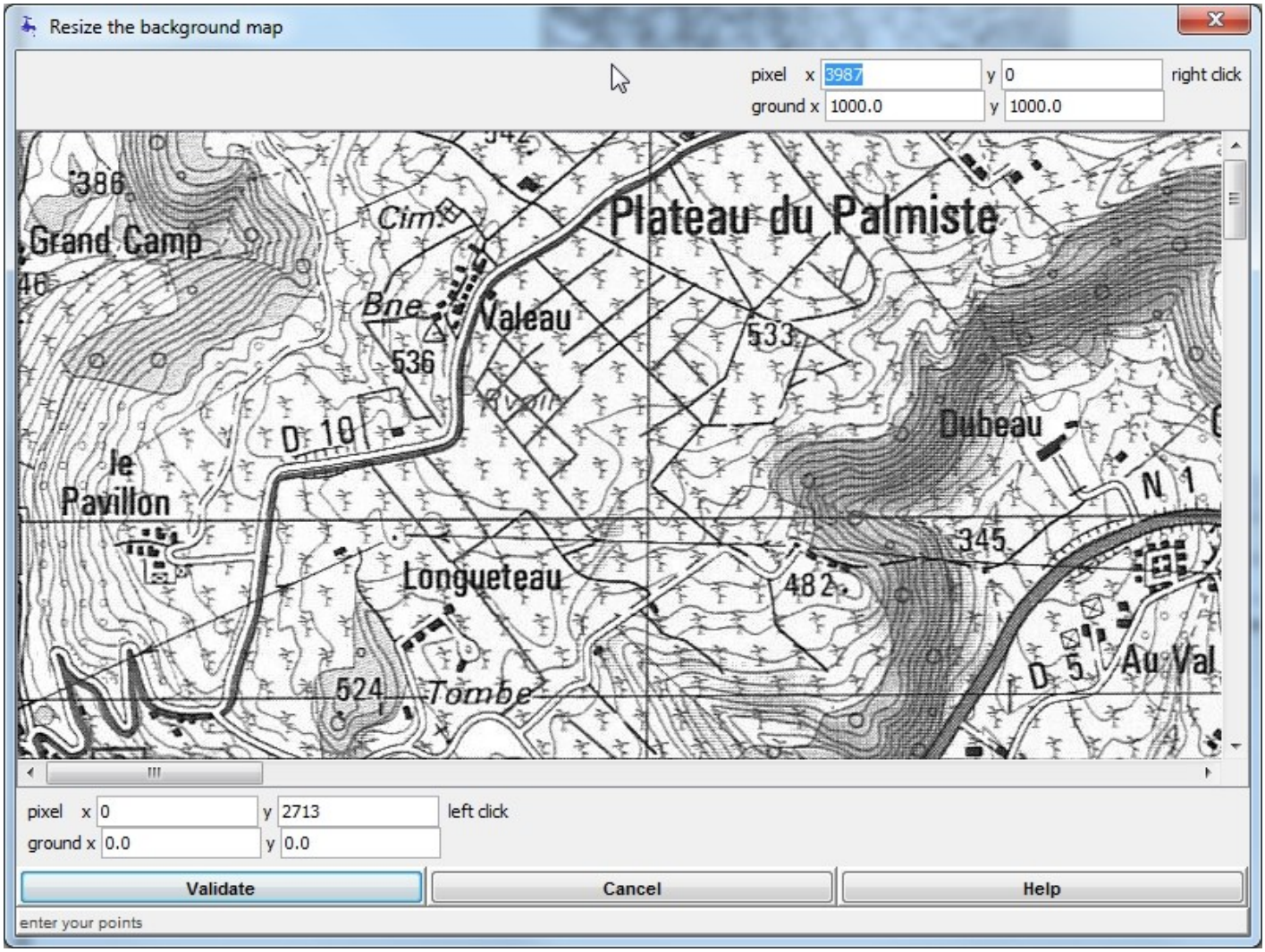
After validation, the image is read and sized by default as a 1000 by 1000 unit (metres) square with the **left lower corner 0,0**, as shown above by the small grey square at the bottom of the Map View.
The background map can be showed or hidden selecting the corresponding menu.

Resize the background map

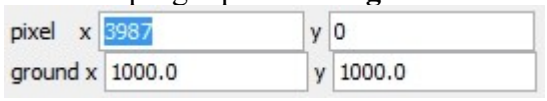
This dialog box is accessible by selecting the menu “View”, submenu “Resize the background map”.



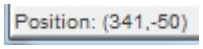
Warning, the map view must be active for this to work. The following dialog box appears. Warning, in the case of large images, if the memory available is insufficient, the display may be interrupted.



It allows the entering of two points of pixel and map coordinates: one for the bottom left and the other for the top right to have enough space between the two to rescale adequately in both X and Y.
To enter the values, the user either types the limits directly in the fields or clicks on points in the image with the mouse.
For the top right point the **right click** on the mouse is active; the values appear in the fields



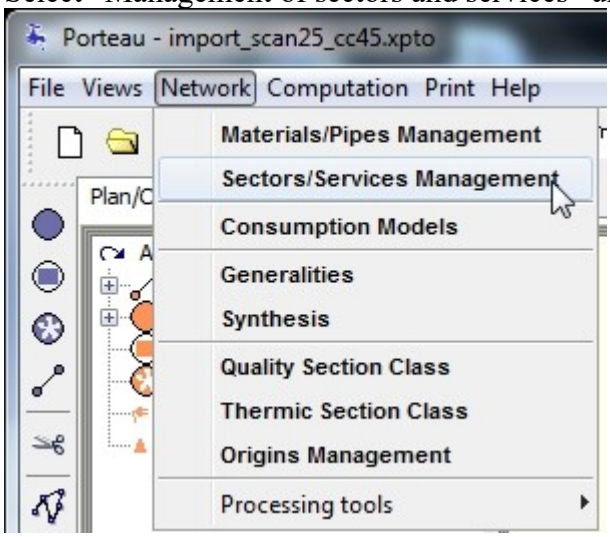
for the pixels, the user then enters the map coordinates corresponding to the point clicked.
For the bottom left point the **left click** on the mouse is active; the values appear in the fields

for the pixels, the user then enters the map coordinates corresponding to the point clicked.
The position of the mouse in pixels is always shown in the status bar of the dialog box 

.
After entering and validating the values, the Map View is updated under the network.

Sectors and Services

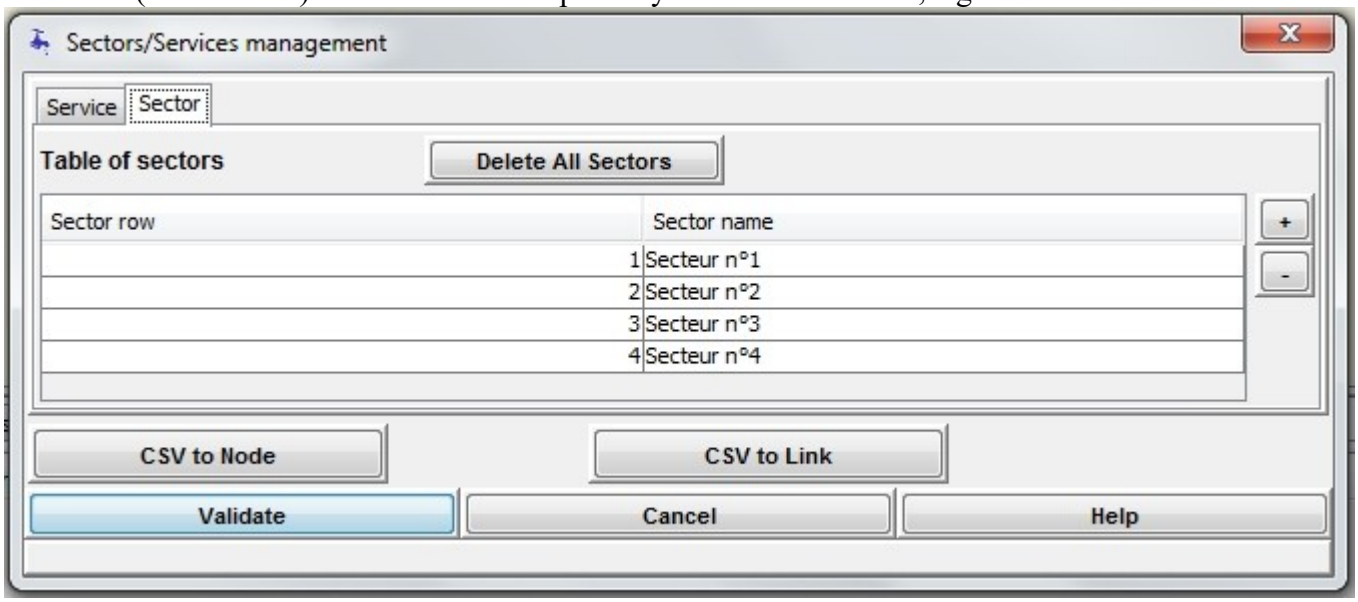
Select “Management of sectors and services” under the menu “Network”.



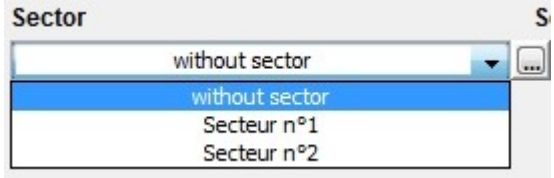
Two dialog boxes in two tabs are opened in order to enter the information.

Sectors

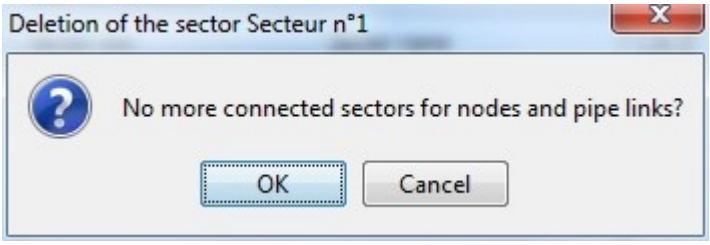
A sector is a geographic notion. You can define as many sectors as necessary to describe the network. Each node or pipe can be associated to any particular sector. Each sector is described by its order number, attributed automatically after its creation (not modifiable by the user) and its name. A value by default is attributed (Sector no. x) that the user can replace by the name of choice, e.g. “Secteur N°1”.



The identification of the sectors chosen in the column “Sector name” are visible in the scrolling list, allowing the choice of sector for an element in the network, as below:



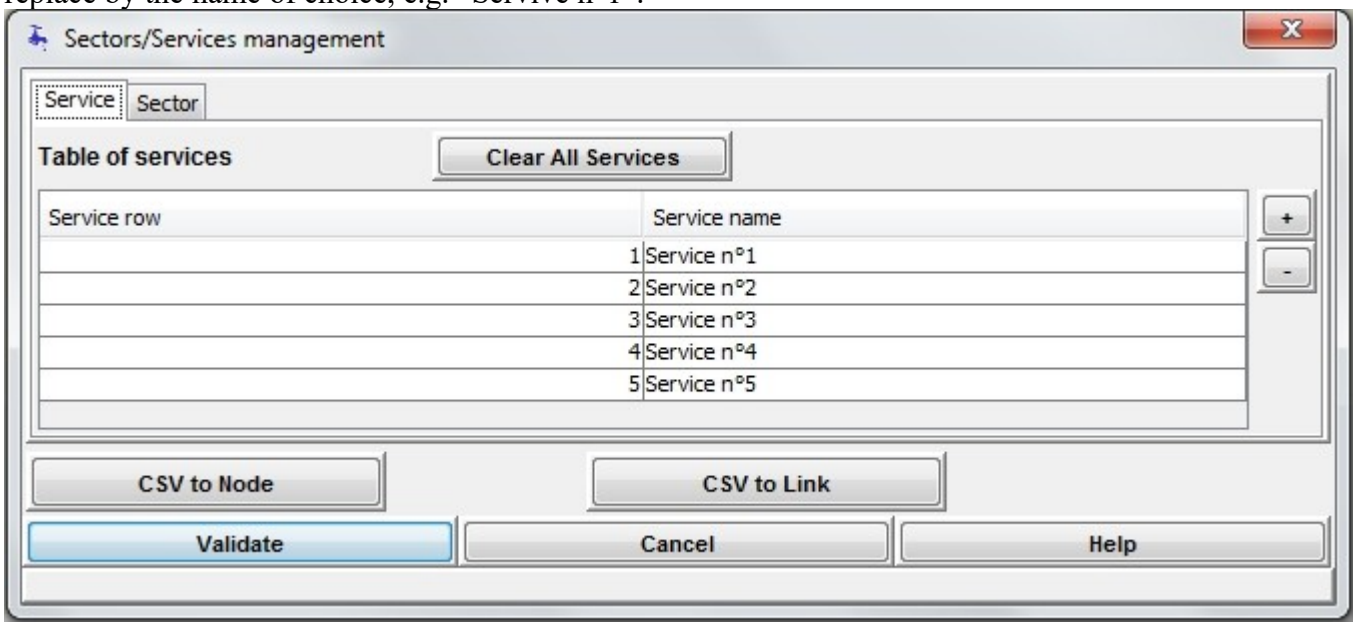
The  and  buttons allow sectors to be added or deleted. Before a selected sector is deleted, the following dialog box is shown:



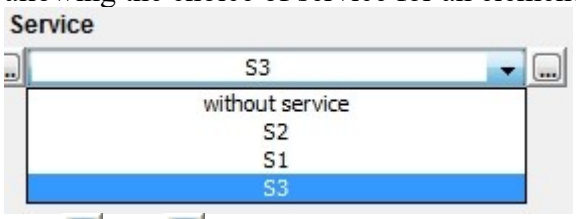
Note that only the affected sectors of model elements (nodes and pipes) can be deleted if you confirm the deletion (by choosing OK). The elements concerned will have no sector associated and will therefore have the attribute “without sector”.
Click on the button "Delete All Sectors" clear all sectors for all nodes and links.
Click on the 2 buttons CSV to Node (Link) is described in a following paragraph.

Services

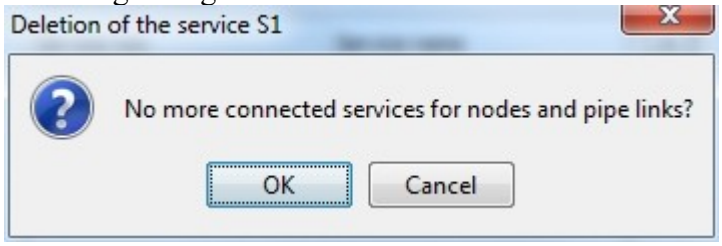
A service is also a geographic notion but applies to elements with the same piezometric head. You can define as many services as necessary to describe the network. Each node or pipe can be associated to any particular service. Each service is described by its order number, attributed automatically after its creation (not modifiable by the user) and its name. A value by default is attributed (Service no. x) that the user can replace by the name of choice, e.g. “Servive n°1”.



The identification of the services chosen in the column “Service name” are visible in the scrolling list, allowing the choice of service for an element in the network, as below:

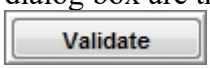


The + and - buttons allow services to be added or deleted. Before a selected service is deleted, the following dialog box is shown:



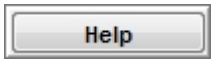
Note that only the affected services of model elements (nodes and pipes) can be deleted if you confirm

the deletion (by choosing OK). The elements concerned will have no service associated and will therefore have the attribute “without service”.
Click on the button "Clear All Services" clear all services for all nodes and links.
Click on the 2 buttons CSV to Node (Link) is described in a following paragraph. At the bottom of the dialog box are three buttons:



to save and exit the dialog box

to cancel and exit the dialog box

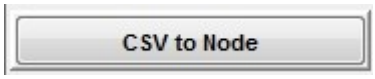


to obtain help.

Import Sector-Service by CSV

To import Sector and/or Service on Node or Link two buttons are available.

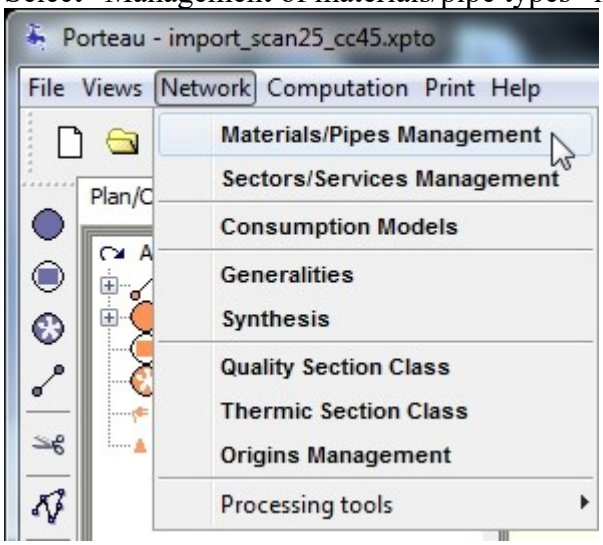
imports a CSV file with 3 columns separated by ";":
1 - Text identifing the Link described by the two ending nodes "NodeA->NodeB" or "NodeA_NodeB"
2 - Text name for the sector (can be empty but separated from next by ;)
3 - Text name for the service (can be empty).



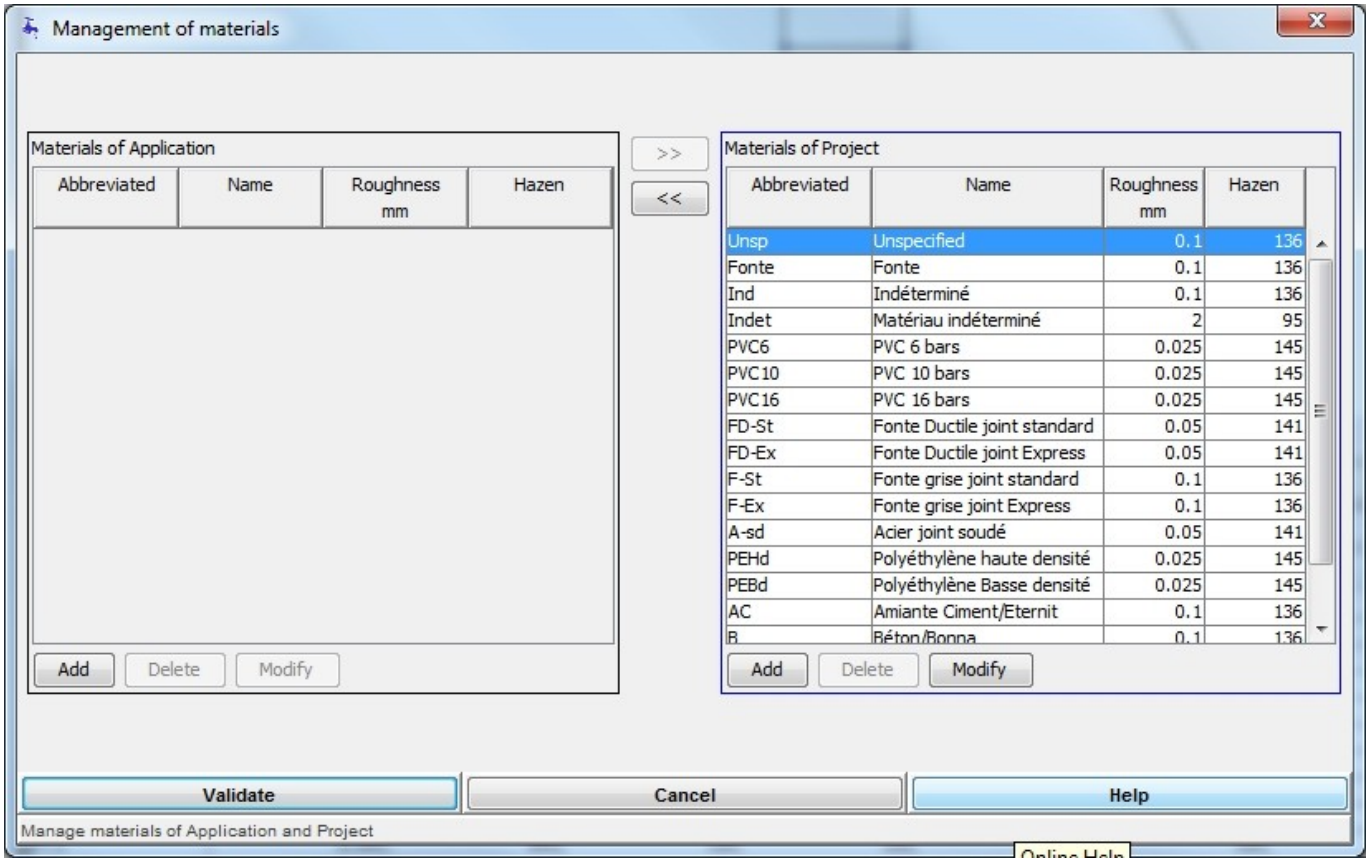
imports a CSV file with 3 columns separated by ";":
1 - Name of the Node
2 - Text name for the sector (can be empty but separated from next by ;)
3 - Text name for the service (can be empty).

Materials & Pipe types

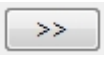
Select “Management of materials/pipe types” in the menu “Network” (keyboard shortcut: <Alt>+R+M)



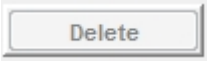
The dialog box “Management of materials” is then shown.

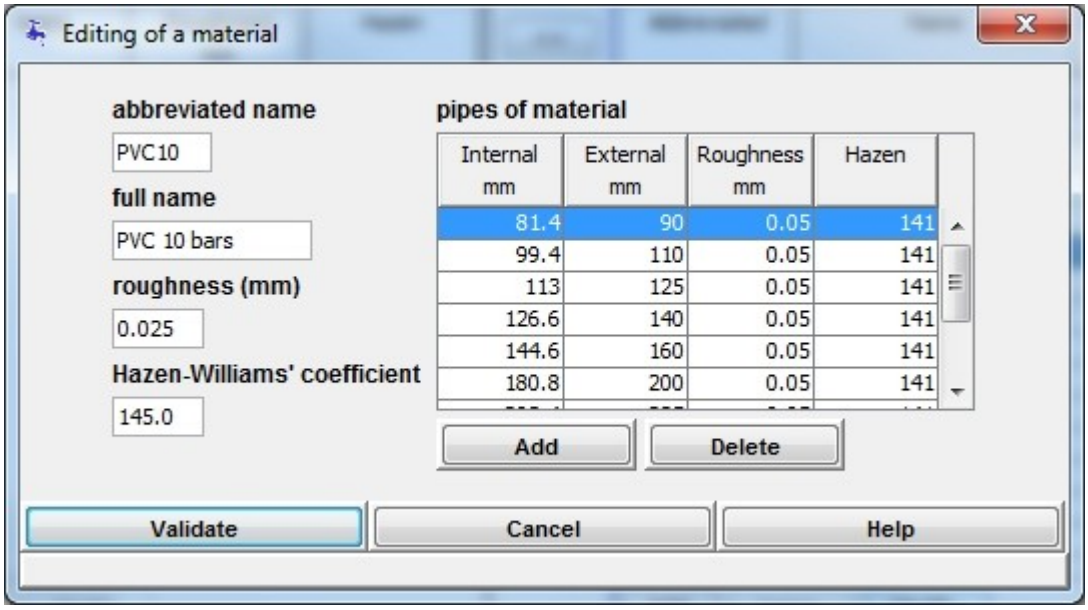


Here, a distinction is made between a list of materials for the application (consisting of a catalogue of generally usable materials) and a list of materials for the project (consisting of materials used in the current project).

These items can be moved from one table to the other by using the buttons in the centre,  or  .

Each of the tables has a button  allowing a new material to be added at the end of the list.

The button  allows a material to be deleted on condition that no pipe type exists referencing this material. If this is the case, the button will be greyed out  . The panel below appears when the button  is clicked on a selected line:



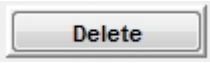
It allows data for attributes associated to the material to be entered on the left:

Material's data : the**abbreviated name** to be entered in the editing zone, e.g. PVC16
the **full name** to be entered in the editing zone, e.g. PVC 16 bars
the **roughness** (Colebrook), e.g. 0.025
the **Hazen Williams** coefficient, e.g. 145.0

The table on the right contains the list of pipe types associated with this material. With the following data attributes that can be entered directly in the table:

Pipe's data : **internal diameter** in mm
external diameter in mm. This field is strictly for information only, it is not used in the roughness calculations.
rugosité en mm
Hazen Williams Coefficient

The button 

allows a new pipe type to be added that initially takes the following default values: 100 mm for the internal diameter, 105 mm for the external diameter and roughness and Hazen-Williams coefficient according to the material default.
The 

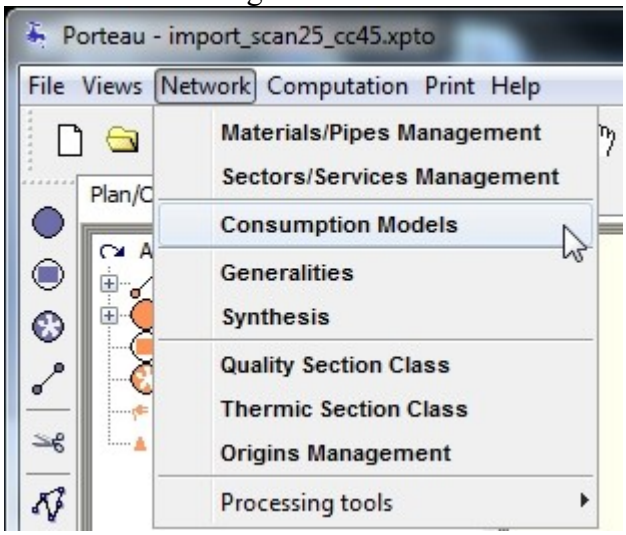
button deletes a pipe type from the list.
The three buttons at the bottom of the screen,


function as with all the data entry dialog boxes.
If pipe parameters used in the network are changed, Porteau will suggest injecting these new values on the corresponding sections, in particular for the internal diameter and for the coefficients of linear headloss. The answers to the possible choices will package following treatment. "To inject" injects the displayed parameter, and moves to the next for a new question. "NOT to inject" does not change the pipes corresponding to the displayed value and advances to the next for a new question. "To inject ALL" injects all changes without asking new questions. "To inject ANY" injects no changes without any new question.

Models of consumption

Common part to all types of "models of consumption"
The Porteau software is based on a demand-driven model. Consumption models are used to represent the consumption of network users. Note that leakage or exchange between networks can also be represented

with the help of consumption models called “localised leakage” or “exchange” at nodes. For pipes, there can only be distributed leakage model.
These models of consumption are used during the specification of the nodes and pipes. At each node it is possible to apply different models of consumption. On a pipe it is only possible to apply domestic consumers model.
Access to the dialog box for the models of consumption



In the creation **no** model is created by default.
Models of consumption in water distribution systems:
There are 4 types of models of consumption:

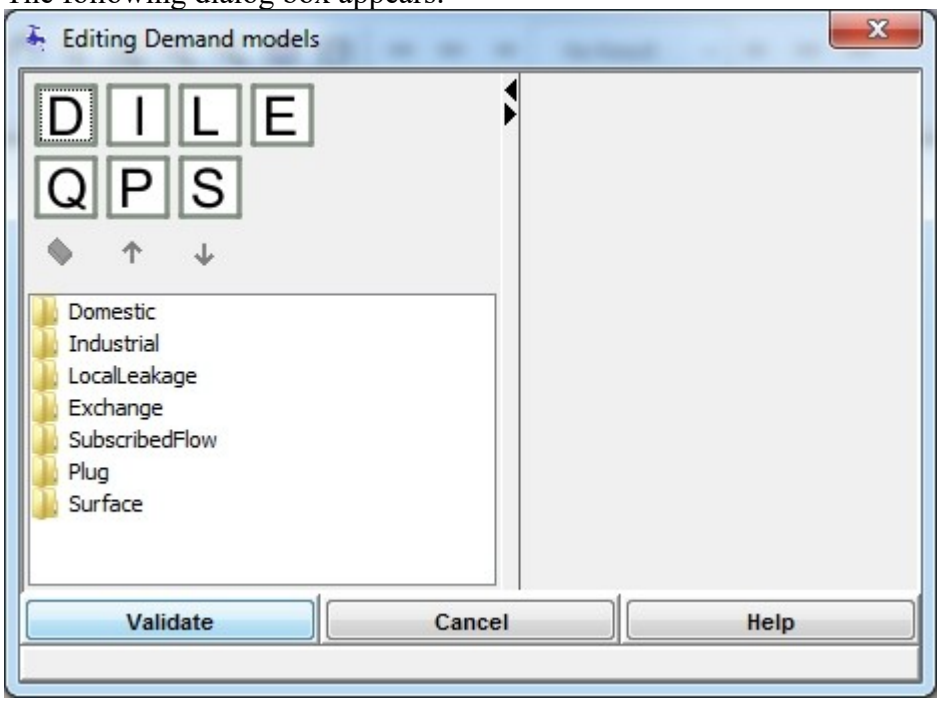
- ✓ **Domestic consumptions**: these models characterise domestic use. It is necessary to introduce the number of consumers (n) at the node level.
- ✓ **Industrial consumptions**: these models characterise factories, offices or other non-domestic consumption. They can also characterise a whole network under the condition that the consumed flows are known hour by hour and thus the total flow. In the case of industrial consumption, it is necessary to introduce the peak hourly flows in l/s at the node level. This flow is considered as that of the Opointe calculations. Industrial consumption under Opointe is thus deterministic whilst domestic consumption is probabilistic (statistical law described by three coefficients, see later for further detail).
- ✓ **Localised leakages**: these leaks are located on ordinary nodes. Their values are fixed by the user at each place where they are identified. Each model of localised leakage is unique and attributed to the node where the leak occurs (representation of the output of an agglomerated sector and a node).
- ✓ **Exchanges**: these exchanges are often situated at the end of antennae. The model of flows given to or provided by neighbouring networks is deterministic. The associated model is composed of a temporal profile of measured or estimated values. It must be able to accept algebraic values (positives, negatives or nil). A coefficient multiplier allows projections to be made in the future and applies to the temporal profile. In the case of importing from a neighbouring network, for quality monitoring it is necessary to know the temporal profiles of residence time and concentration. If there is an export of water to a neighbouring network, the quality calculation is classic. In the case of a mixed exchange, it is only necessary to enter the data for the importation.

The models of consumption with irrigation:
There are 3 types of models of consumption with Irrigation:

- **Subscribed flows** with flows rule's profile of downstream subscribed flows (model with no-differentiated plugs)

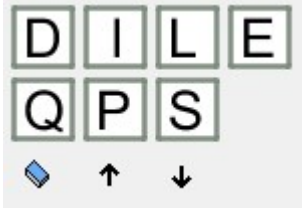
- **Irrigated areas** with flows rule's profile of downstream area flows (model with no-differentiated plugs)
- **Watering plugs** with flows rule's profile of downstream watering plugs flows (model with differentiated plugs)

The following dialog box appears:

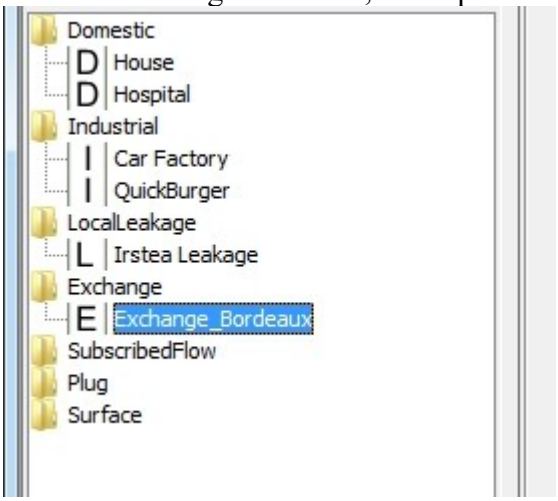


It contains three zones:

- The management buttons of network models.

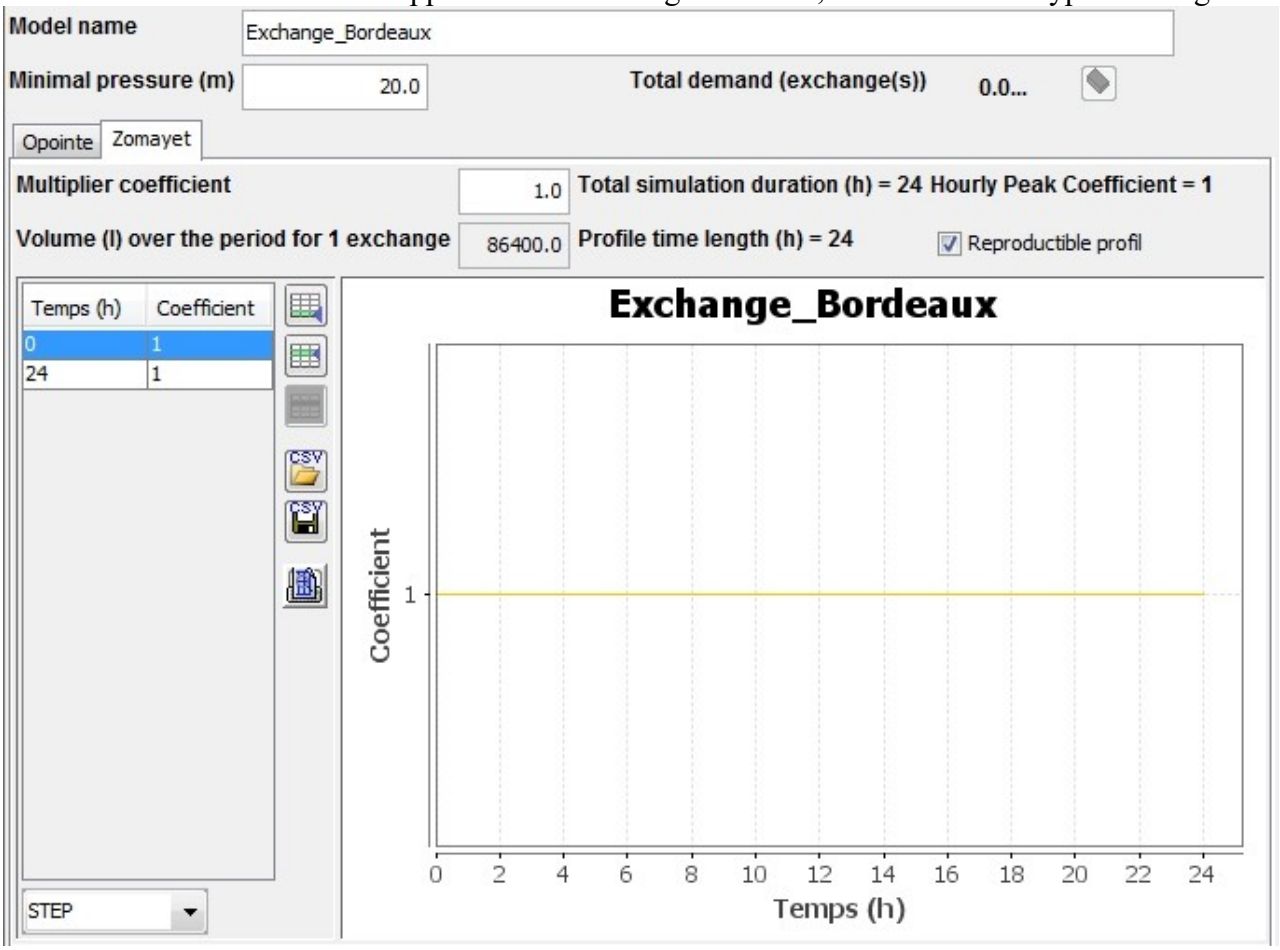


- A tree describing all models, a complex example here already entered.



the edition of a model is done by selection in the tree.

- The model selected in the tree appears then in the right window, here a model of type exchanges.

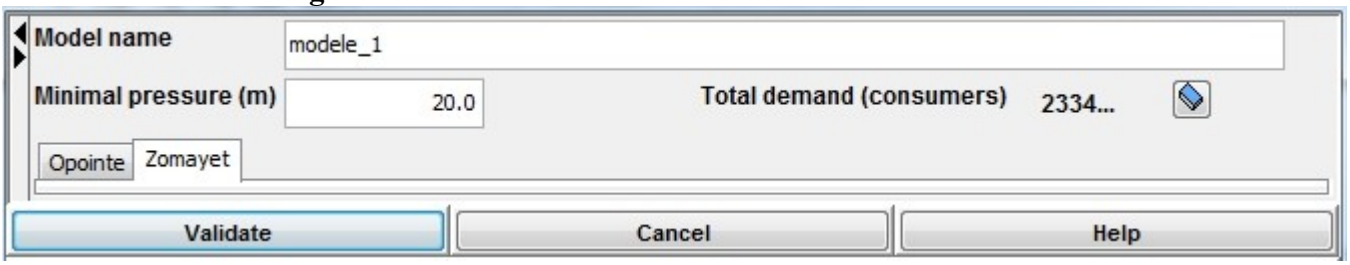


The button bar for the management of the tree contains the following buttons:

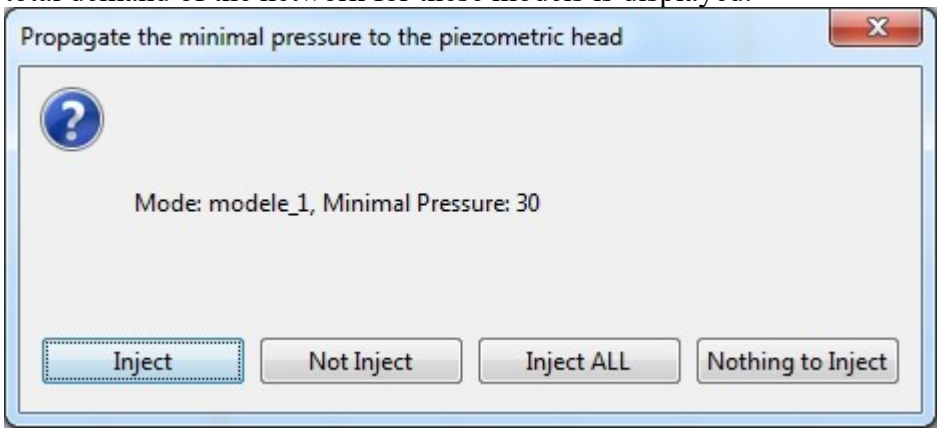
- D add a model of domestic consumption
- I add a model of industrial consumption
- L add a model of localised leakage
- E add a model of exchange
- Q add a model of subscribed flow of irrigation
- P add a model of watering plug of irrigation
- S add a model of irrigated area
- delete the selected model if it has no associated water withdrawal
- ↑ up the selected line in the tree to convert the model with entered consumptions
- ↓ down the selected line in the tree to convert the model with entered consumptions

Depending on the selected row, the buttons are active or not, to show the possible actions.
Let us see first the common zones of edition.

Common zone of dialogue boxes



In the common zone for all models, a name and a desired minimum pressure are required.
The name of the models which are already created can be changed. This name has to be unique.
In the cases of the models are used at the nodes and at the pipe sections for the distributed demand, the total demand of the network for these models is displayed.

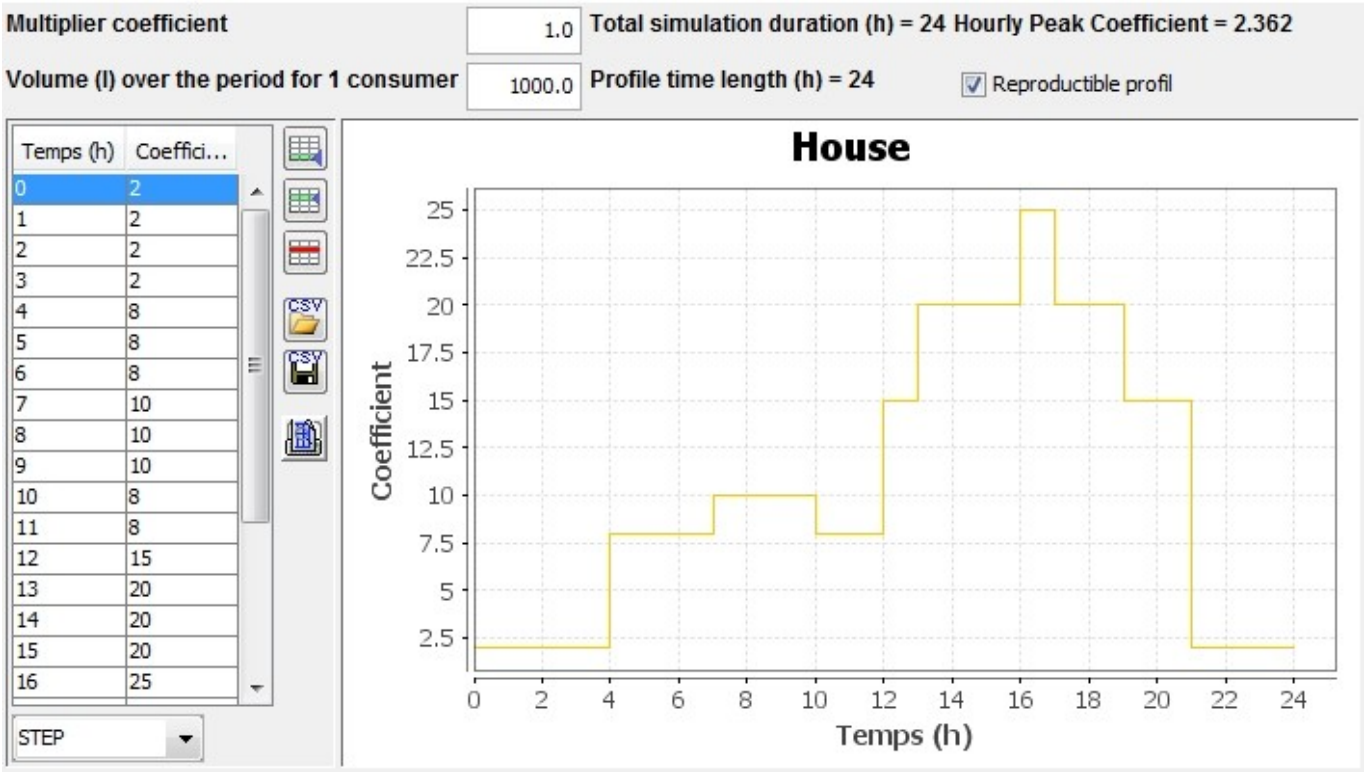


If the desired pressure is changed, all the nodes of the network carrying the type of consumptions can see its desired minimum piezometric impacted or not, depending on the choice of the user. Indeed, when clicking on the OK button, all the minimum pressures are inspected and those which are modified impacting the nodes engender the display of a dialog box suggesting to inject their values or not, either model by model by choosing "to inject" or "not to inject", either all the models at once without asking the question and by choosing "to inject everything" or "to inject nothing".
The gum allows to delete **all** consumptions attached to the selected model; be careful, this operation is not cancellable.

Zomayet

Temporal profiles for these models should be based on a simulation in hours. In the case of calculations longer than the temporal profile, it can (or not) be automatically reproduced by checking the box Profil reproductible .
In addition, for all the types of model, it is necessary to describe data specific the duration of simulation, the duration of the profile being entered, the hourly peak coefficient of the profile. These values are updated only to validate the data being entered. The Zomayet module allows a simulation over 24 hours or several days of the behaviour of a meshed distribution network or the transportation of water under pressure by calculating the flows and pressures at every hour of the day.
This module adopts a deterministic approach that signifies that the user demand (domestic or industrial) is described by time step, during a period hour by hour over the entire day, with a curve, in the window defining the consumer models.

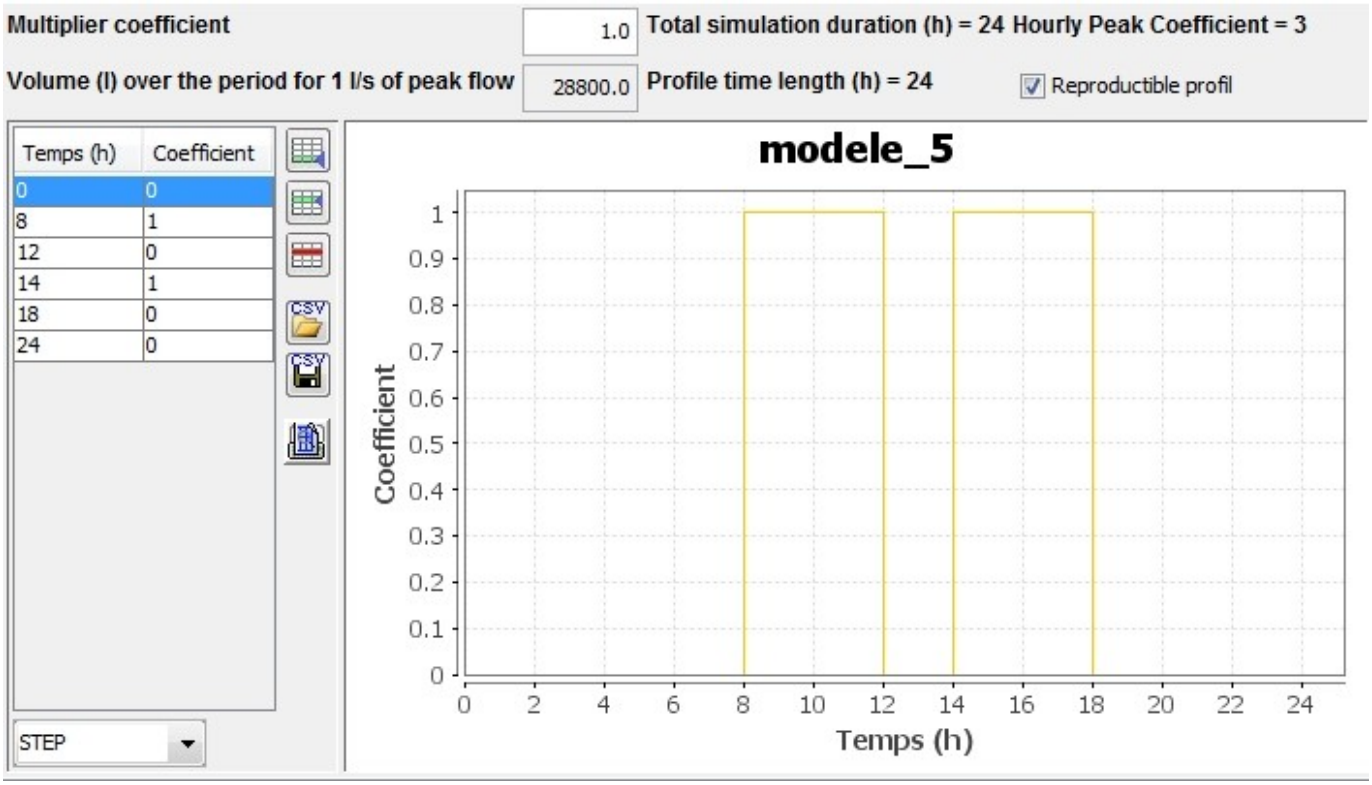
Domestics consumers



Domestic data : **Coefficient multiplier** applied to the number of consumers. Default value: 1.0.
Non-dimensional temporal profile of volume modulation.
Volume in liters for 1 consumer entered over the temporal profile duration.

The given Volume is distributed along the surface described by the curve, Zomayet calculates the surface attributed to the time step and affect the proportion of the total volume assigned to the length of the curve.

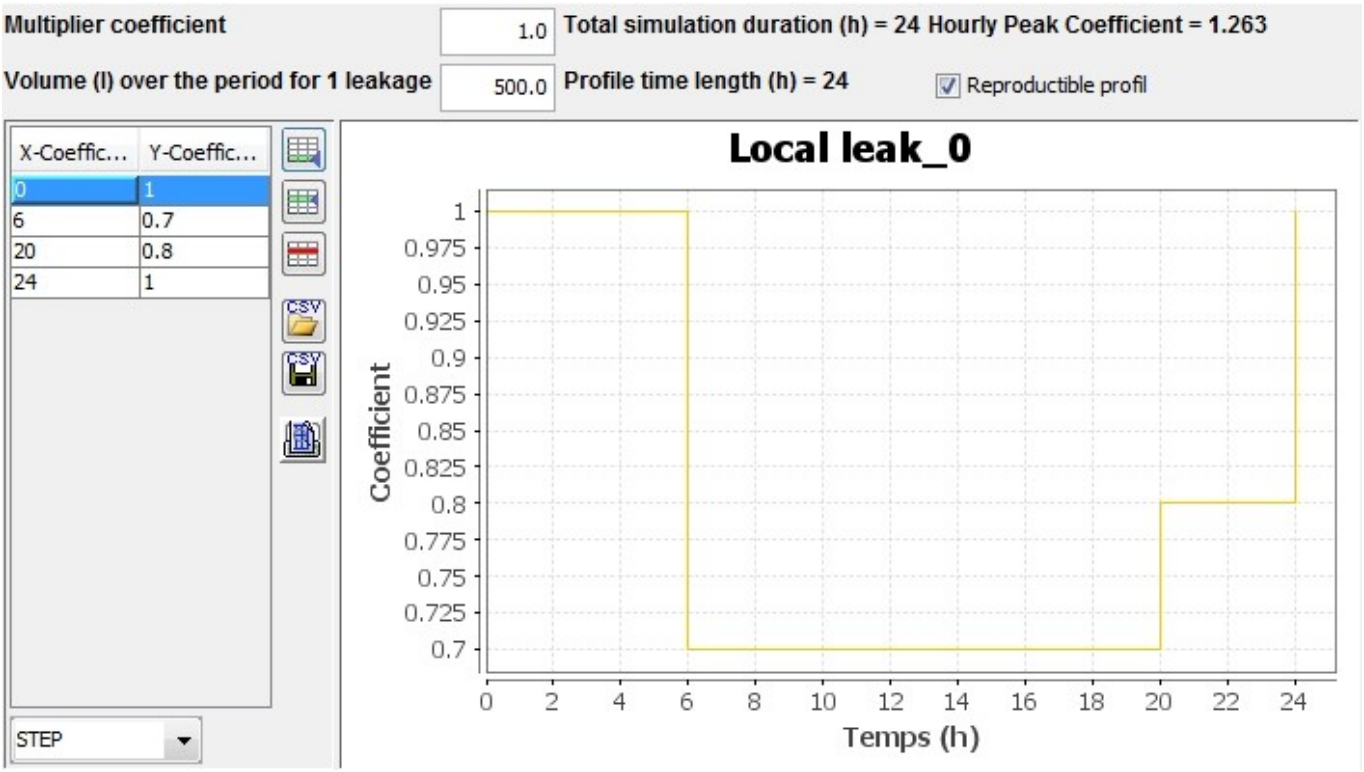
Industrials consumers



Industrial data : **Coefficient multiplier** applied to peak flow. Default value: 1.0.
Non-dimensional temporal profile of weighting of the peak flow.
Volume in liters for 1L/s entered over the temporal profile duration.

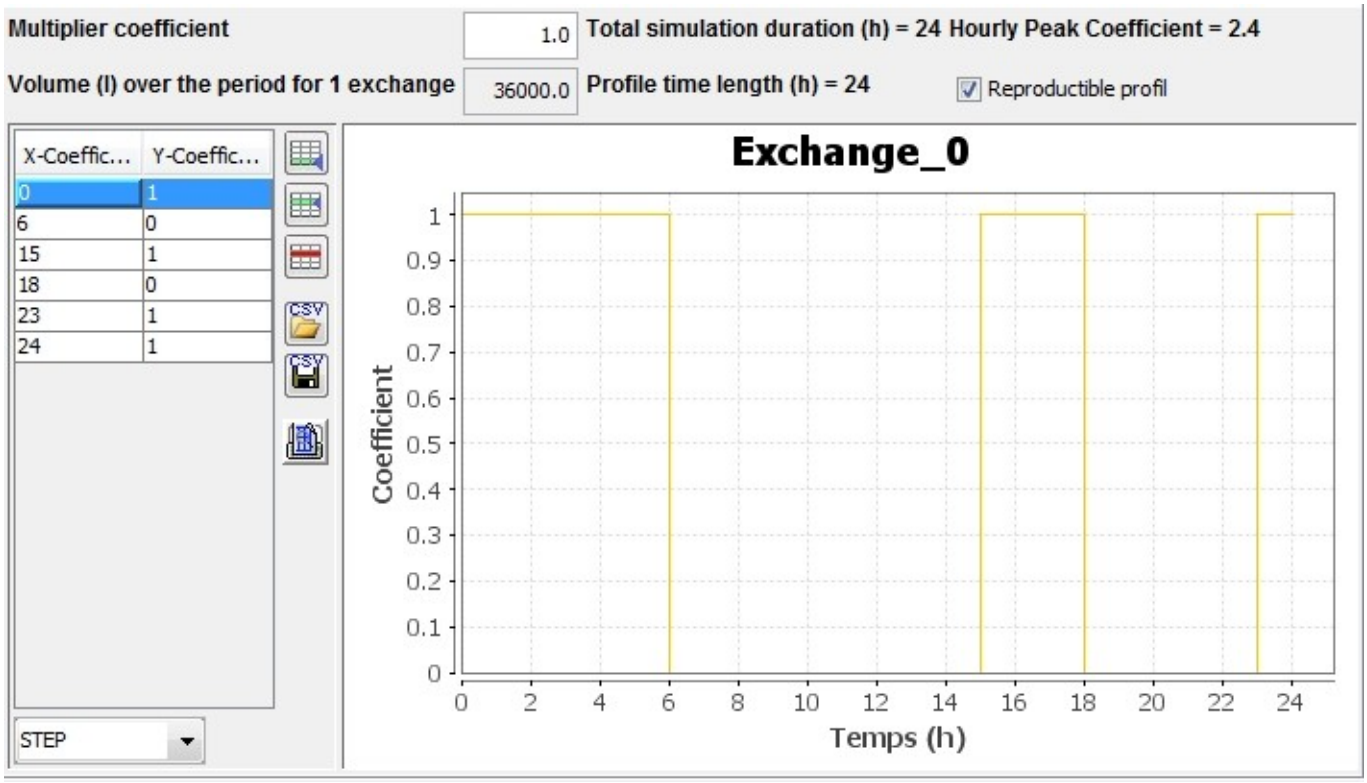
The peak flow given to the node is affected to the maximal value of the curve, all others values are calculated by weighting between the value at time step and the maximum value of the entire curve.

Localised leakage



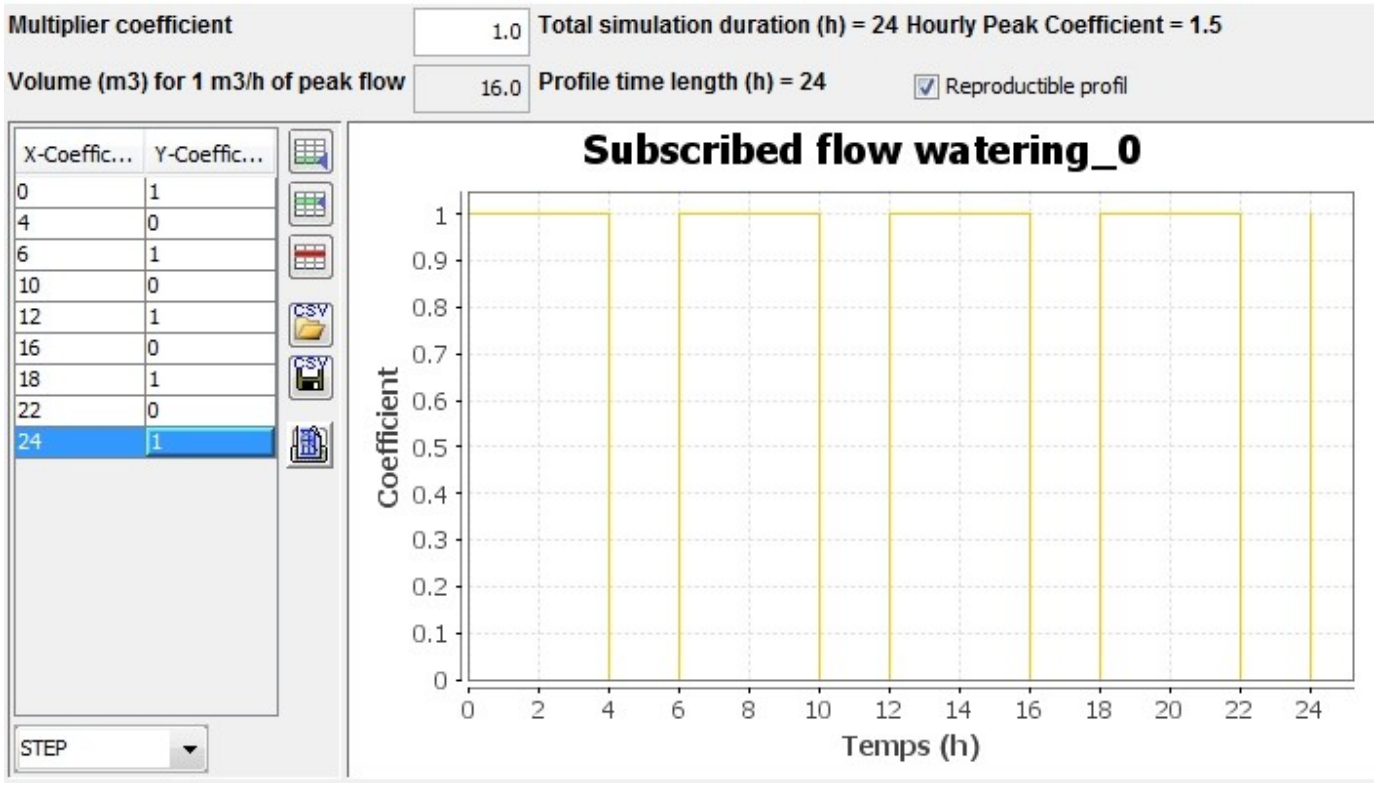
Localised leakage data : **Coefficient multiplier** applied to peak flow. Default value: 1.0.
Volume in liters for the leakage entered over the temporal profile duration.
Non-dimesional temporal profile of localised leakage volume modulation.

Exchanges with neighbouring networks



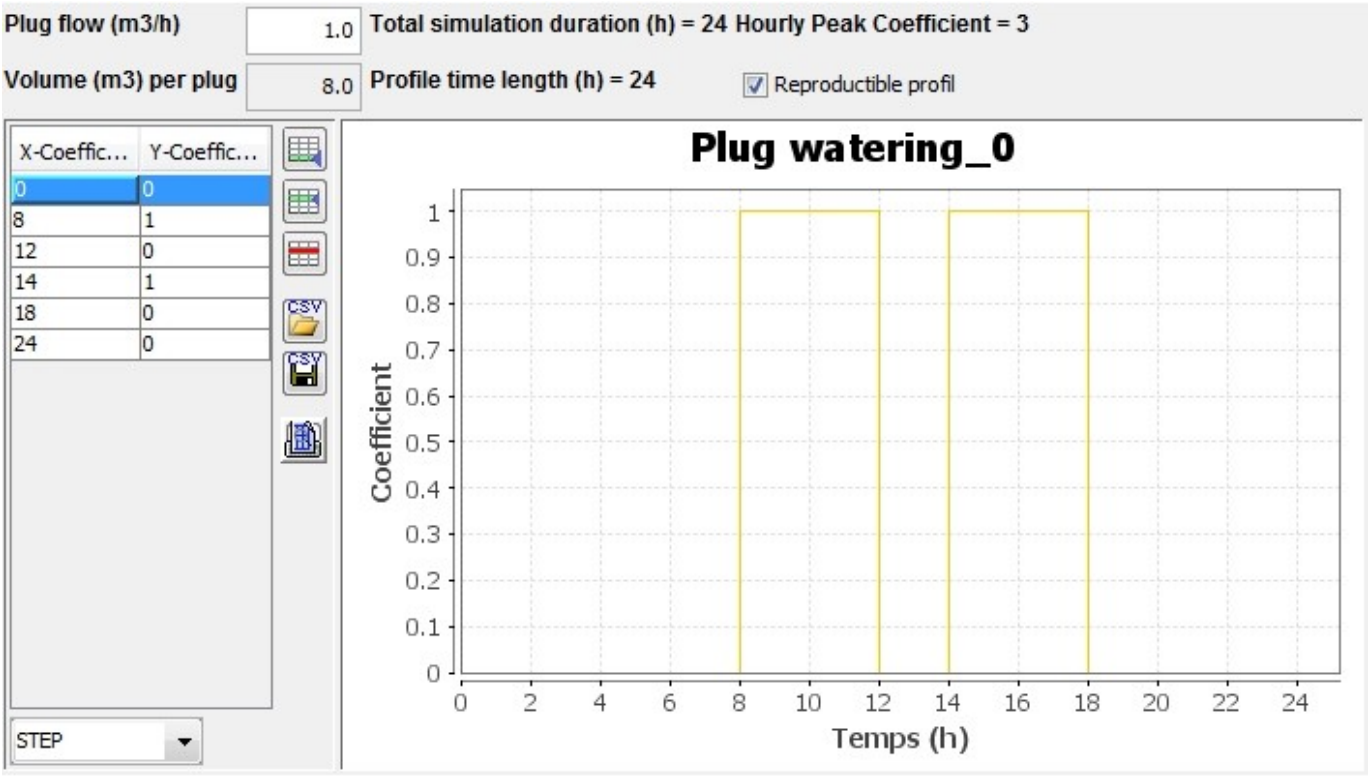
Exchange data : **Coefficient multiplier** applied to temporal profile values. Default value: 1.0.
Volume in liters for the exchange entered over the temporal profile duration.
Temporal profile of flow in (l/s).

Subscribed flow models



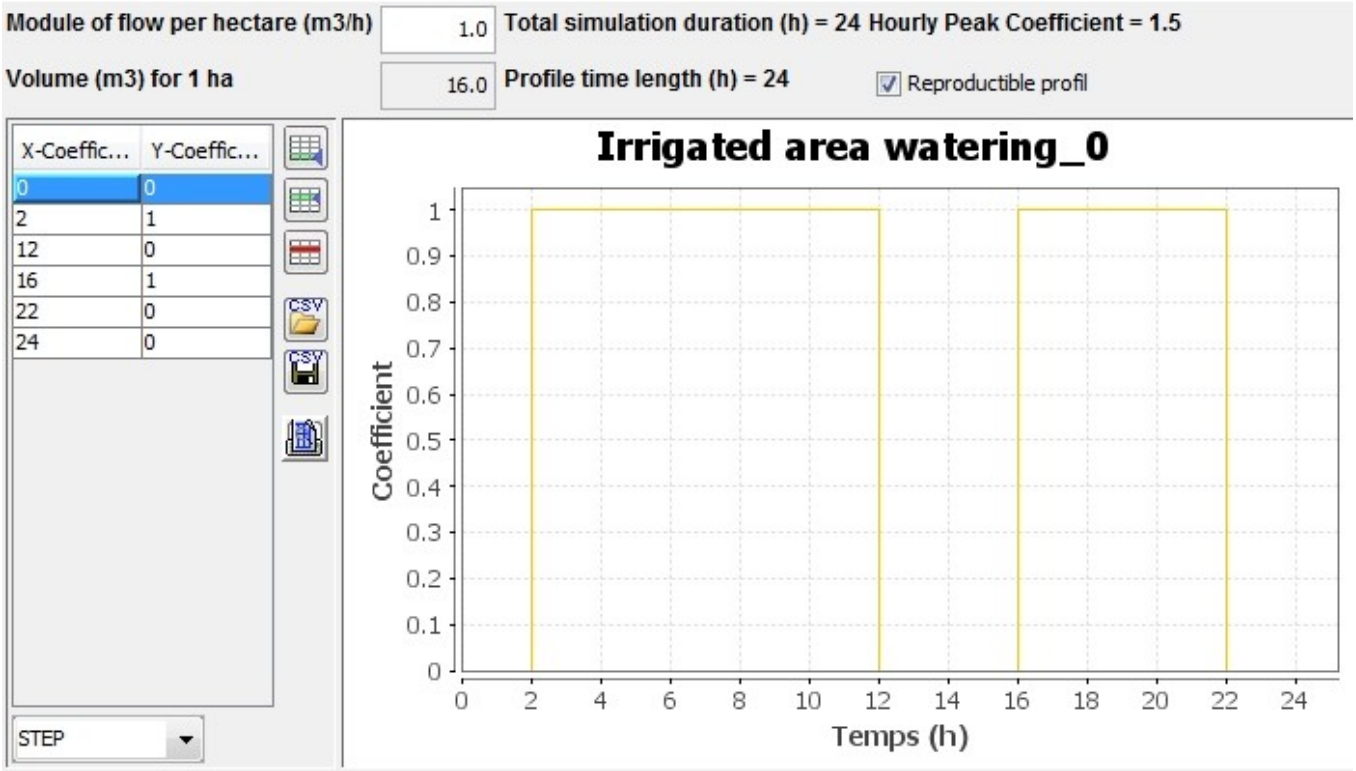
Subscribed flow data : **Coefficient multiplier** applied to temporal profile values. Default value: 1.0.
Volume in m3 for 1 m3/h entered over the temporal profile duration.
Non-dimensional temporal profile of the subscribed flow.

Watering plug models



Watering plug data : **Flow of plug in m3/h** applied to temporal profile values. Default value: 1.0.
Volume in m3 for 1 plug entered over the temporal profile duration.
Non-dimensional temporal profile applied to the flow of plug.

Irrigating area models

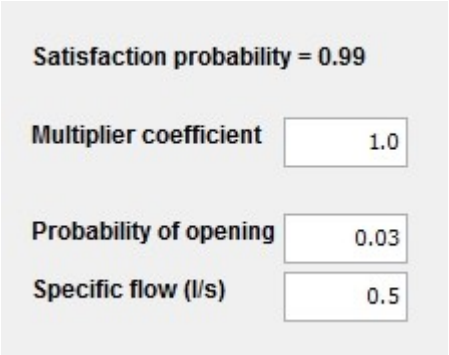


Irrigating area data : **Flow module in m3 per hectare** applied to temporal profile values. Default value: 1.0.
Volume in m3 for 1 ha entered over the temporal profile duration.
Non-dimensional temporal profile applied to the flow per hectare.

Opointe

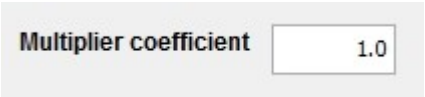
The Opointe module allows the simulation of the behaviour of a meshed water distribution network under pressure by calculating, using a statistical approach, the peak flows and pressures, i.e. at the time of day when demand is at the most.

Domestic consumers



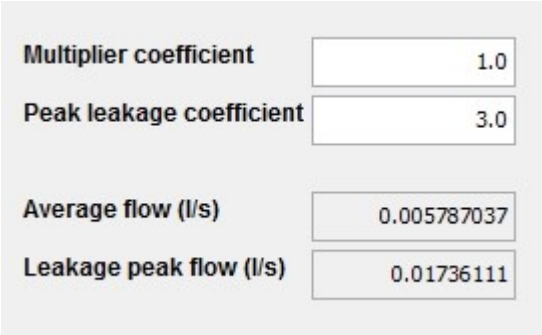
Domestic data : **Probability of satisfaction** of consumers: number between two values thresholds by default 0.5 and 0.999. The value is entered in the Opointe tab under the menu General.
Probability of consumer opening tap: number between two values threshold by default 0 and 1.0 (excluded bounds).
Specific flow (l/s). Default value: 0.5. It corresponds to the opening of two taps on average in France.
Opointe **coefficient multiplier** applied to the number of consumers. Default value: 1.0.

Industrials consumers



Industrial data : Opointe **coefficient multiplier** applied at peak flow. Default value: 1.0.

Localised leakage



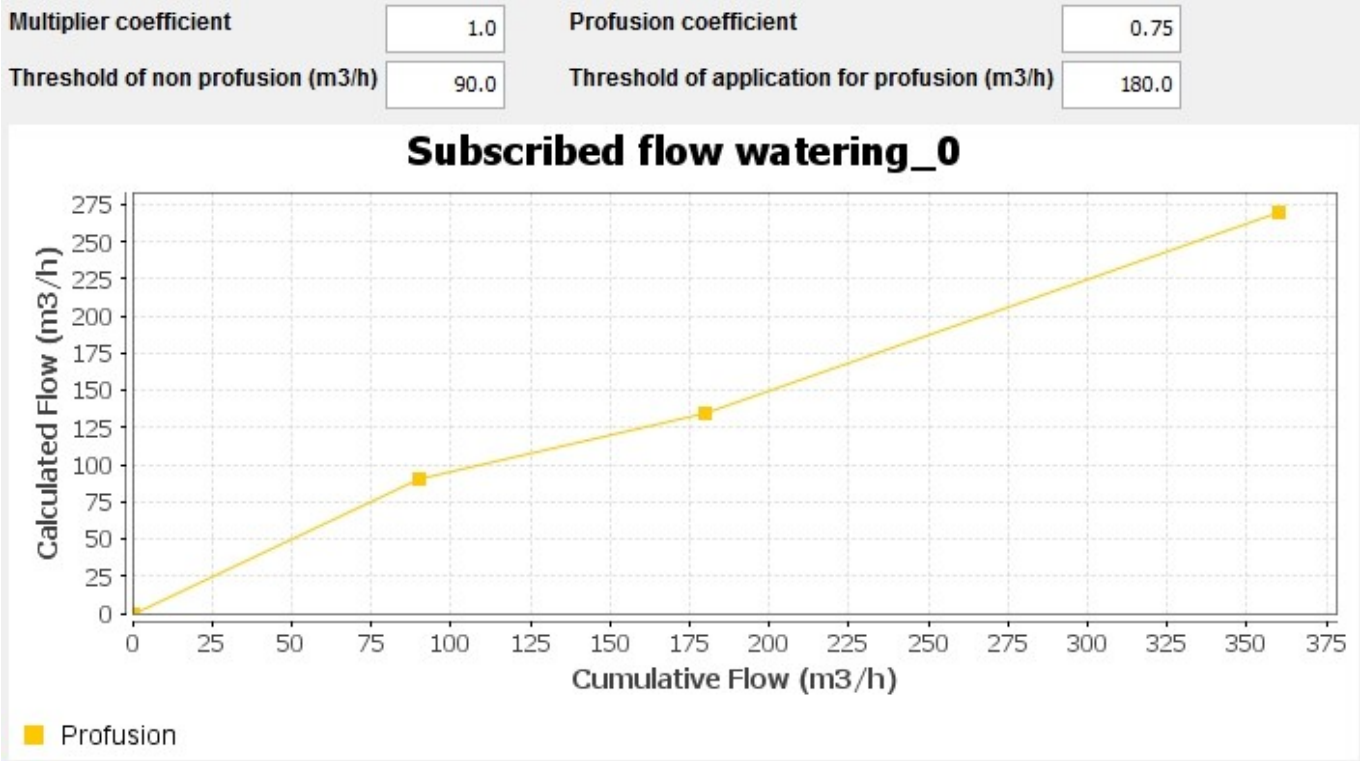
Localised leakage data : Opointe **coefficient multiplier** applied at peak flow. Default value: 1.0.**Peak coefficient** applied to the average flow to calculate the leakage rate for Opointe.

Average and peak flows are calculated from the validated data.

Exchanges with neighbouring networks

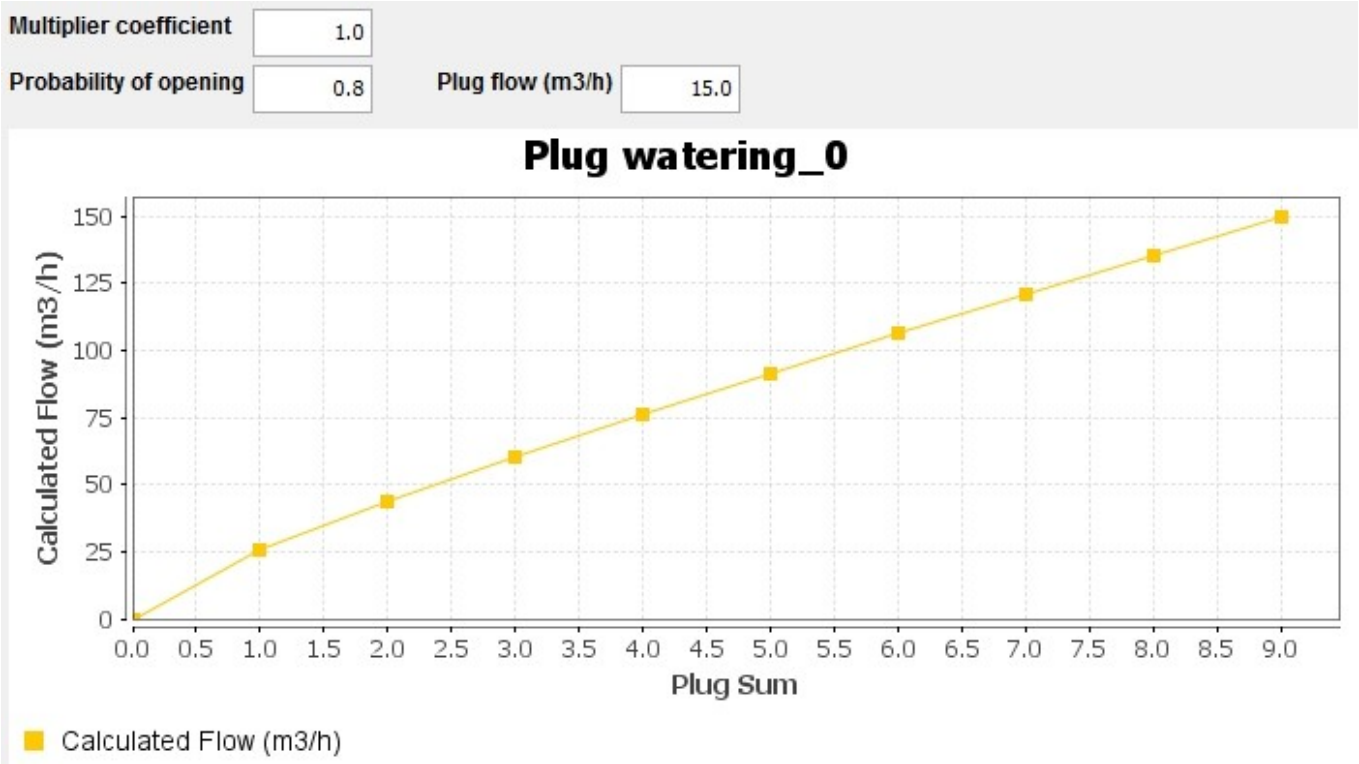
Exchange data : **Coefficient multiplier** applied at peak flow. Default value: 1.0.
Peak flow in l/s.

Subscribed flow



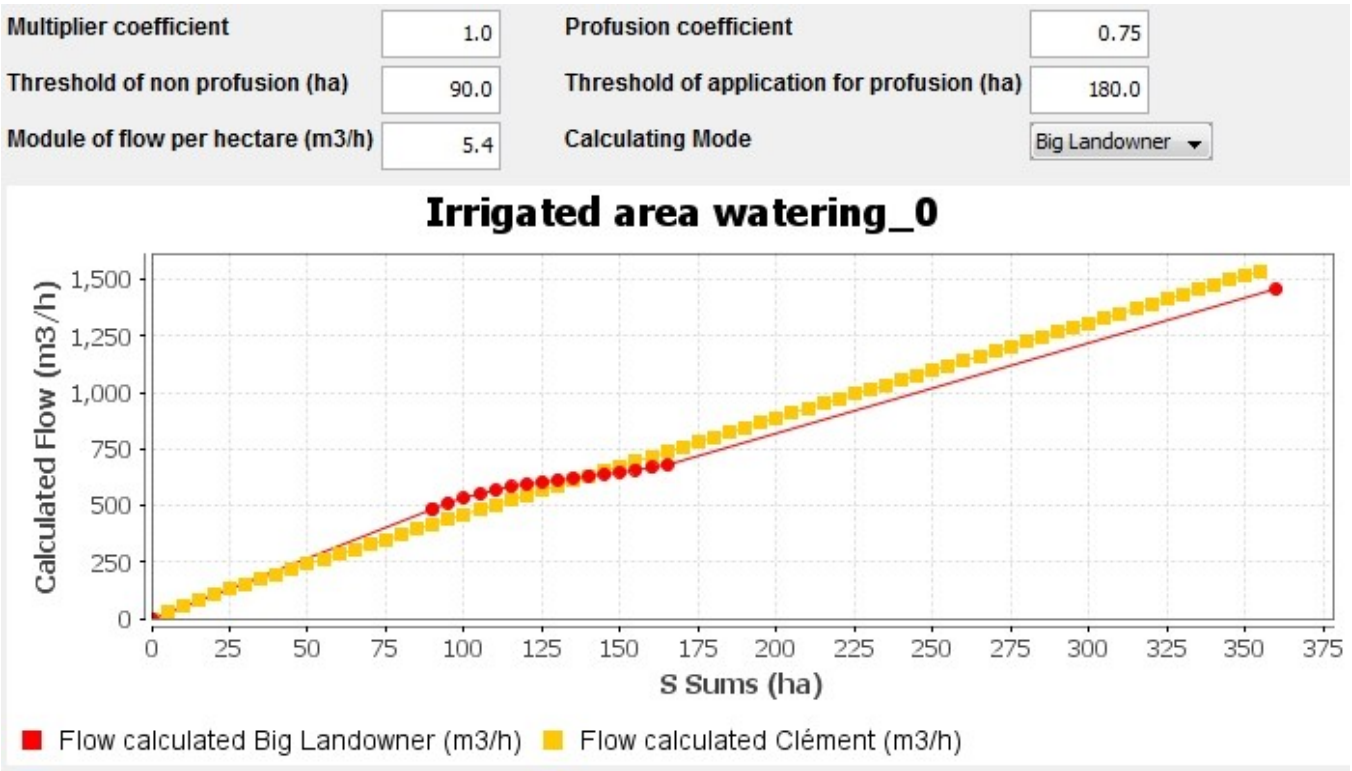
Subscribed flow data : **Coefficient multiplier** applied to flow. Default value: 1.0.
Profusion coefficient: type a number included between the default threshold values 0 and 1.0 (excluded bounds).
Threshold of no profusion (m3/h): highest bound of application of coefficient of no profusion = 1.
Threshold of profusion application (m3/h): lowest bound of application of coefficient of profusion < 1

Watering plug



Watering plug data : **Coefficient multiplier** applied to flow. Default value: 1.0.
Probability of opening plugs simultaneously: type a number included between the default threshold values 0 and 1.0 (excluded bounds).
Plug flow (m3/h): Default value: 15.

Irrigated Aera



Irrigated area data : **Coefficient multiplier** applied to flow. Default value: 1.0.
Profusion coefficient: type a number included between the default threshold values 0 and 1.0 (excluded bounds).Displayed for a Big Landowner calculation.

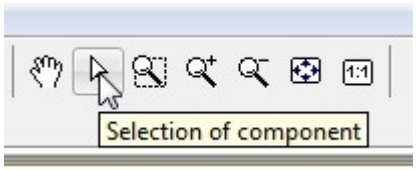
Probability of opening: type a number included between the default threshold values 0 and 1.0 (excluded bounds).Displayed for a Clement's calculation.
Threshold of no profusion (m3/h): highest bound of application of coefficient of no profusion = 1
Threshold of application of profusion (m3/h): lowest bound of application of coefficient of profusion < 1
Module of flow (m3/h) per hectare
Choice of the calculation option:

- Big Landowner: linear law in three parts
- Clement: probabilistic law calculated according to Clement's method

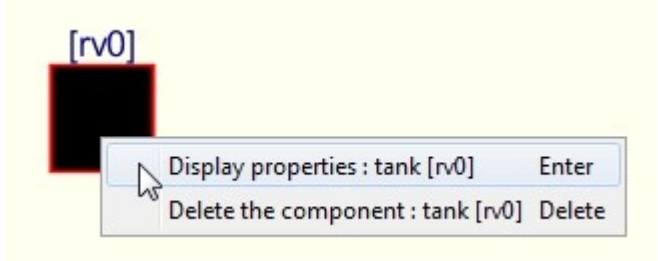
Node

Common data to all types of node

This dialog box is accessible by double left clicking on a node in Schematic View or Map View using the “select” cursor

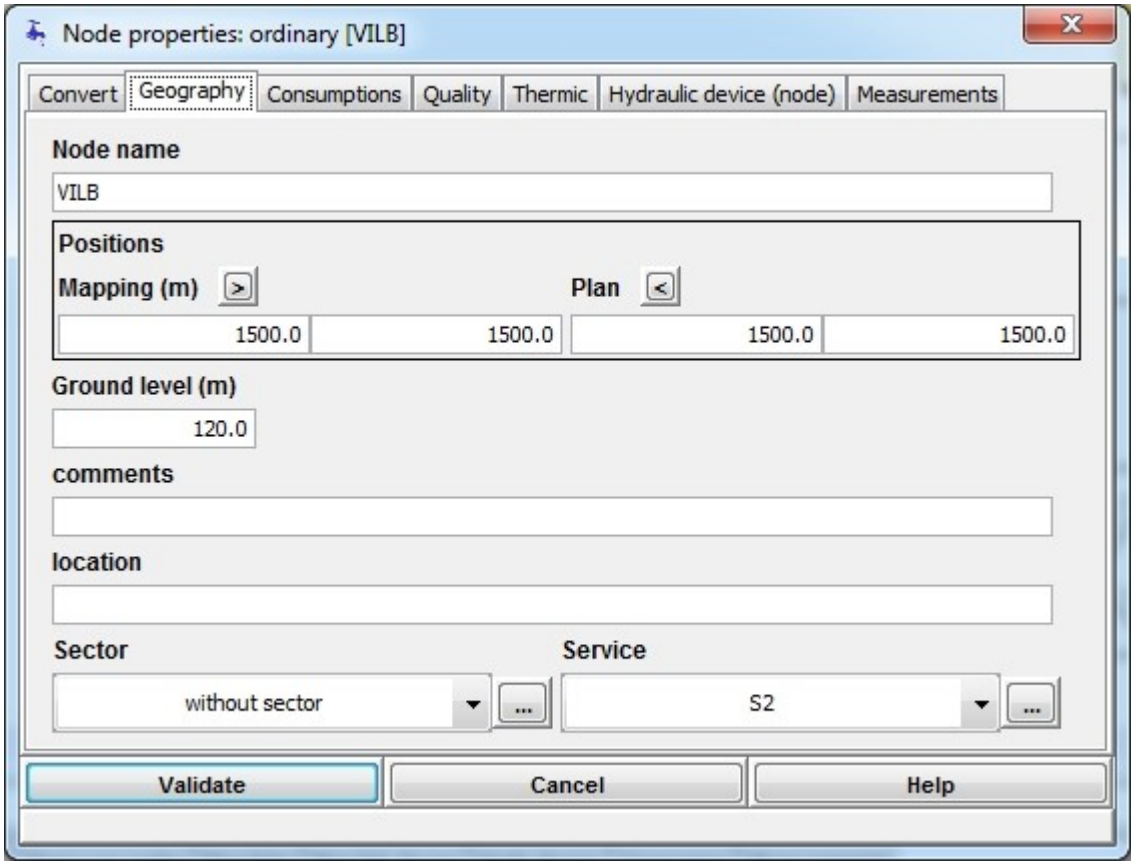


Or by double right clicking and choosing the appropriate line in the contextual menu

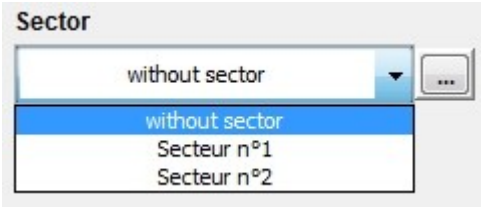


The data entered at this level of the dialog box can be either essential, ie mandatory for the calculations, or supplementary, and not required for the calculation runs. For all the nodes, specification of the node occurs in the first tab, “Geography”. The dialog box title states the type of node and its name.

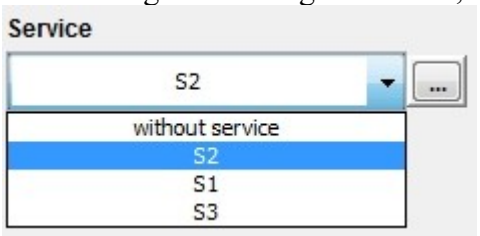
Geographic tab



Data : The **node name** is a chain of characters enabling the node to be uniquely identified; this field cannot be the same for two nodes and cannot be empty. The **coordinates** for the two viewing systems (schematic or map) are entered into the reference system chosen by the user, no constraints or controls are made on these fields. The **ground level** contains the head chosen by the user to represent the altitude of the node for calculations; it serves as a basis for the pressure calculation of the difference with the piezometric head. The **comments** field allows characters to be entered without control. The **location** field allows characters to be entered without control.



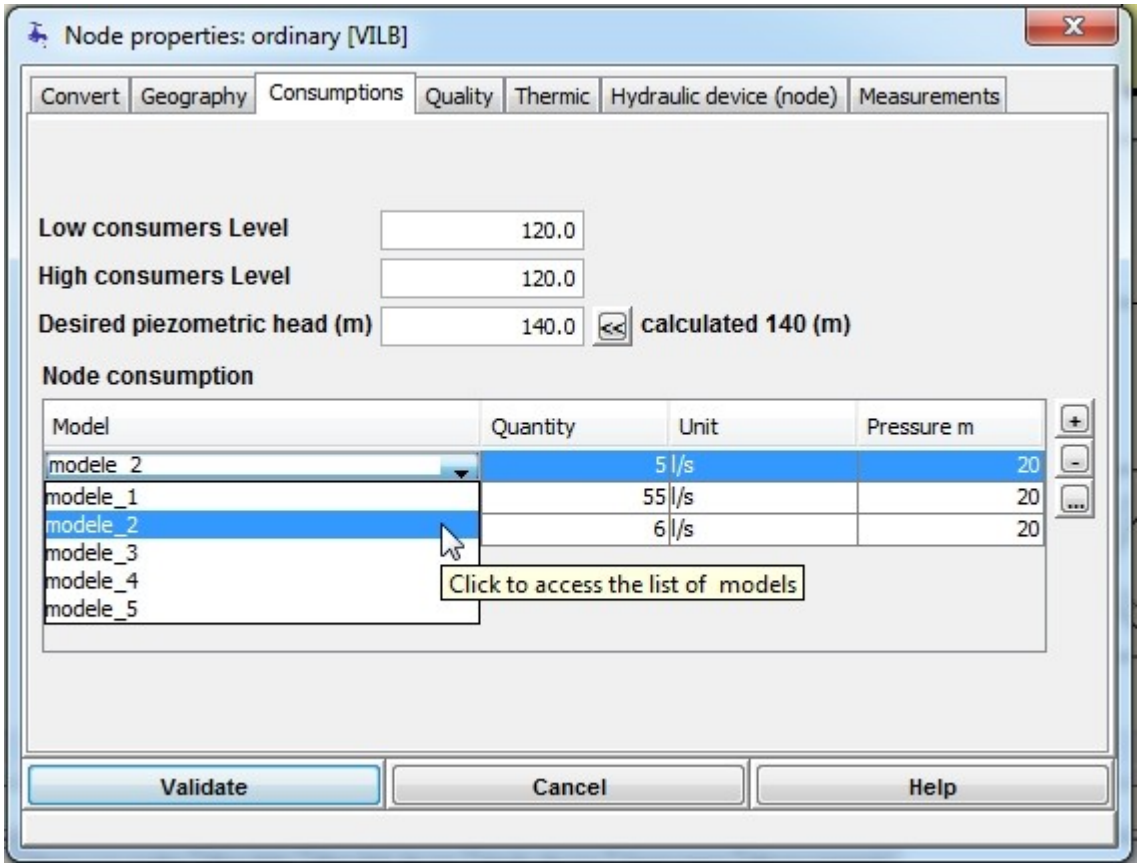
The **sector** and **service** fields allow the user to choose from a list of sectors and/or services to which the node belongs. If nothing is selected, the choice remains as “without sector” and/or “without service”.



The  buttons open a dialog box for entering lists of possible sectors and services.

Ordinary Node

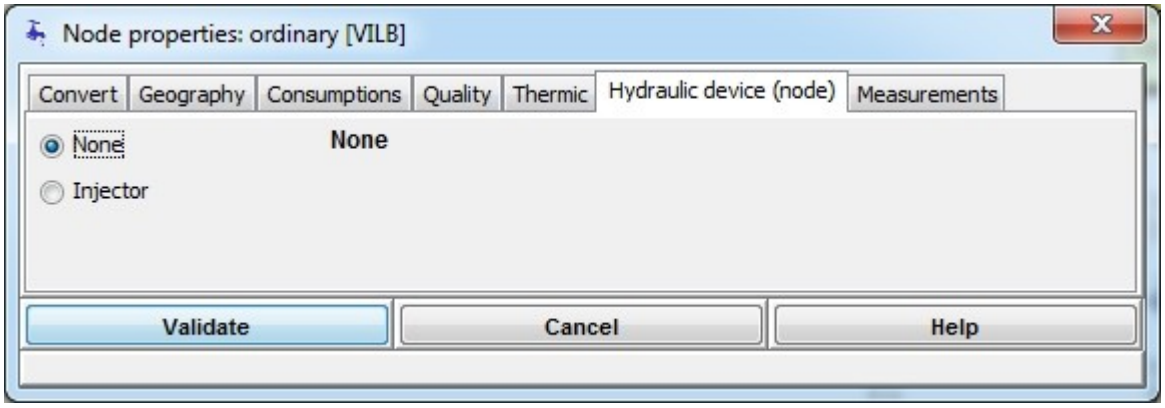
Consumptions tab



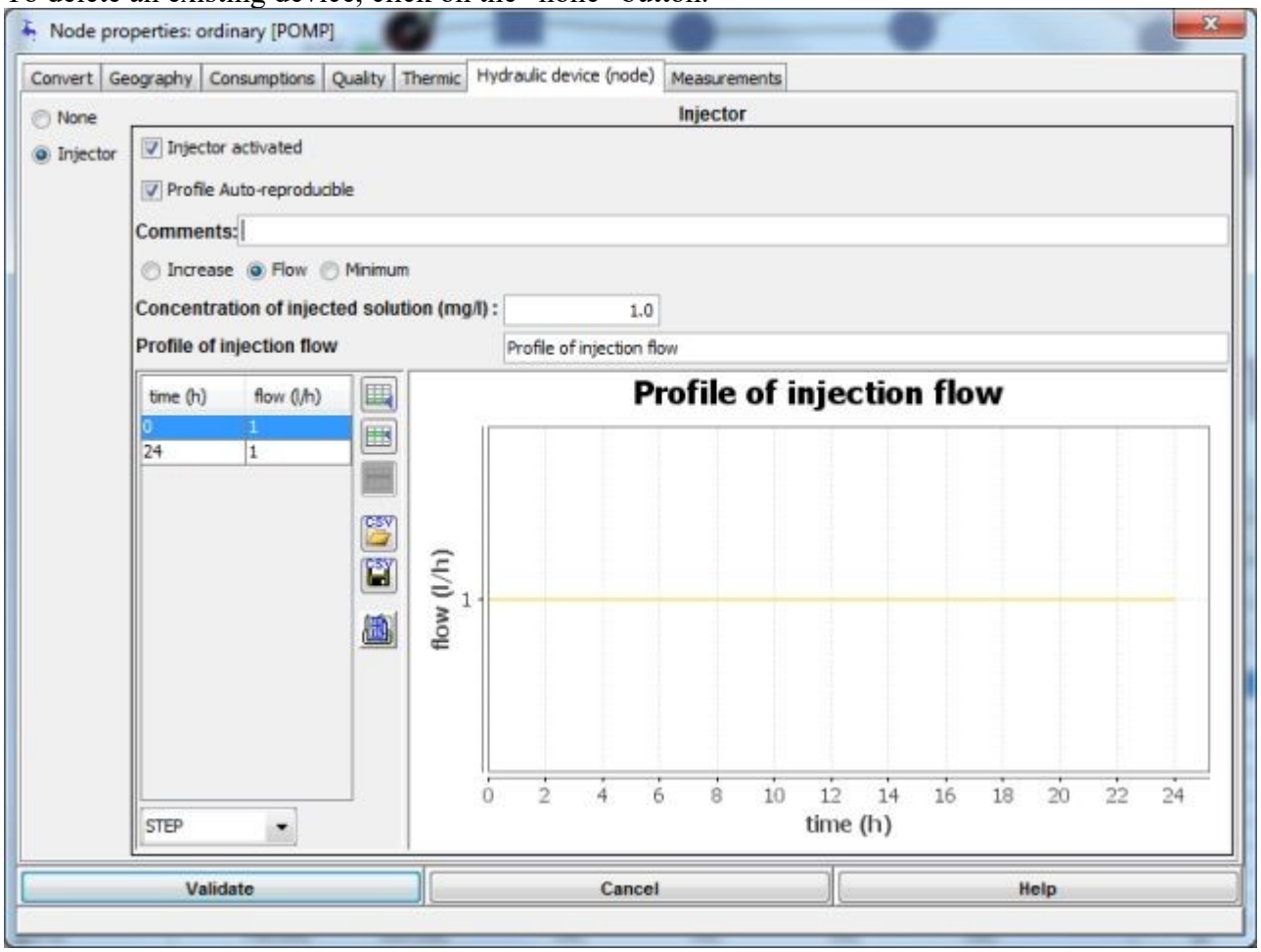
An ordinary node can symbolise a group of consumers, remote in distance and altitude. To take into account this dispersion, it is possible to enter in the corresponding field, the **“Low user head”** and the **“High user head”**. These fields are not used in the simulations but may allow the identification of pressures outside the limits set by the user (for example by a colour code).
In case the ground level is modified they are automatically changed to this value.
The **piezometric head required** field allows a value to be assigned for the minimum head that the user requires in the hypothesis calculations. It is automacally calculated and proposed, but it is not updated automatically. The proposed value follows any change in the rating of the ground level added the maximum between the standard desired pressure done in the "generalities tab preference" and the maximum value of the minimum pressure necessary to consumers of the node.

The button << allows inject the proposed value in the field.
The “consumption at node” table contains the pair “consumption model” and “quantity consumed”. To add a pair, click on the + button. To delete a pair, select the line and click on the - button.
Access to the dialog box for consumer models is made by clicking on the ... button.
To modify the model of the pair, click on the cell of the line in the “Model” column and a scrolling list proposes all the models available. In the example, the four different types of model are illustrated. As a function of the type chosen, the “Quantity: and “Unit” columns are updated. For the “local leakage” or “exchange” models, the quantity must be in “l” and so the unit is left blank.

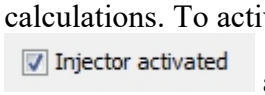
Node hydraulic device tab

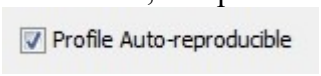


An ordinary node may or may not be equipped with a hydraulic device that could affect the quality calculations. By default, no device is present.
To modify the type of device at a node, click on the radio button corresponding to the type of device required. Only one device can be attached to each node.
To delete an existing device, click on the “none” button.



The “Injector” button leads to a dialog box for entering data for a disinfectant injector. By default this device is disabled when created so the data can be conserved even without being used in the quality calculations. To activate the injector, click on the 

button which then becomes 

and vice versa.
The graph describing the injection as a function of time can have a duration different to that of the simulation. In this case, it is possible to prolong it by repeating the cycle as many times as necessary by clicking on 

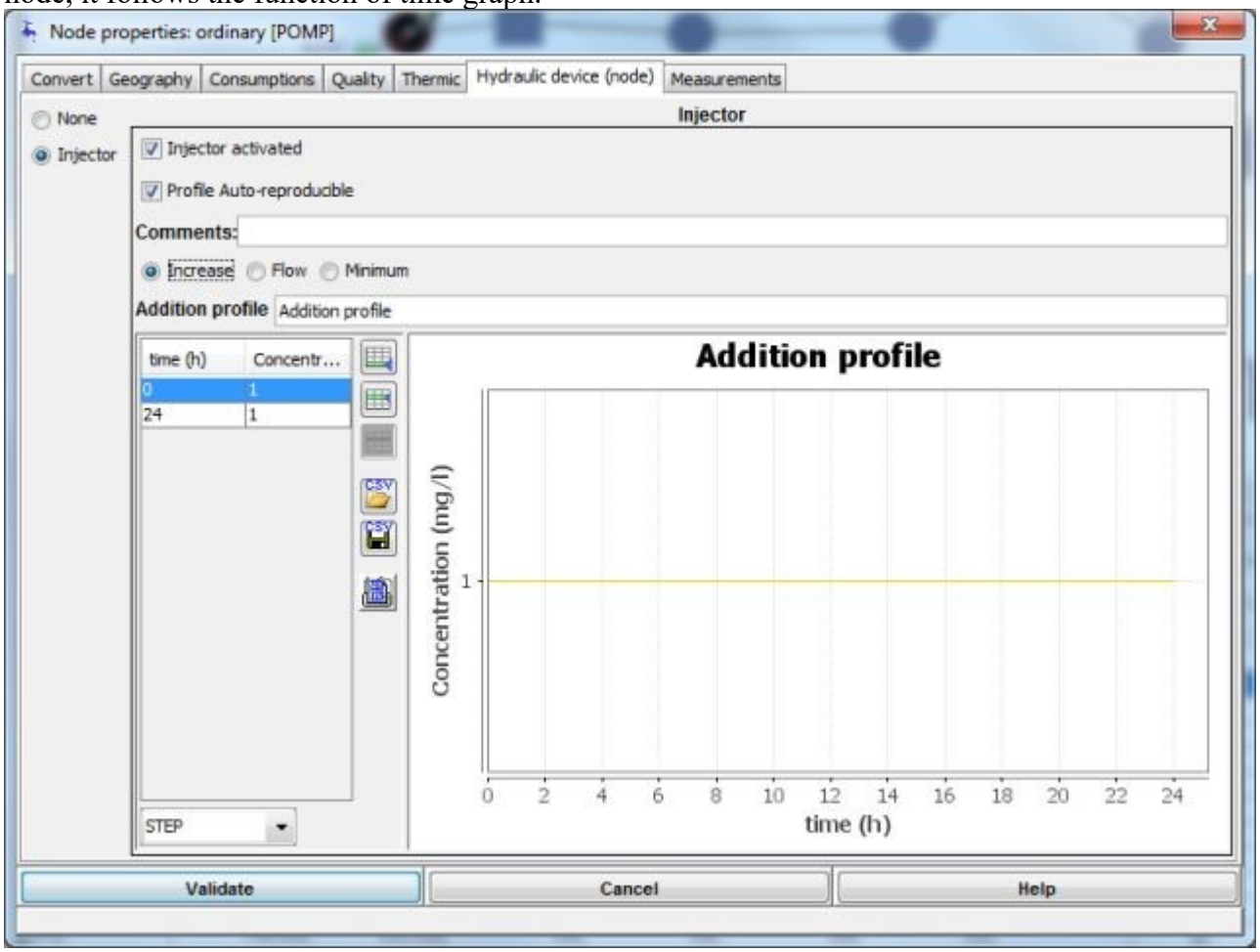
. Otherwise, the injection can stop at the end of its cycle by clicking on 

.

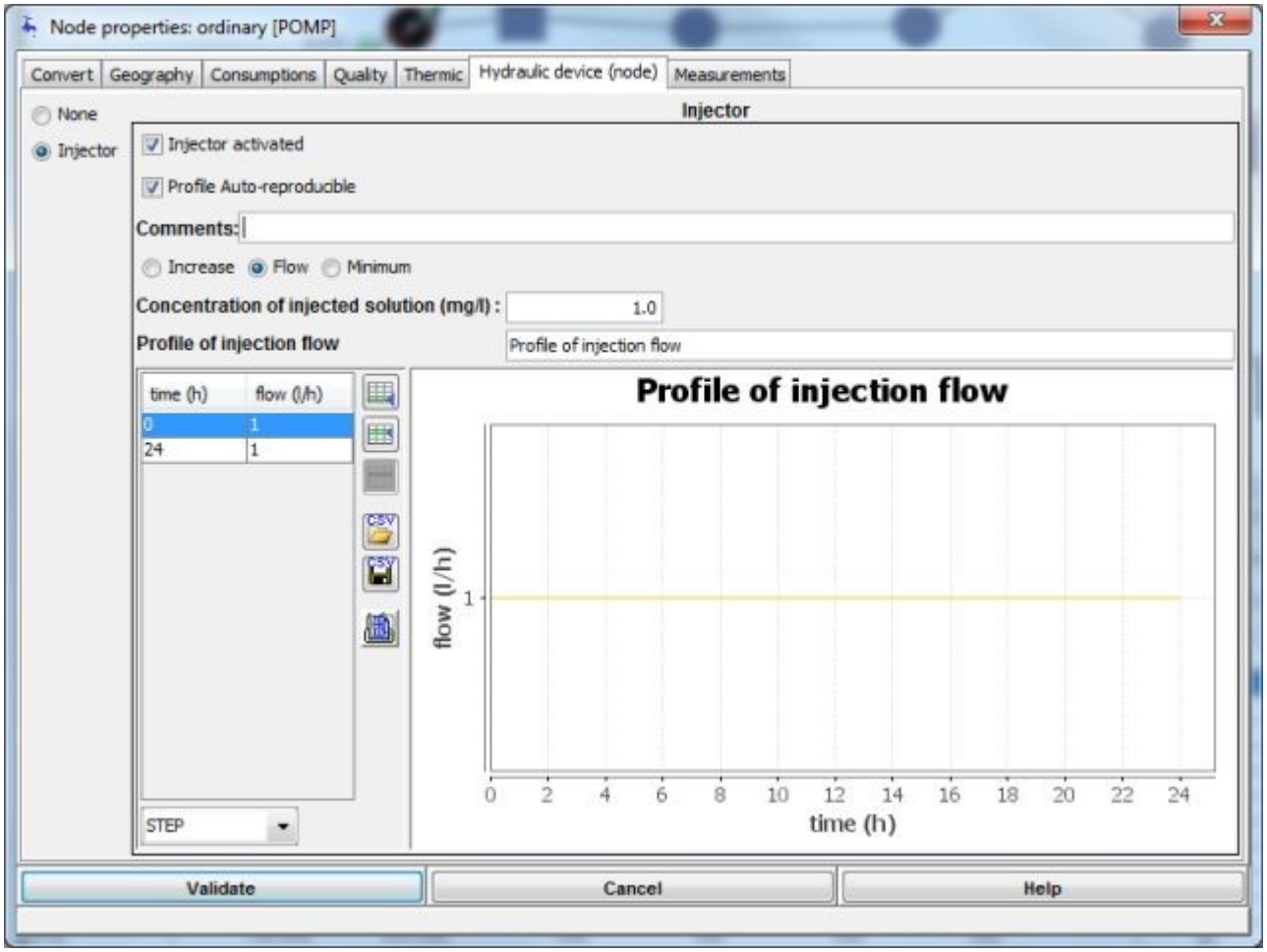
The **comments** field allows a field of any length to be entered without control.
The type of injector chosen, “Concentration of the injected solution” allows the initial disinfectant concentration to be specified; this is constant.

The injector type is chosen by selecting one of the three radio buttons: 

.
Increase: the dose of disinfectant entered is added to that already present in the water at the injection node, it follows the function of time graph.

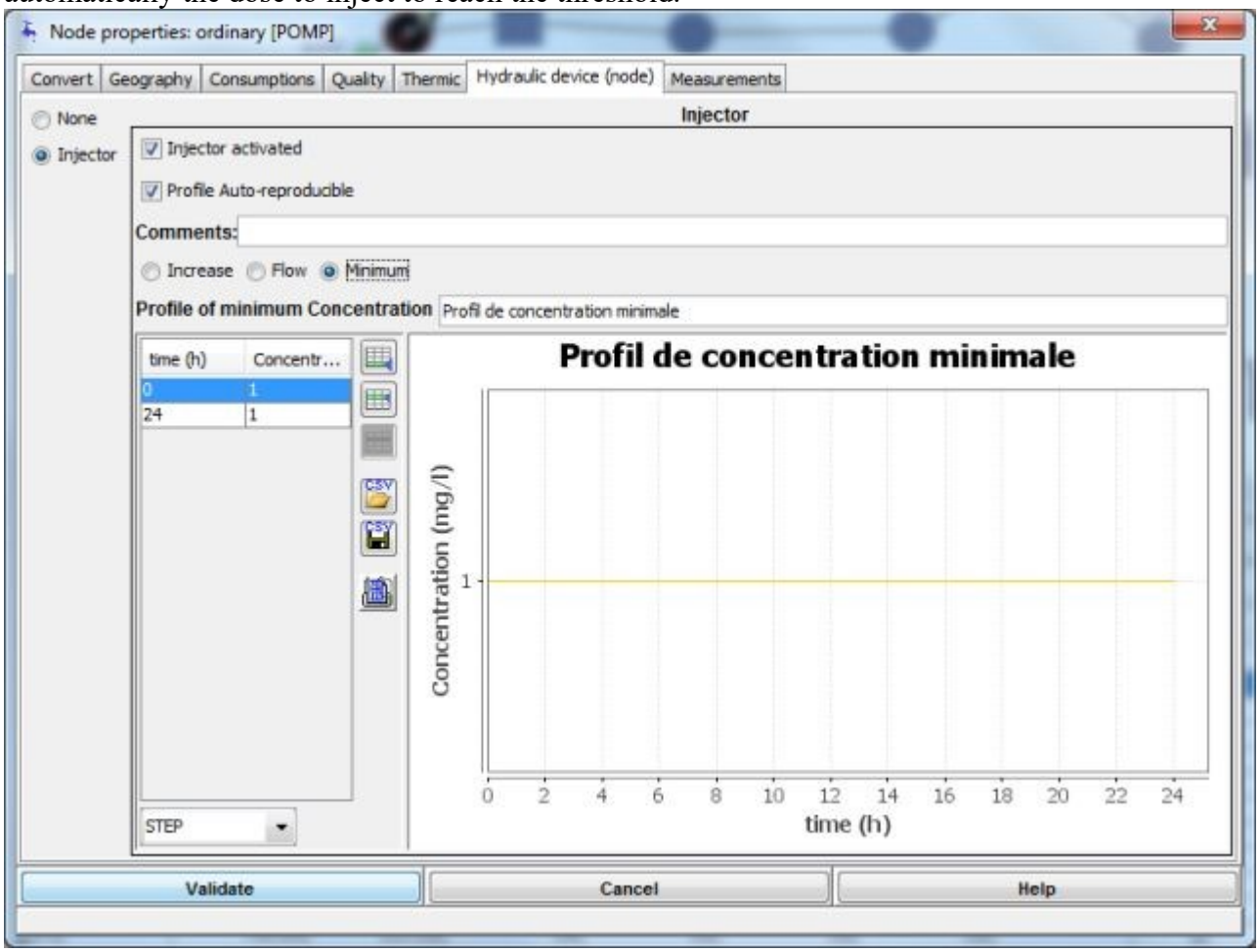


Flow: the calculated dose of disinfectant is provided by injecting the flow entered according to the curve at the initial concentration.



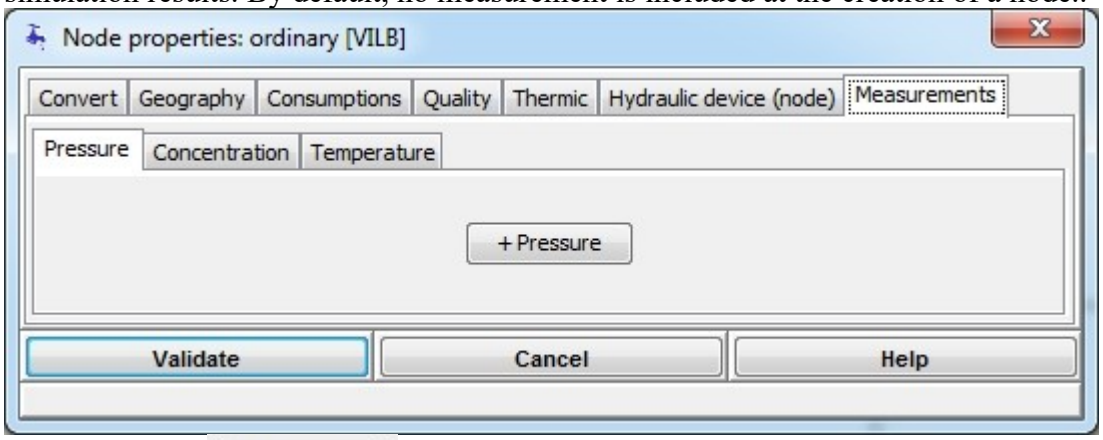
Minimum: the concentration exiting the injection node must always reach a minimum value entered in the graph; if it is already greater at the entry, no disinfectant is added. The simulation calculates

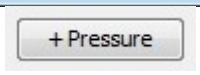
automatically the dose to inject to reach the threshold.



Measurements tab

It is possible to enter measurement values at an ordinary node. These will be superimposed on to the simulation results. By default, no measurement is included at the creation of a node..

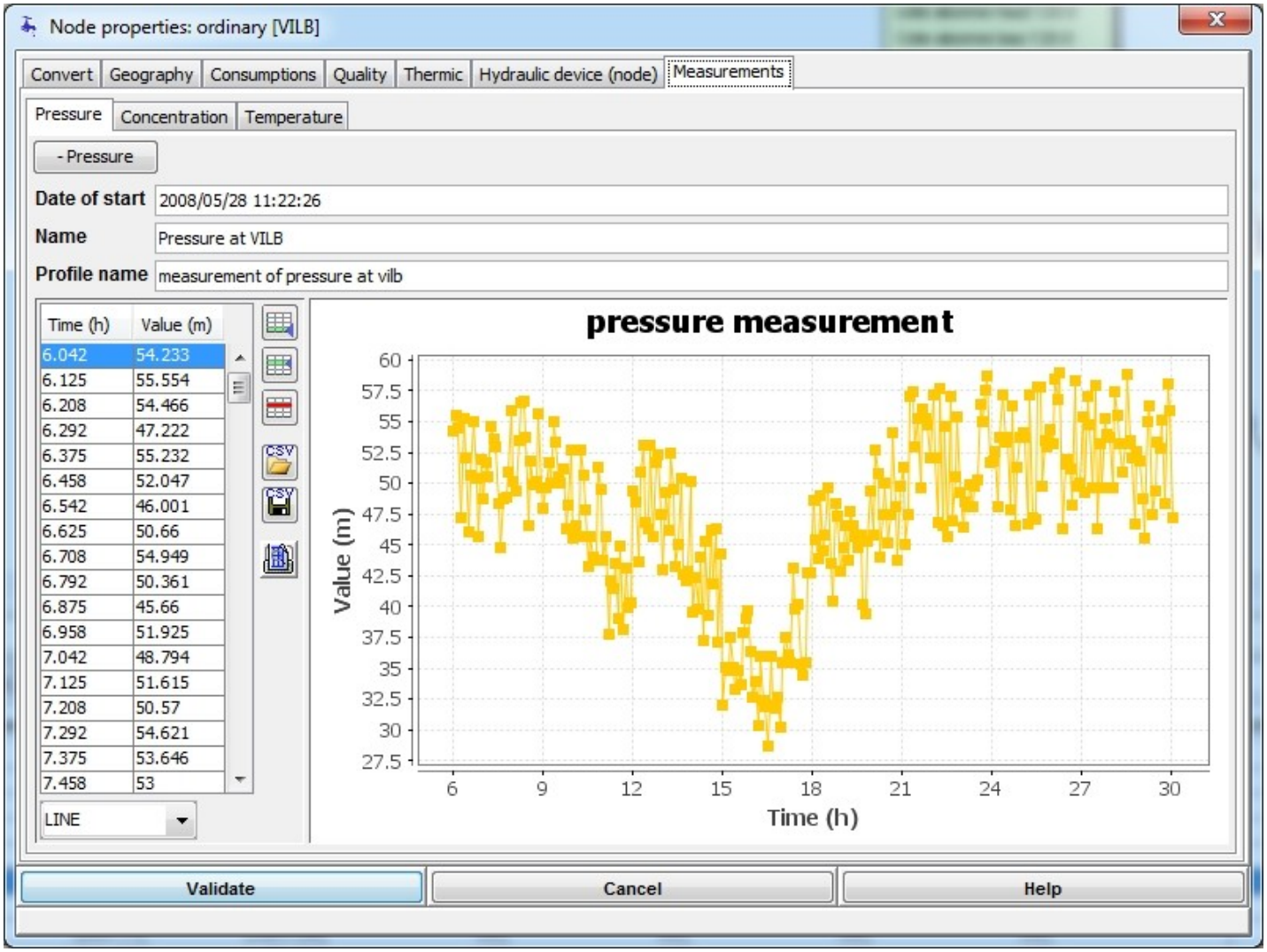


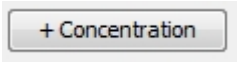
By clicking the 

button, the dialog box for entering a pressure measurement is shown.

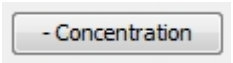
To delete a pressure measurement, click on the 

button.



By selecting the “Concentration” tab and clicking the 

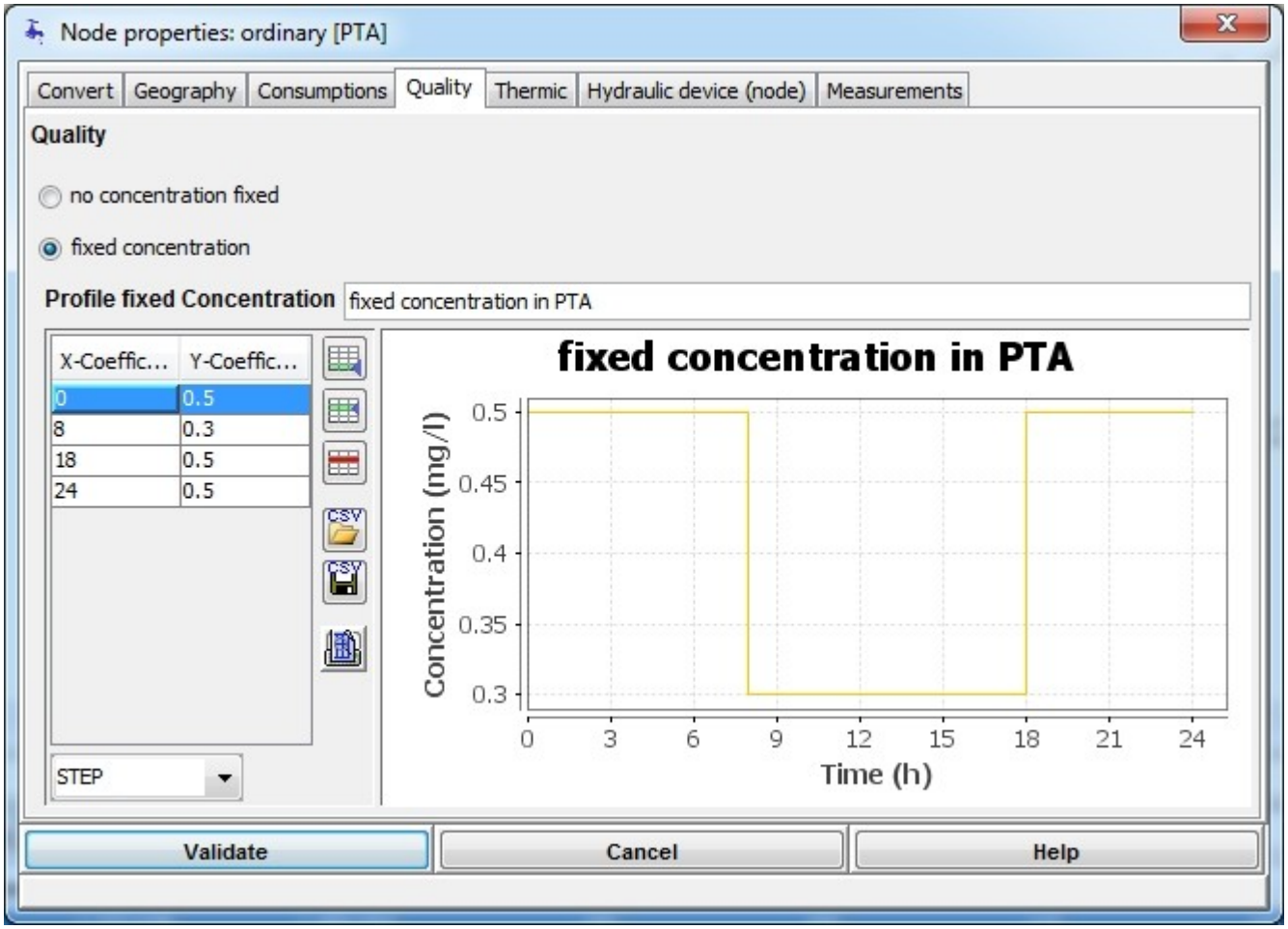
button, the dialog box for entering a concentration measurement is shown.

To delete a concentration measurement, click on the 

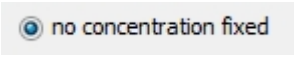
button.

This is the same operation for a temperature mesurement.

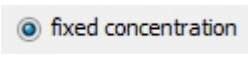
Quality tab



For qualtiy module limits conditions may be fixed.

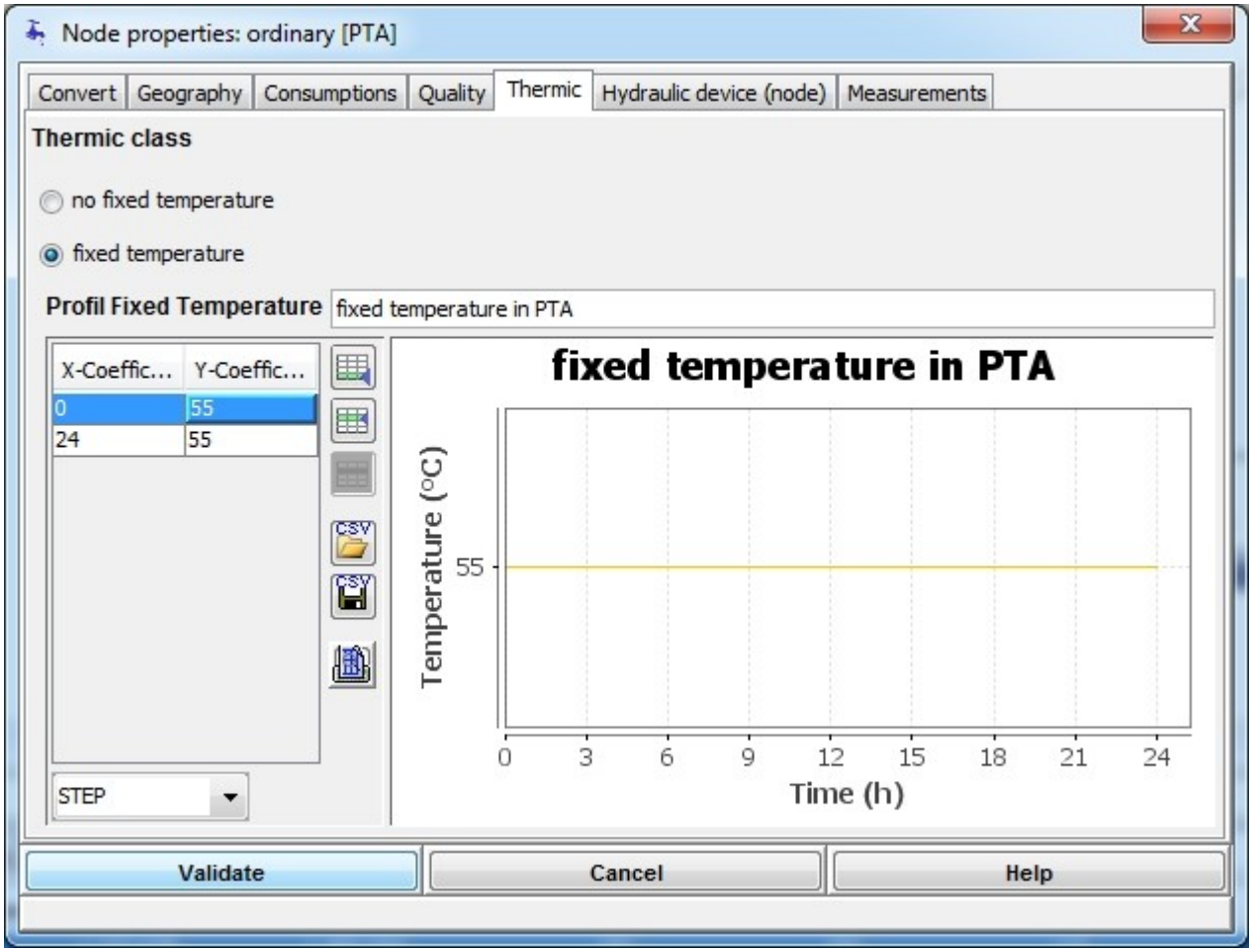
If the button 

is selected, concentration is free and calculated along the simulation.

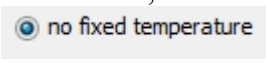
If the button 

is selected, a profile of concentration fixes, during the simulation, the concentration. If the length of the profile is less than the duration of simulation, the profile is copied from its start time as many times as necessary.

Thermic tab



For the Thermic module, limits conditions at the node can be specified.

If the button 

is selected, temperature is free and calculated for the entire simulation.

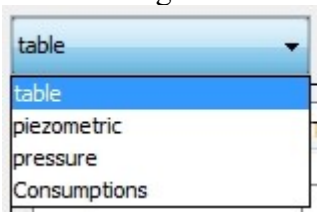
If the button 

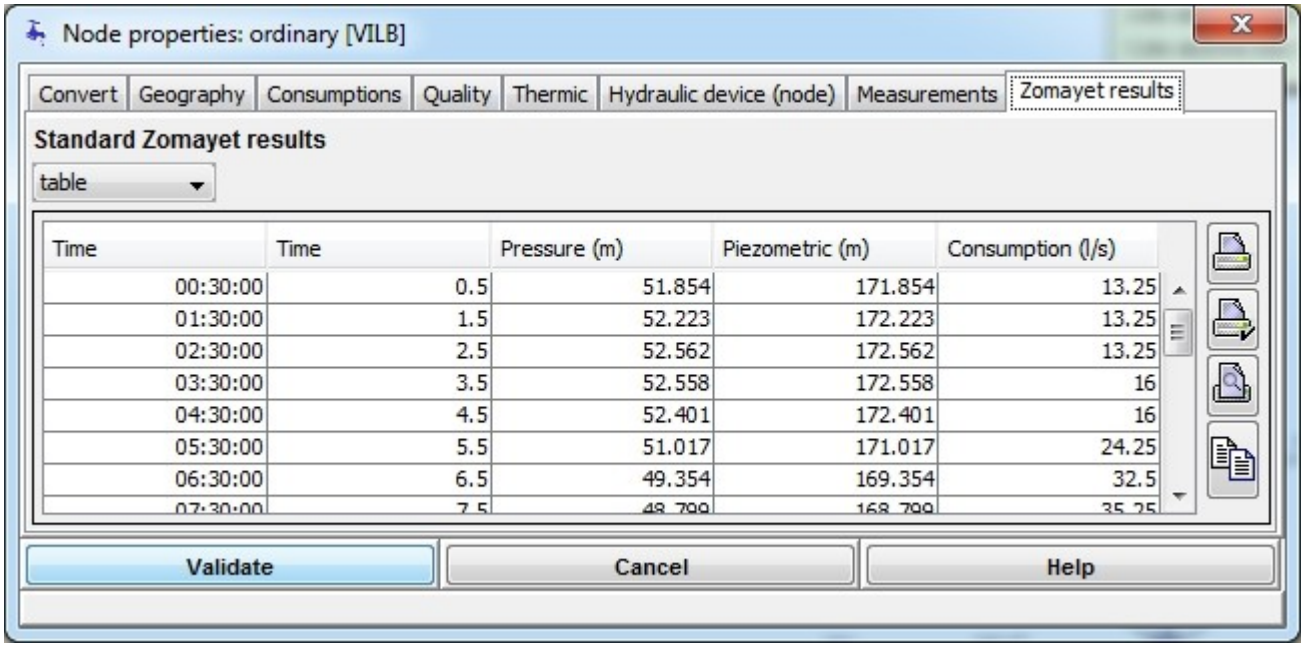
is selected, a profile of temperature fixes, during the simulation, the temperature. If the lenght of the profile is less than the duration of simulation, it is copied from its start time as many times as necessary.

Zomayet Results tab

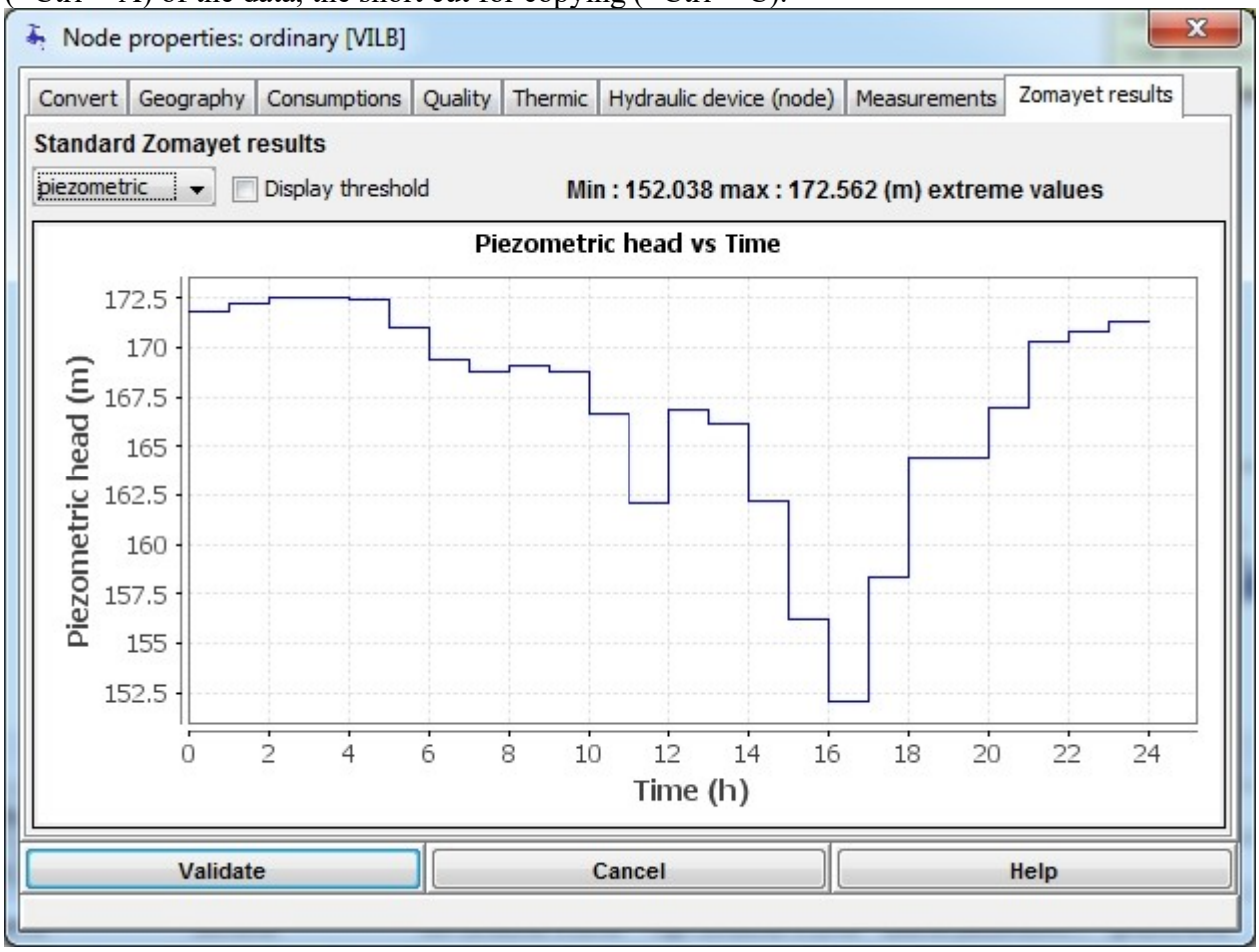
If any results from the Zomayet module are present for the selected node, the “Zomayet results” tab appears. Warning, even after modifying the data used for the Zomayet calculations, the results remain present until another calculation replaces them.

The scrolling list below allows the choice between different representations of the results for the node.

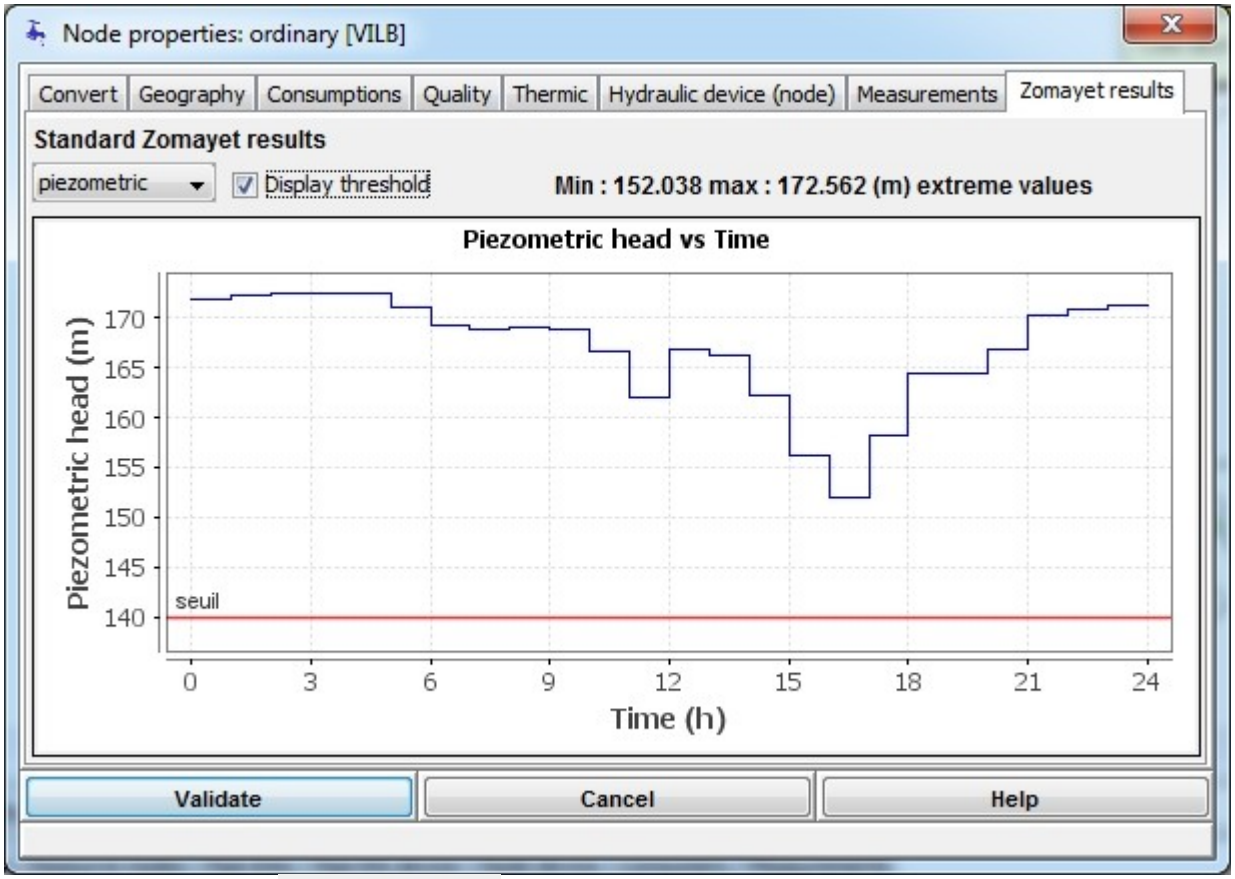


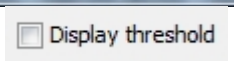


The choice “table” allows all the numerical values of all the calculated fields to be consulted: piezometric head, pressure and consumption. It allows copying to other tabular applications by selecting some or all (<Ctrl>+A) of the data, the short cut for copying (<Ctrl>+C).

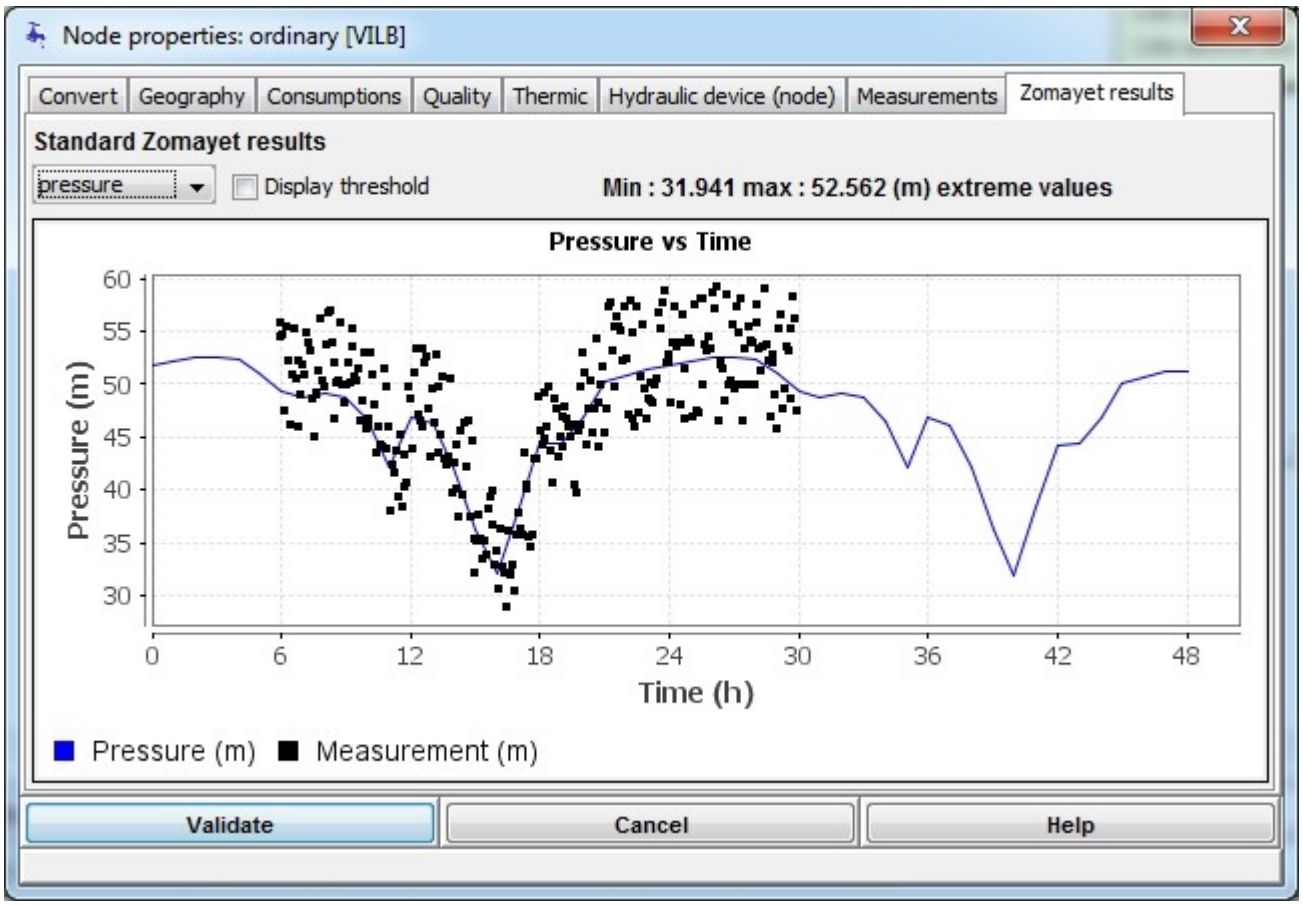


The choice “piezo” produces a graph of the piezometric head calculated at the node as a function of time. Above the graph are shown the extreme values reached during the simulation.



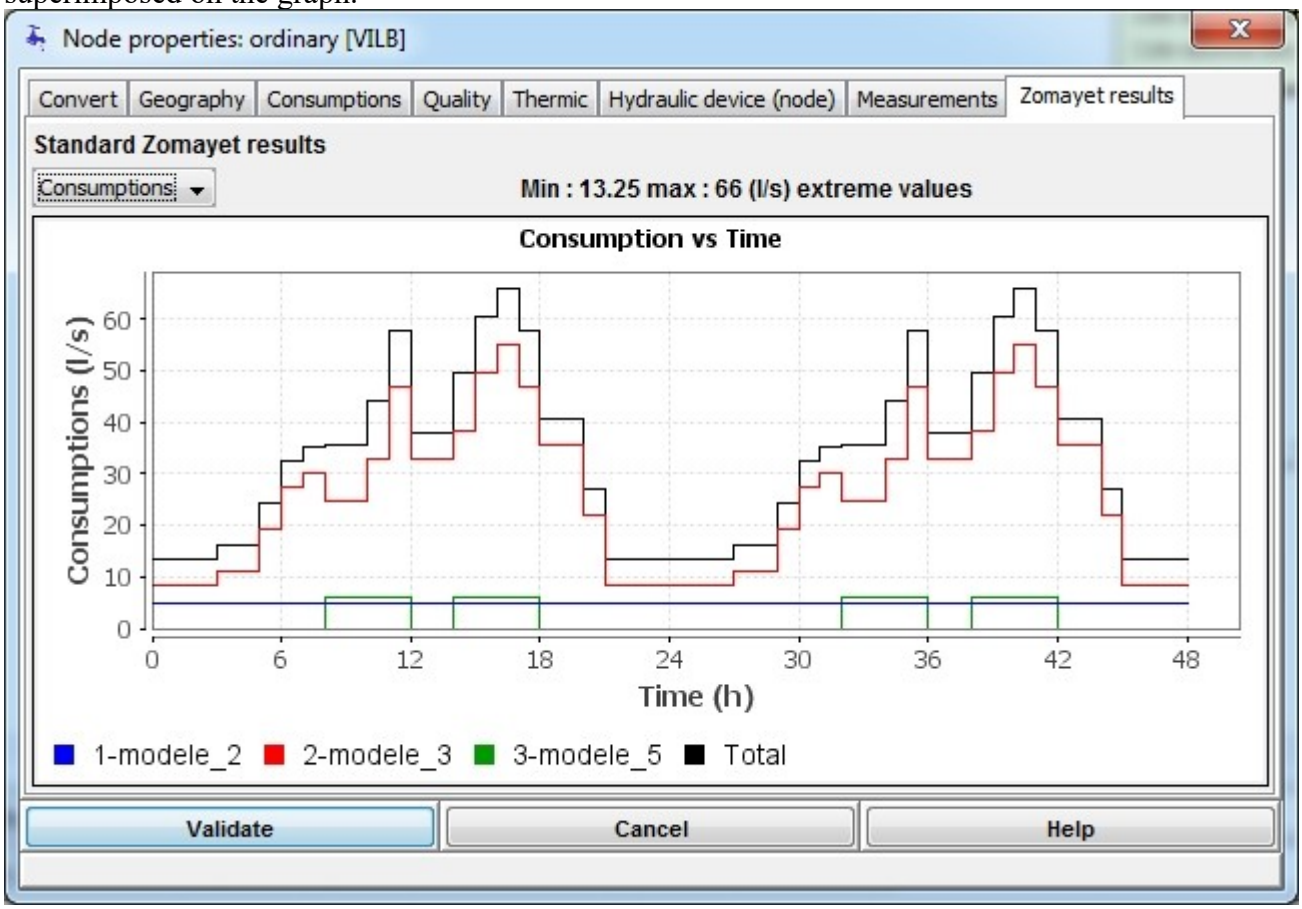
The activation of the 

shows a red line representing the required piezometric head for the node.



The choice “pressure” produces a graph of the pressure calculated at the node as a function of time.
Above the graph are shown the extreme values reached during the simulation.
The display of required piezometric head is transformed into pressure.

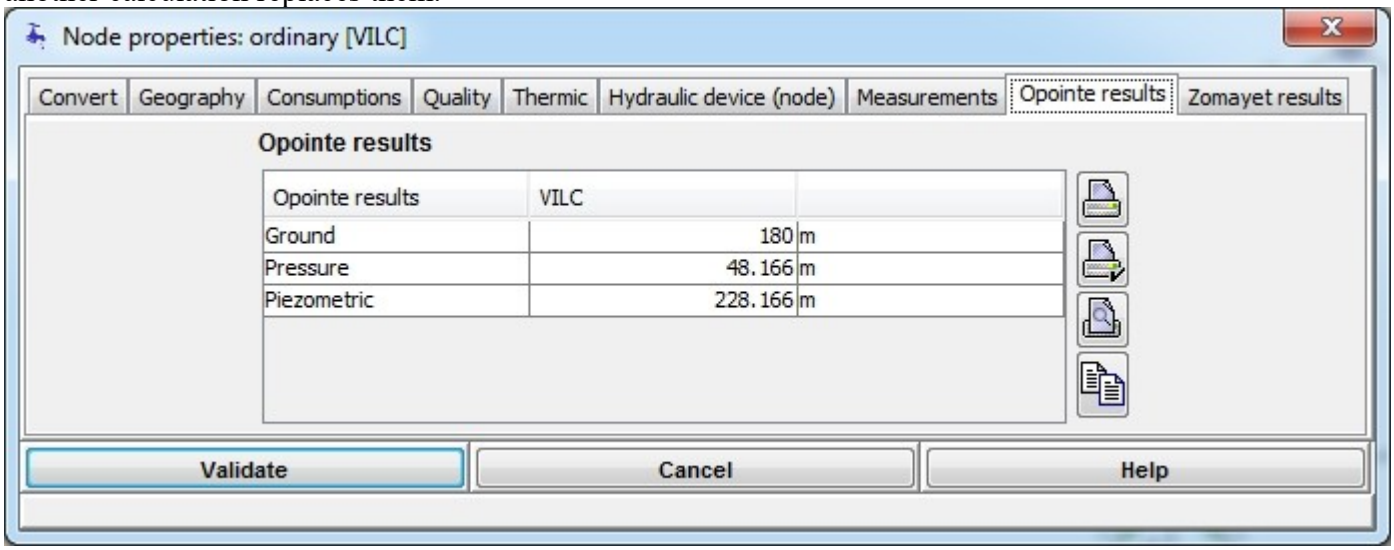
If a pressure measurement has been entered at the node, points representing the measured values are superimposed on the graph.



The choice “consumptions” produces a graph of the consumptions calculated at the node as a function of time. Above the graph are shown the extreme values reached during the simulation. Each of the consumers is represented as well as the algebraic total for the node.

Opointe results tab

If any results from the Opointe module are present for the selected node, the following tab is shown. Warning, even after modifying the data used for the Opointe calculations, the results remain present until another calculation replaces them.



The result of the peak calculation gives the piezometric head at the node; the difference with the ground level gives the pressure.

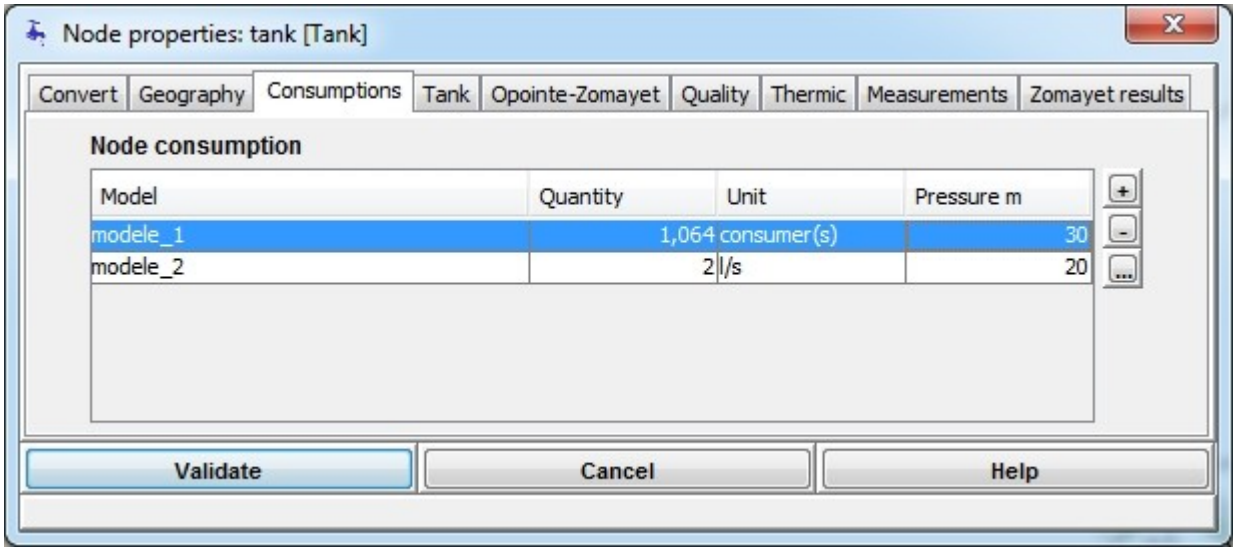
Onglet Résultats Qualité et Thermic

If any results from the Quality and Thermic modules are present for the selected node, the corresponding tabs are shown. Their respective functionings are identical to those of the Zomayet results, for variables of their own.

Tank Node

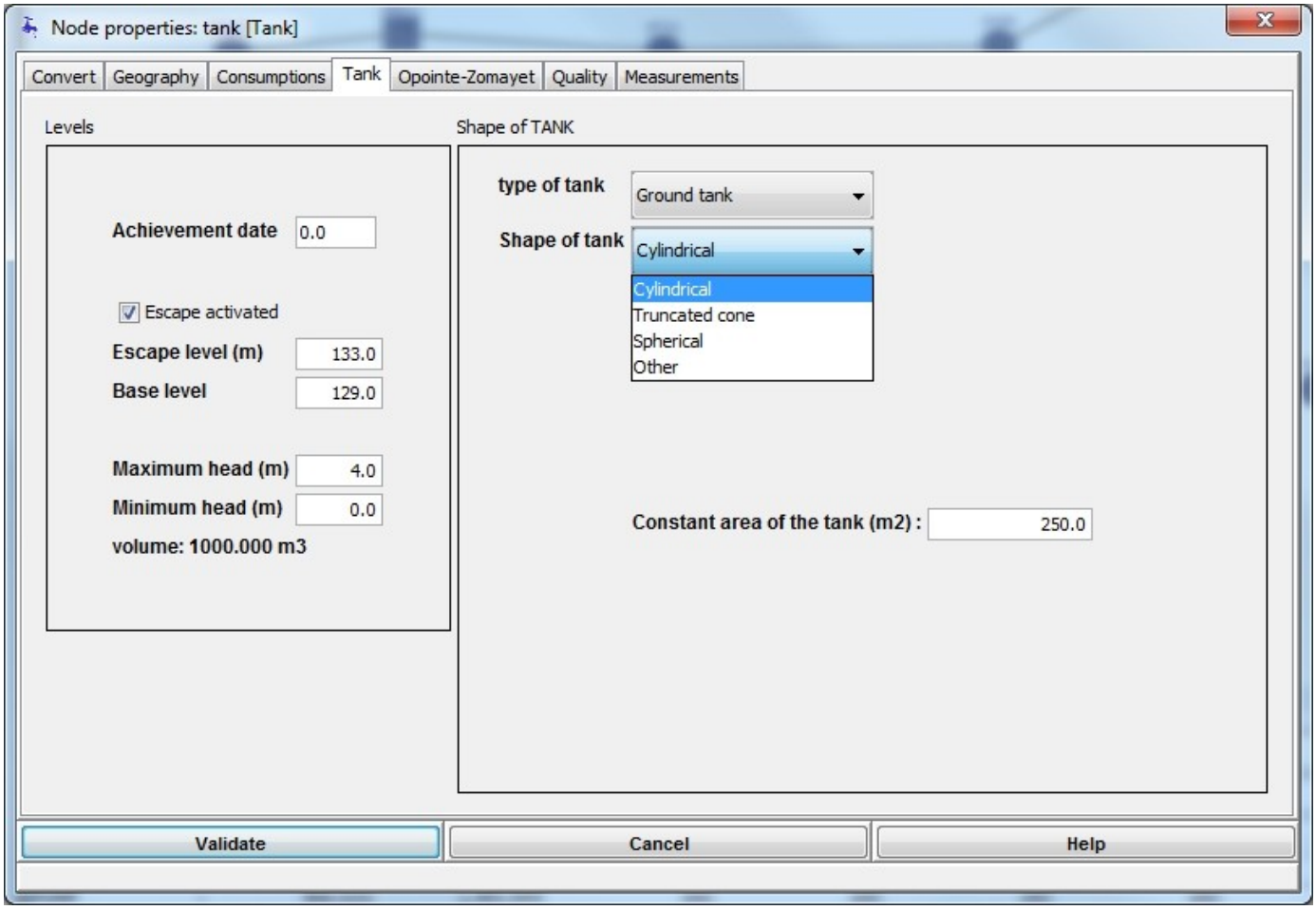
A tank node represents a storage tank which can hold one or several consumptions.
Typically, this is used to avoid simulating a flawless distribution network in a scheme of transportation between works. This network represents the downstream part of the works; in Zomayet it is not necessarily useful to handle it, but it is necessary to consider the information at the exit which consists of the sum of consumptions met for this downstream part.

Consumptions tab



The entering of consumption data for a tank node functions the same as for an **ordinary node**. However, only domestic and industrial model types can be used at this type of node.

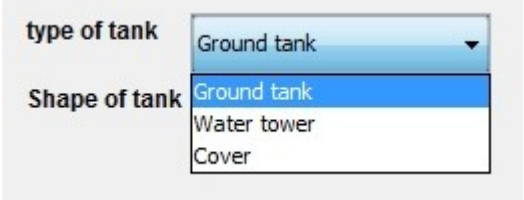
Tank Tab



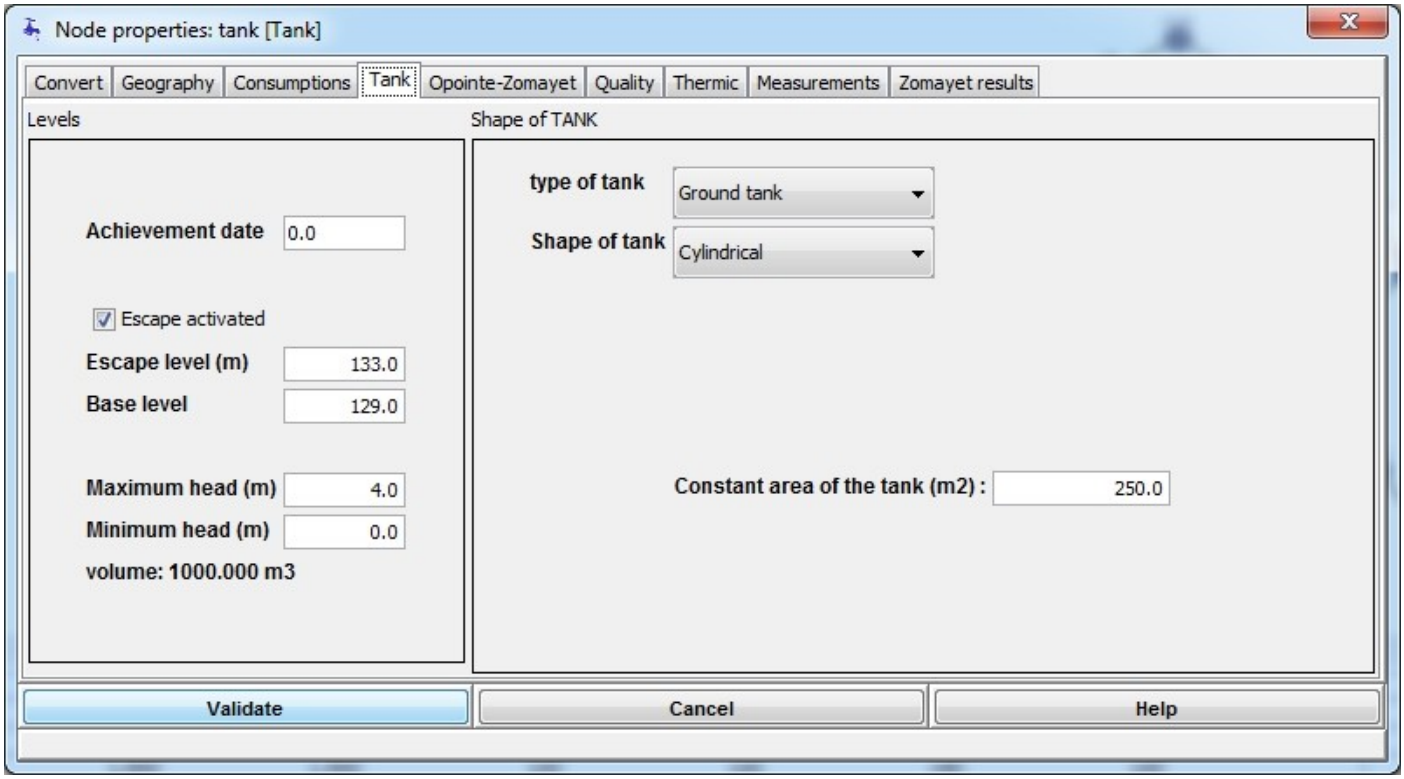
The tank at a tank node is described by attributes limited to the elevation head that can be reached by the water during the simulation and its shape.
The **"Achievement date"** field allows a date of end of construction to be entered but does not affect the calculations.

The checkbox allows the overflow of the tank to be simulated when the water in the tank reaches the overflow level.

If the checkbox is cleared, the level calculations don't have an upper limit.
The **"Overflow level"** field represents the height of water provoking an overflow.
The**"Base level"** field represents the height at the bottom of the tank. If the water reaches this level during the simulation, at the next time step the tank will not be allowed to deliver a flow.
The **"Maximum height"** field represents the maximum level of water fixed for exploitation purposes; it is only used in the graphical representation of results.
The **"Minimum height"** field represents the minimum level of water fixed for exploitation purposes; it is only used in the graphical representation of results.
The**volume** is automatically calculated following any modification of the various fields used in its calculation. It is calculated between the base and overflow levels.



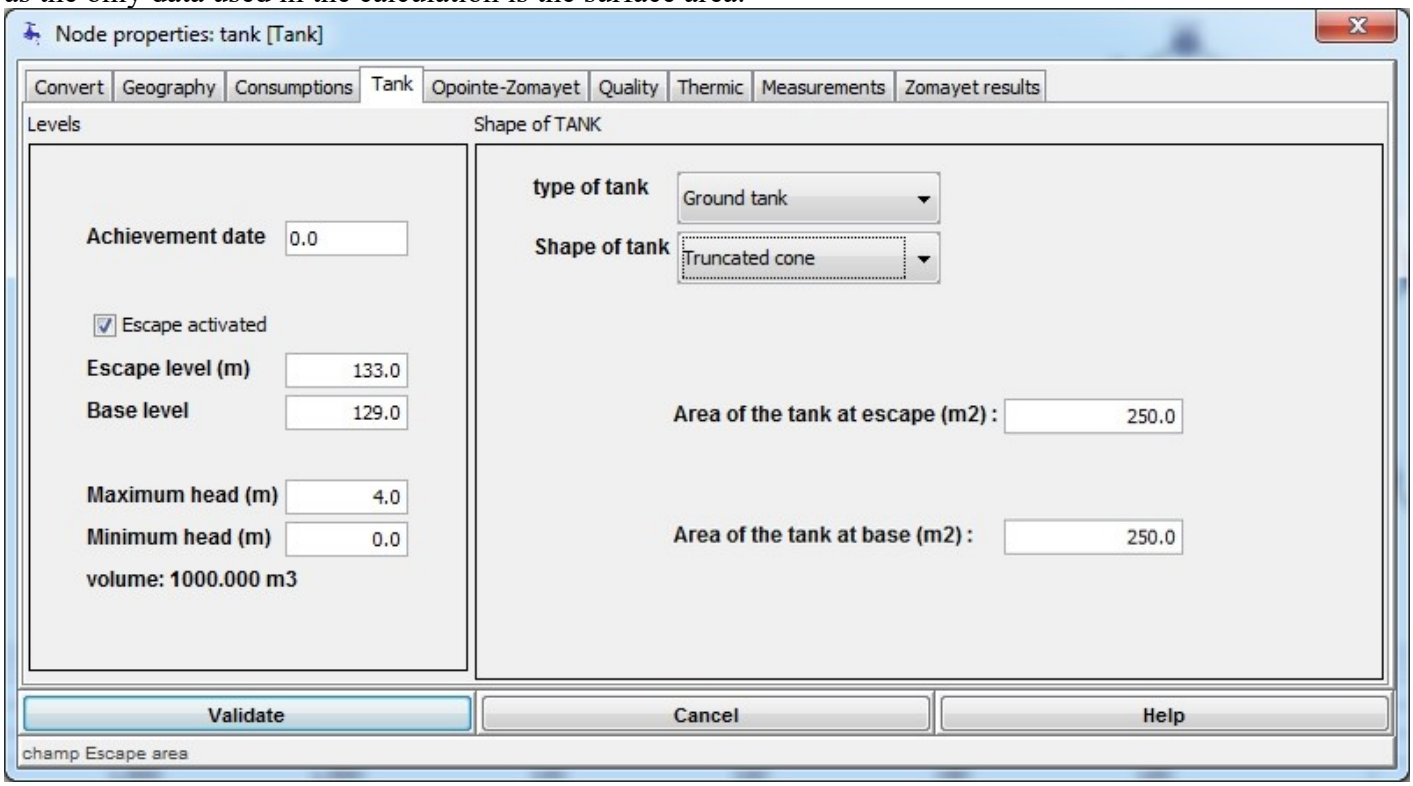
The choice of **tank type** is made from a scrolling list; this is not used in the calculations.



The choice of **tank shape** is made from a scrolling list; there are four possibilities:

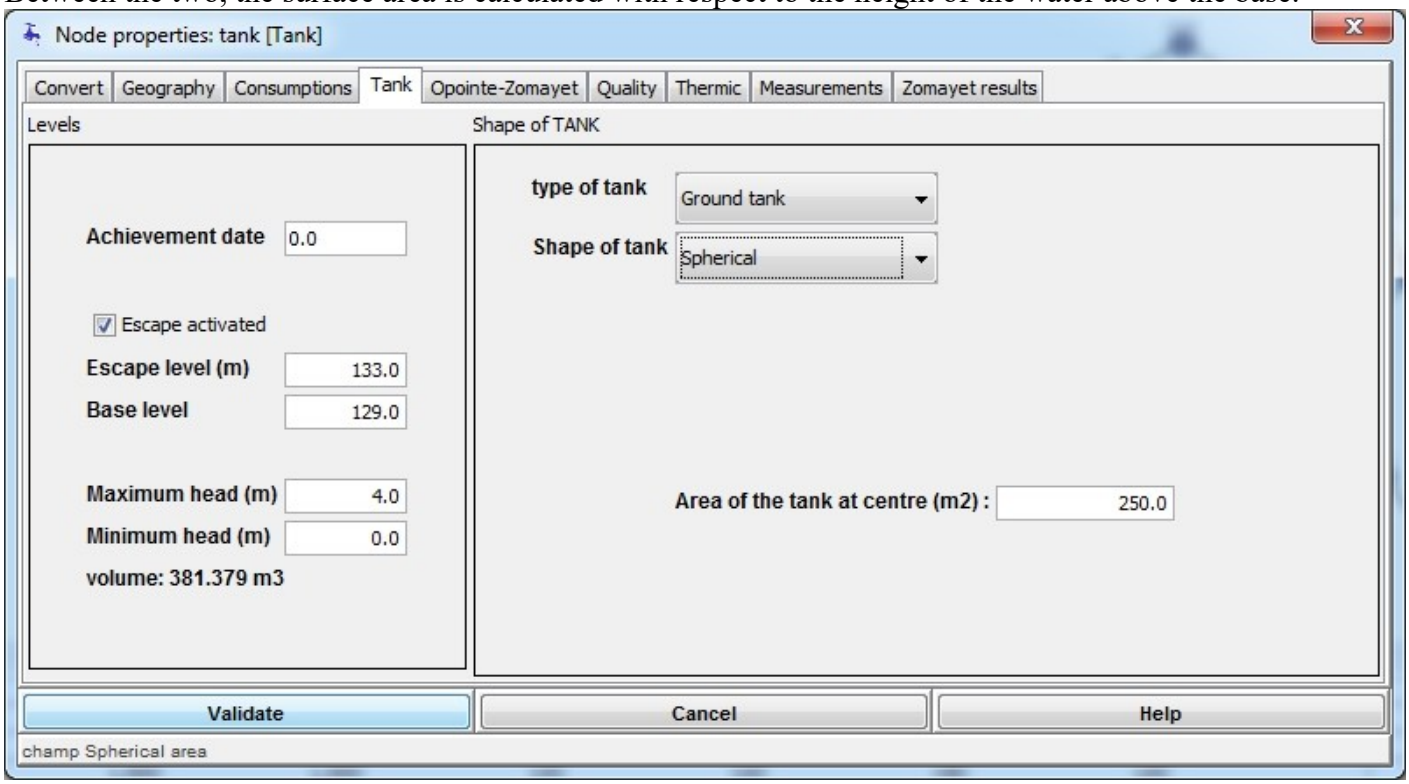
- Cylindrical
- Conical
- Spherical
- Other

For **cylindrical** tanks, the surface area of the tank is constant; the shape can be something else, e.g. cubic as the only data used in the calculation is the surface area.

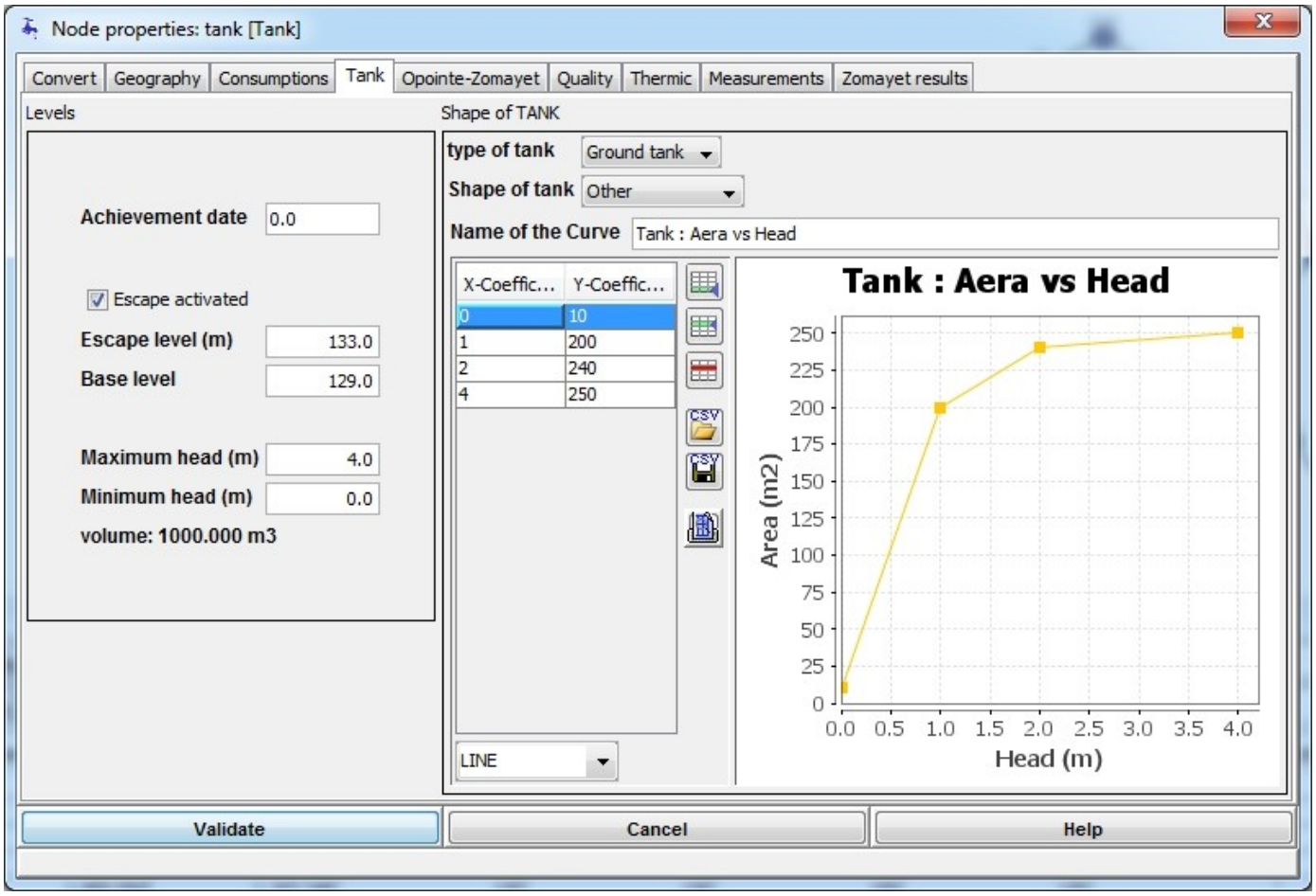


For **conical** tanks, two surface areas are used for the calculation – that at the base and that at the overflow.

Between the two, the surface area is calculated with respect to the height of the water above the base.

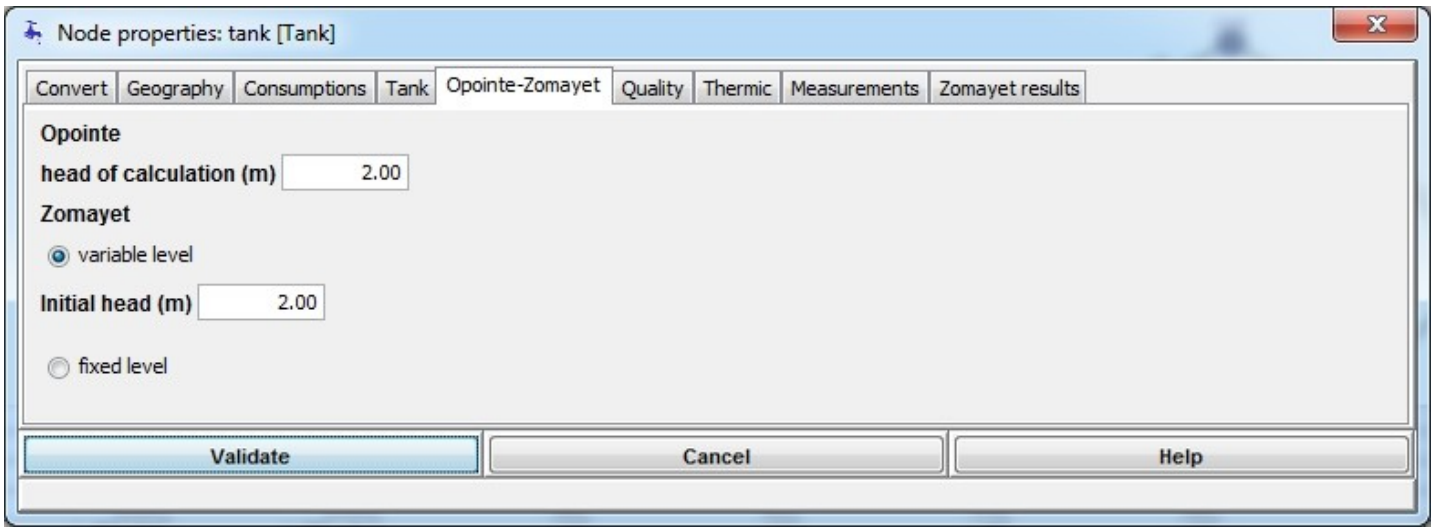


For **spherical** tanks, only the surface area at the centre of the tank is necessary for calculating the shape of the tank and its volume is a function of the calculated head.

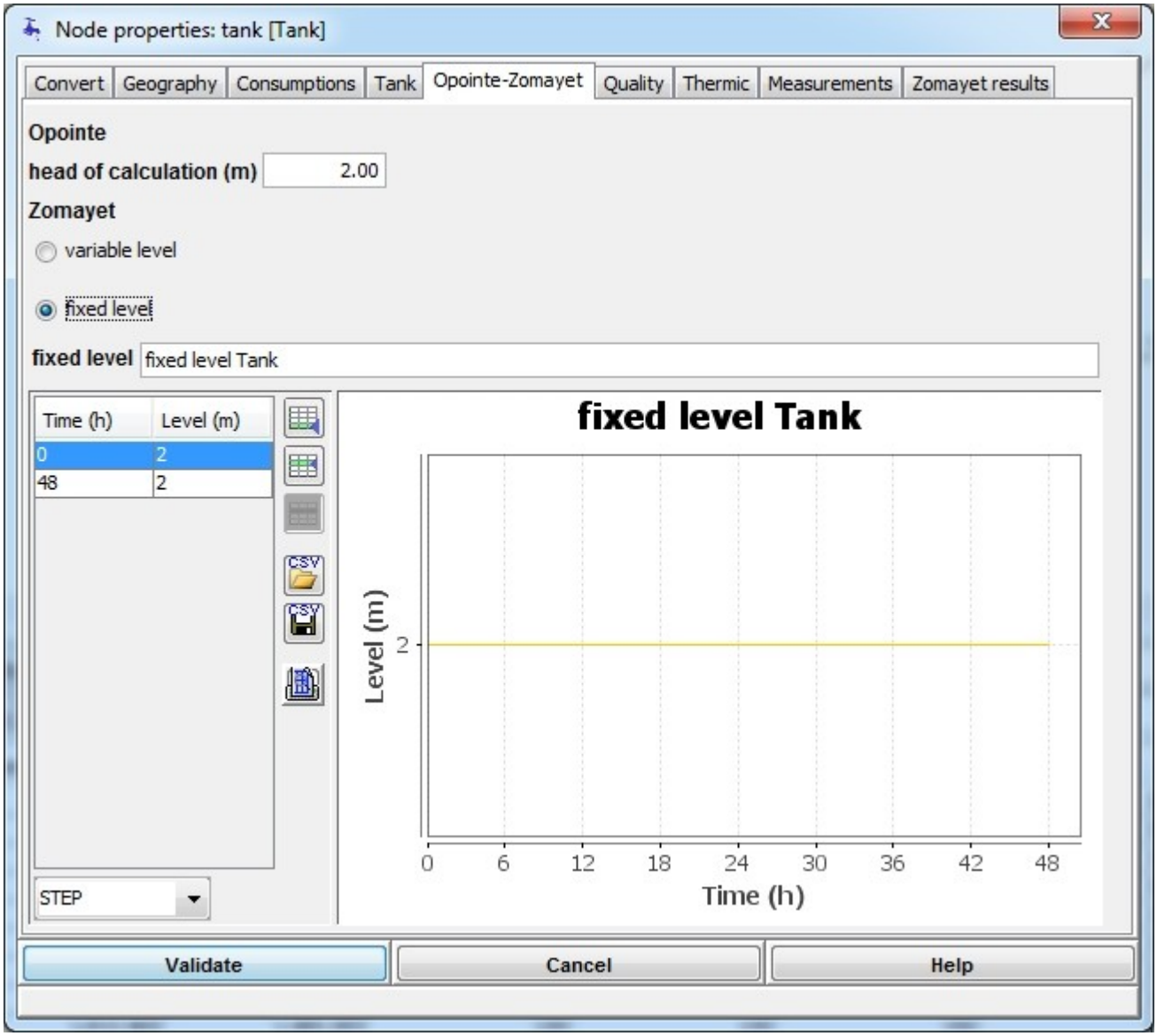


For**other** shapes, choose “other”. This allows the surface area to be entered as a function of the water level in the tank by pairs of values (height, area).

Opointe-Zomayet Tab

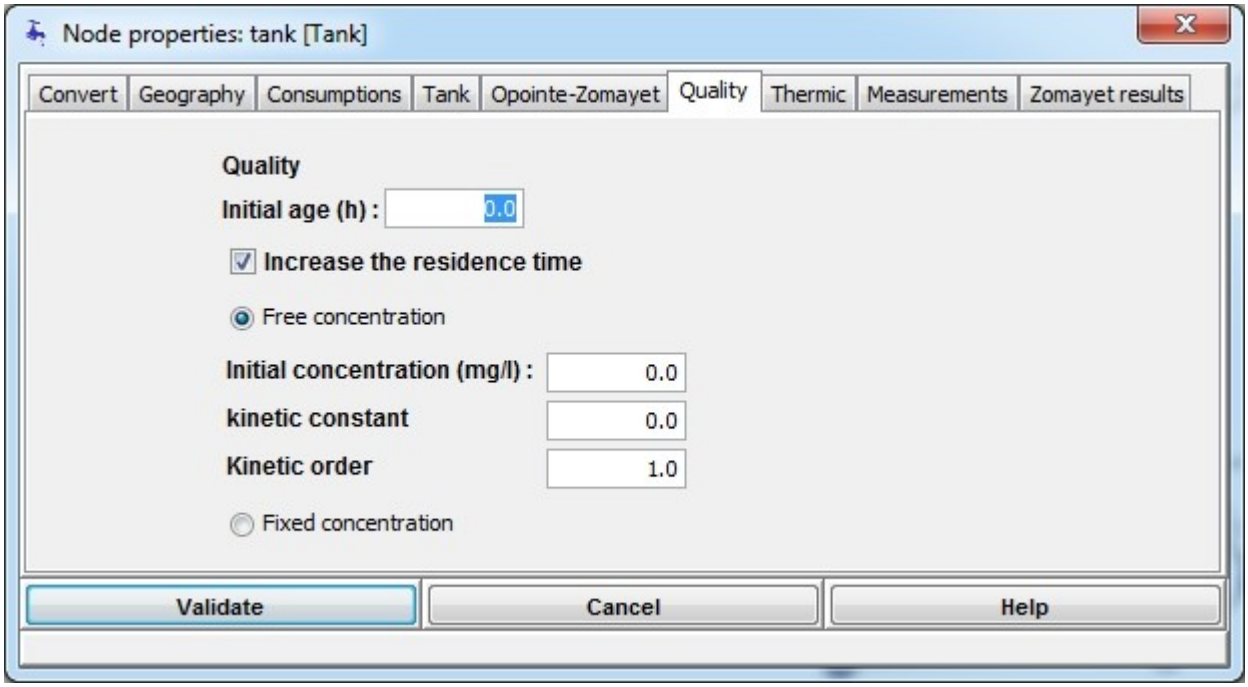


The conditions of initial head are fixed in the Opointe-Zomayet tab.
For Opointe, the calculation level is entered in the **height of calculation** field.
For Zomayet, two options are possible:
- If the variable level button is selected, the level is variable, only the initial level is given.
- If the fixed level button is selected, the level is imposed and is calculated following the entered graph as a function of time.



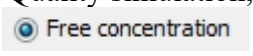
In the screen above, the level for Zomayet is imposed and varies linearly between 0 and 24 hours.

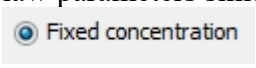
Quality tab



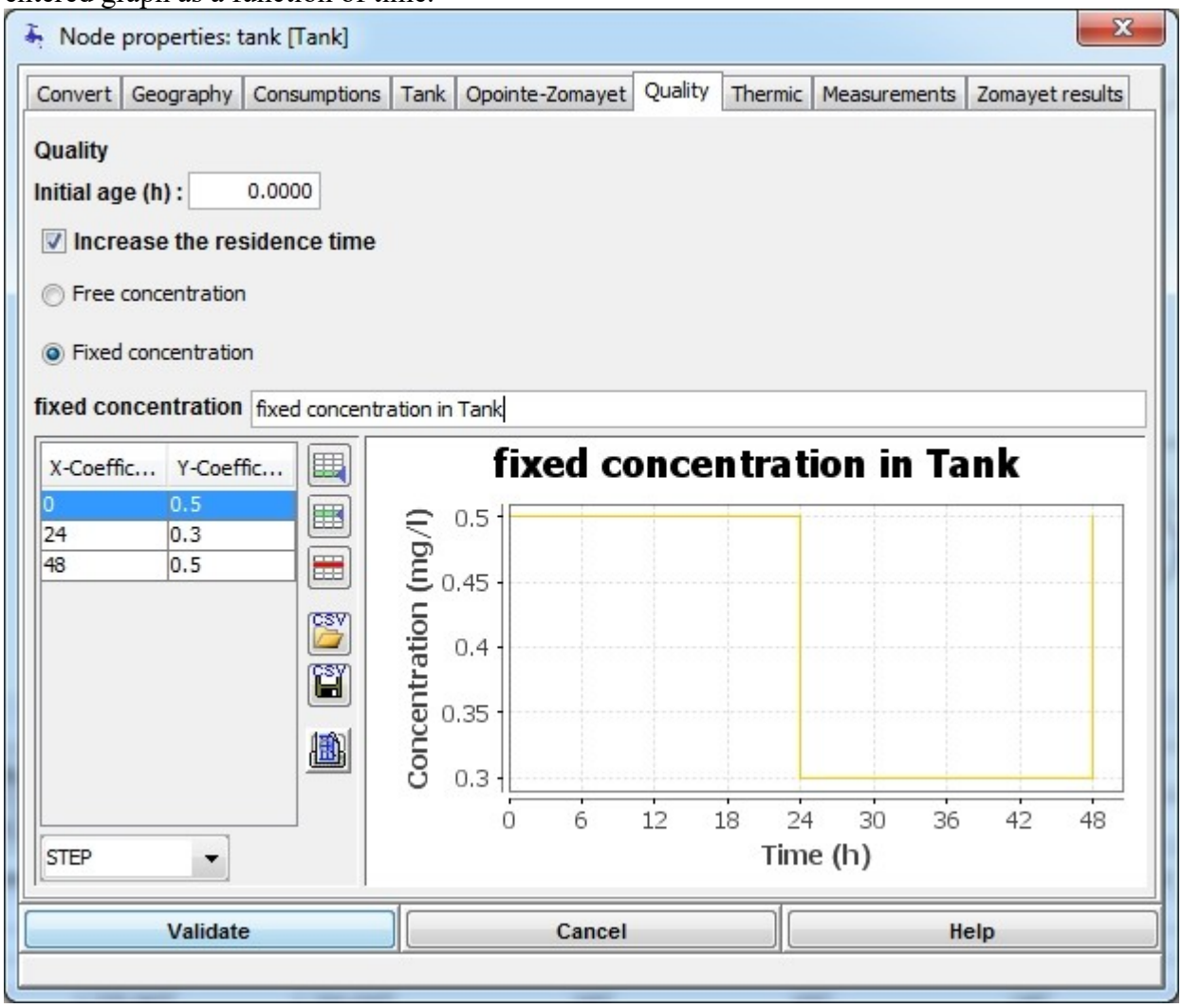
The **Initial age** field allows an initial age of the water other than zero for the calculation of residence time.

For the Quality simulation, two choices are possible:

- If the  button is selected, the simulated concentration is variable, only the initial level is given in the “Initial concentration” field; the “Kinetic constant” and “Kinetic order” fields contain the kinetic law parameters simulated in an evenly mixed tank.

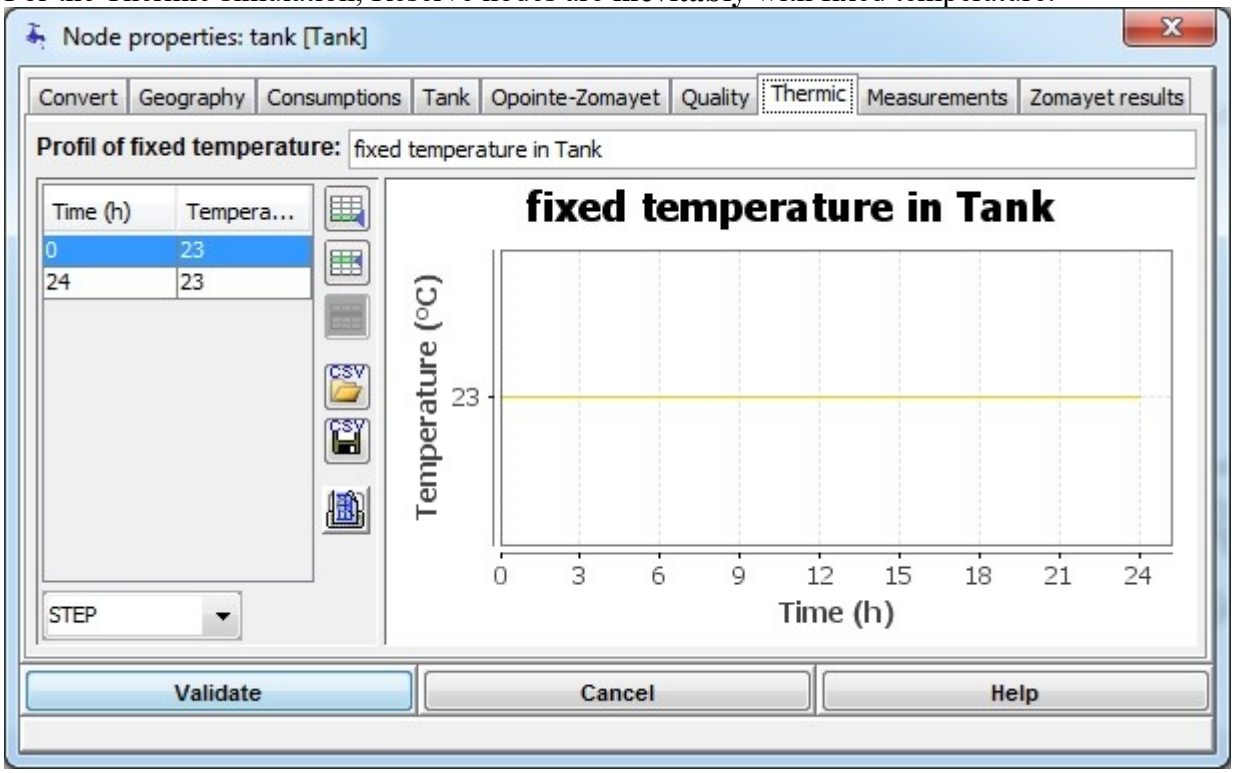
- If the  button is selected, the simulated concentration is calculated following the

entered graph as a function of time.



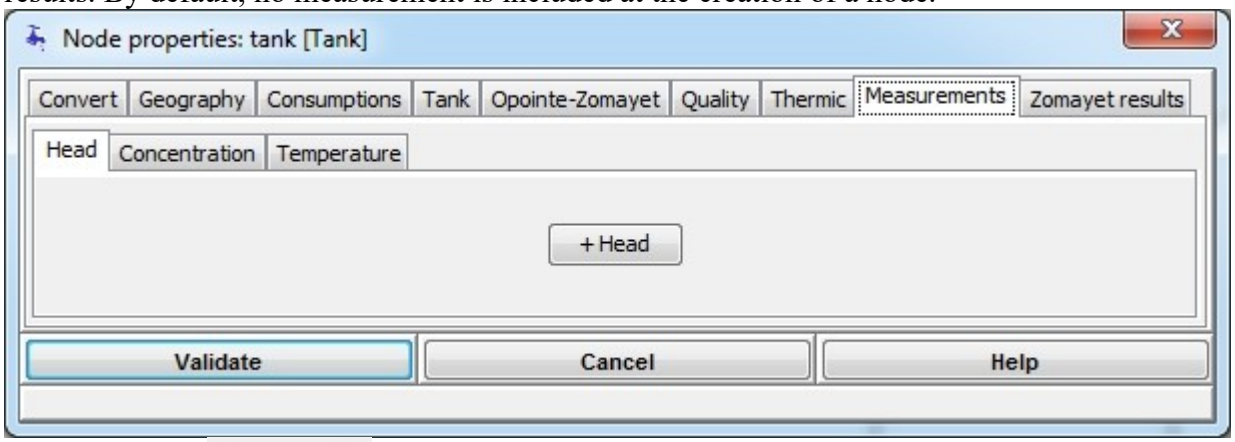
Onglet Thermic

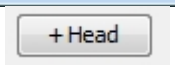
For the Thermic simulation, Reserve nodes are **inevitably** with fixed temperature:

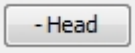


Measurements tab

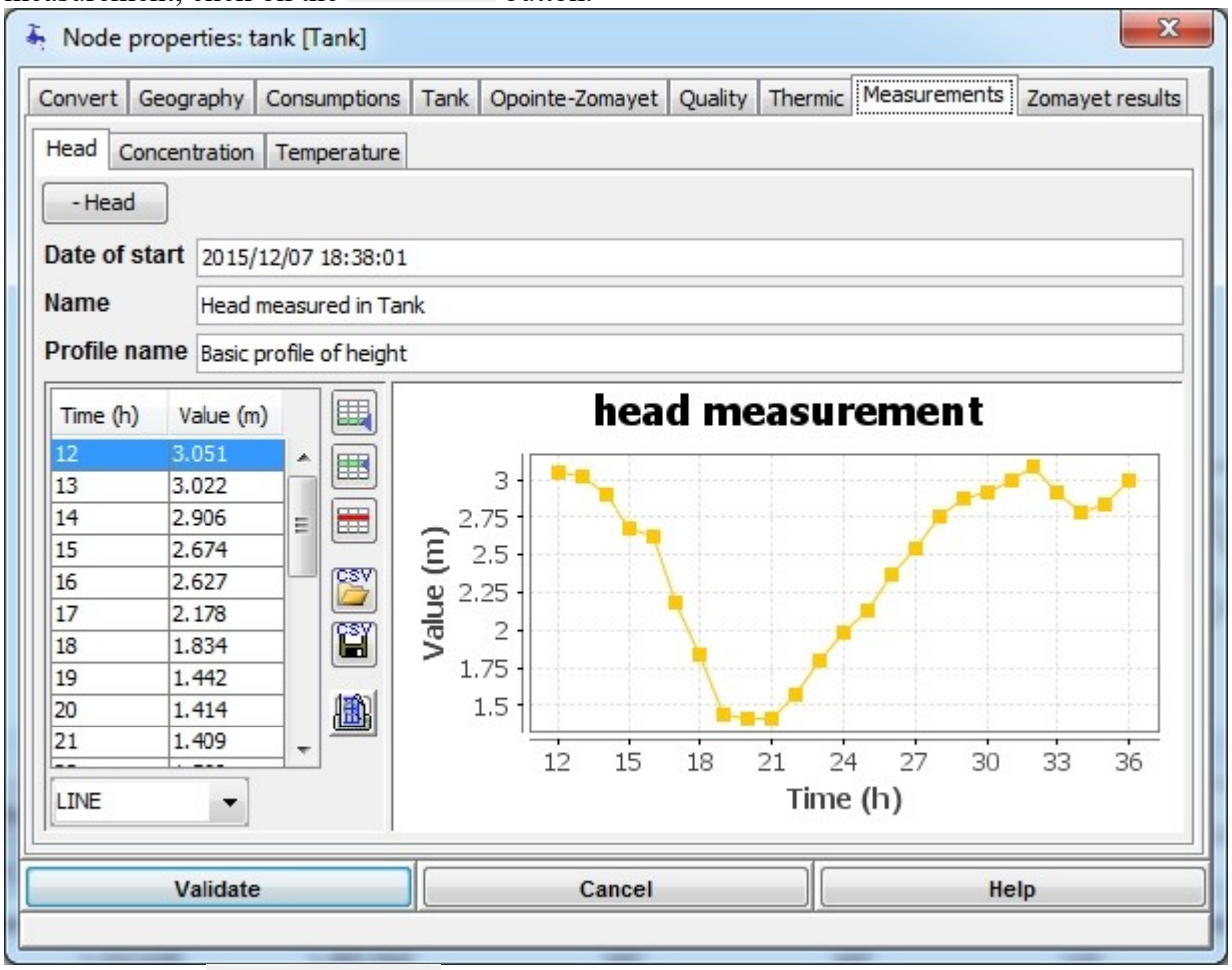
It is possible to enter measurement values at a tank node. These will be superimposed on to the simulation results. By default, no measurement is included at the creation of a node.

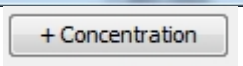


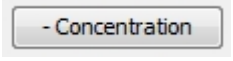
By clicking the 

button, the dialog box for entering a height level is shown. To delete the measurement, click on the 

button.



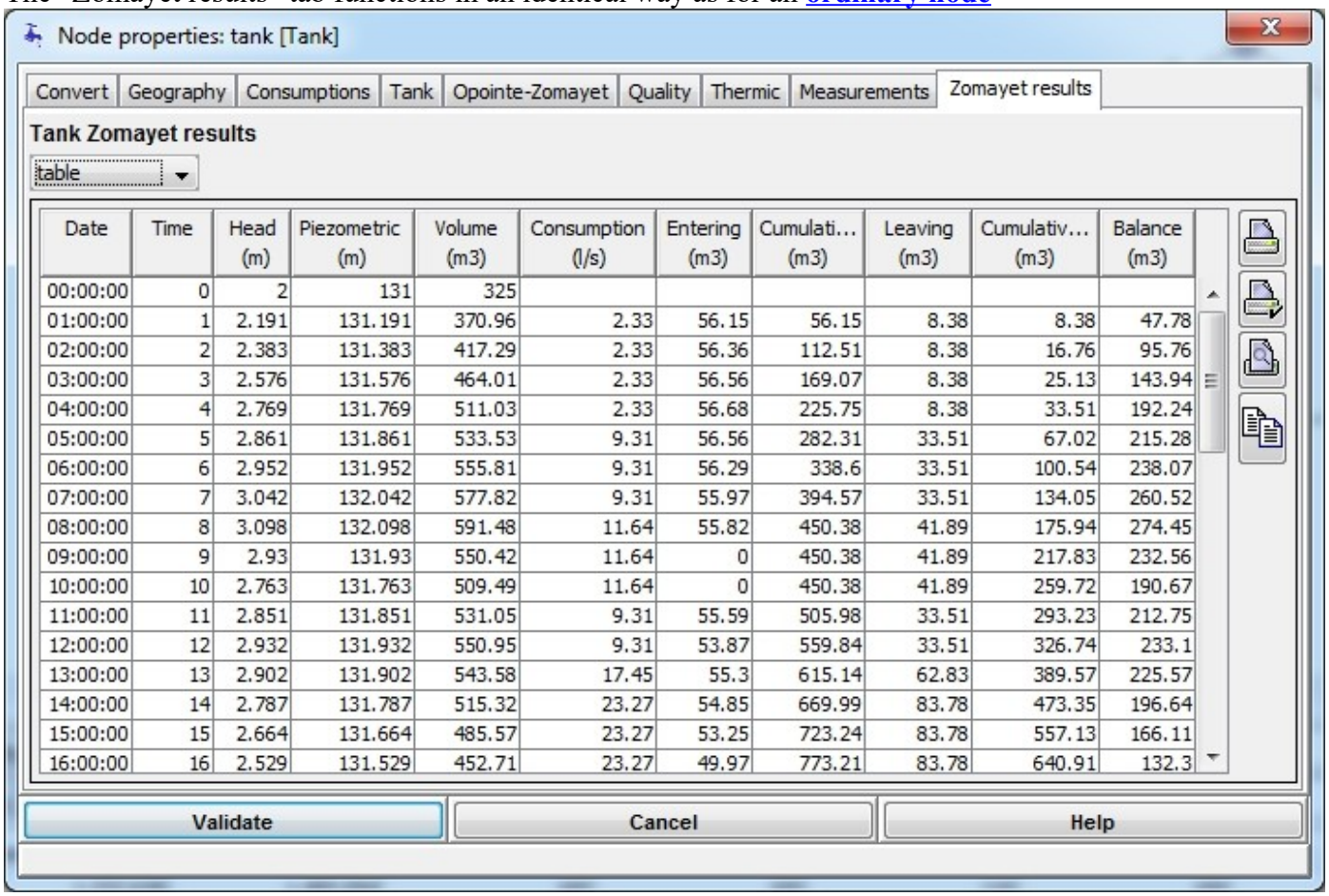
By clicking the 

button, the dialog box for entering a concentration measurement is shown. To delete a concentration measurement, click on the 

button.
This is the same functionning as the temperature measurement.

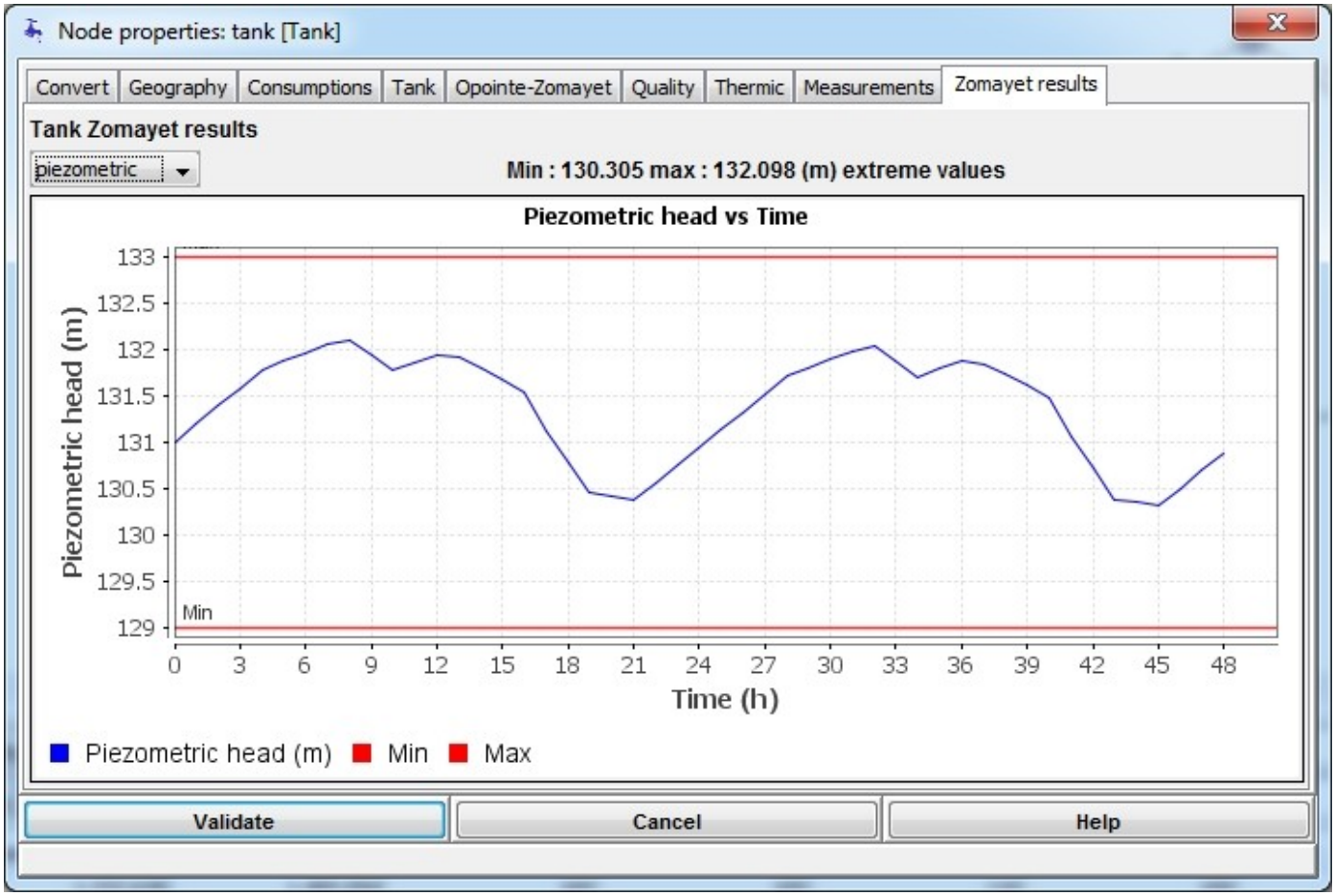
Zomayet Results Tab

The “Zomayet results” tab functions in an identical way as for an **ordinary node**

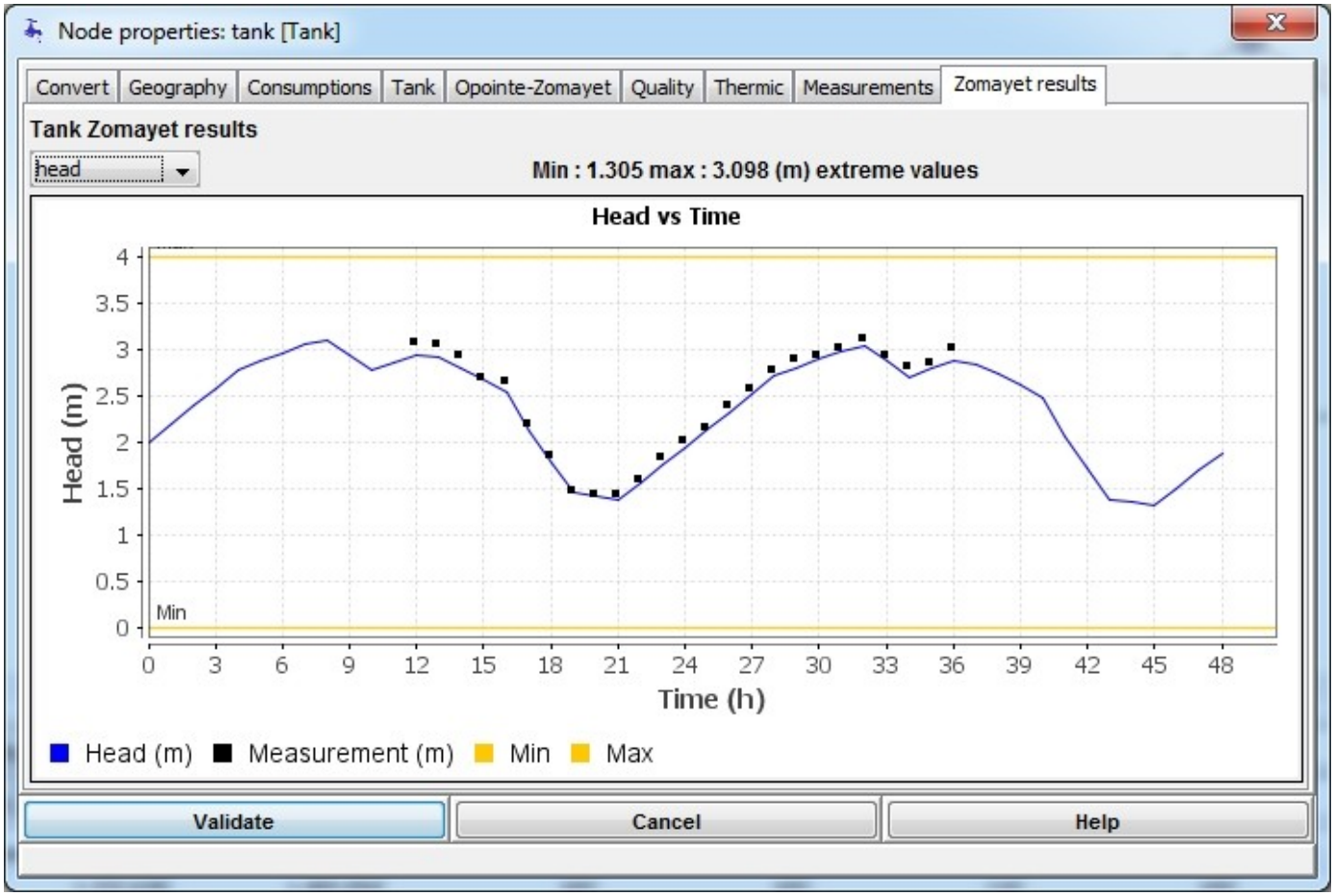


Date	Time	Head (m)	Piezometric (m)	Volume (m3)	Consumption (l/s)	Entering (m3)	Cumulati... (m3)	Leaving (m3)	Cumulativ... (m3)	Balance (m3)
00:00:00	0	2	131	325						
01:00:00	1	2.191	131.191	370.96	2.33	56.15	56.15	8.38	8.38	47.78
02:00:00	2	2.383	131.383	417.29	2.33	56.36	112.51	8.38	16.76	95.76
03:00:00	3	2.576	131.576	464.01	2.33	56.56	169.07	8.38	25.13	143.94
04:00:00	4	2.769	131.769	511.03	2.33	56.68	225.75	8.38	33.51	192.24
05:00:00	5	2.861	131.861	533.53	9.31	56.56	282.31	33.51	67.02	215.28
06:00:00	6	2.952	131.952	555.81	9.31	56.29	338.6	33.51	100.54	238.07
07:00:00	7	3.042	132.042	577.82	9.31	55.97	394.57	33.51	134.05	260.52
08:00:00	8	3.098	132.098	591.48	11.64	55.82	450.38	41.89	175.94	274.45
09:00:00	9	2.93	131.93	550.42	11.64	0	450.38	41.89	217.83	232.56
10:00:00	10	2.763	131.763	509.49	11.64	0	450.38	41.89	259.72	190.67
11:00:00	11	2.851	131.851	531.05	9.31	55.59	505.98	33.51	293.23	212.75
12:00:00	12	2.932	131.932	550.95	9.31	53.87	559.84	33.51	326.74	233.1
13:00:00	13	2.902	131.902	543.58	17.45	55.3	615.14	62.83	389.57	225.57
14:00:00	14	2.787	131.787	515.32	23.27	54.85	669.99	83.78	473.35	196.64
15:00:00	15	2.664	131.664	485.57	23.27	53.25	723.24	83.78	557.13	166.11
16:00:00	16	2.529	131.529	452.71	23.27	49.97	773.21	83.78	640.91	132.3

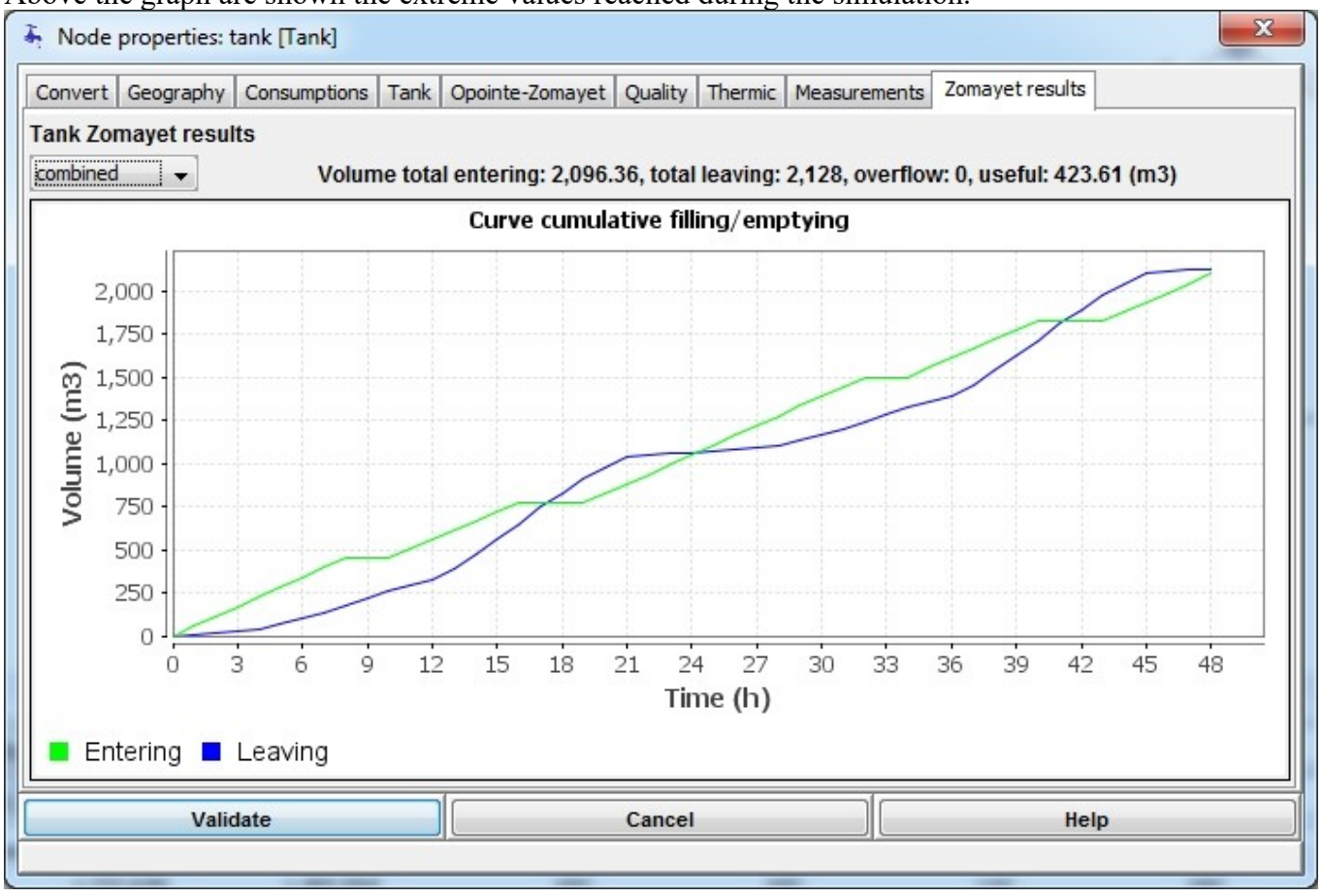
The choice **table** allows all the numerical values at the node calculated by Zomayet to be consulted. The first line only contains the initial values. Then the values entering and leaving the tank and their relative cumulative values can be calculated by time step. Finally, the balance is deduced.



The choice **piezometric** produces a graph of the piezometric head imposed by the tank during the simulation.
The graph is scaled using the minimum and maximum heads chosen for the tank. Above the graph are shown the extreme values reached during the simulation.



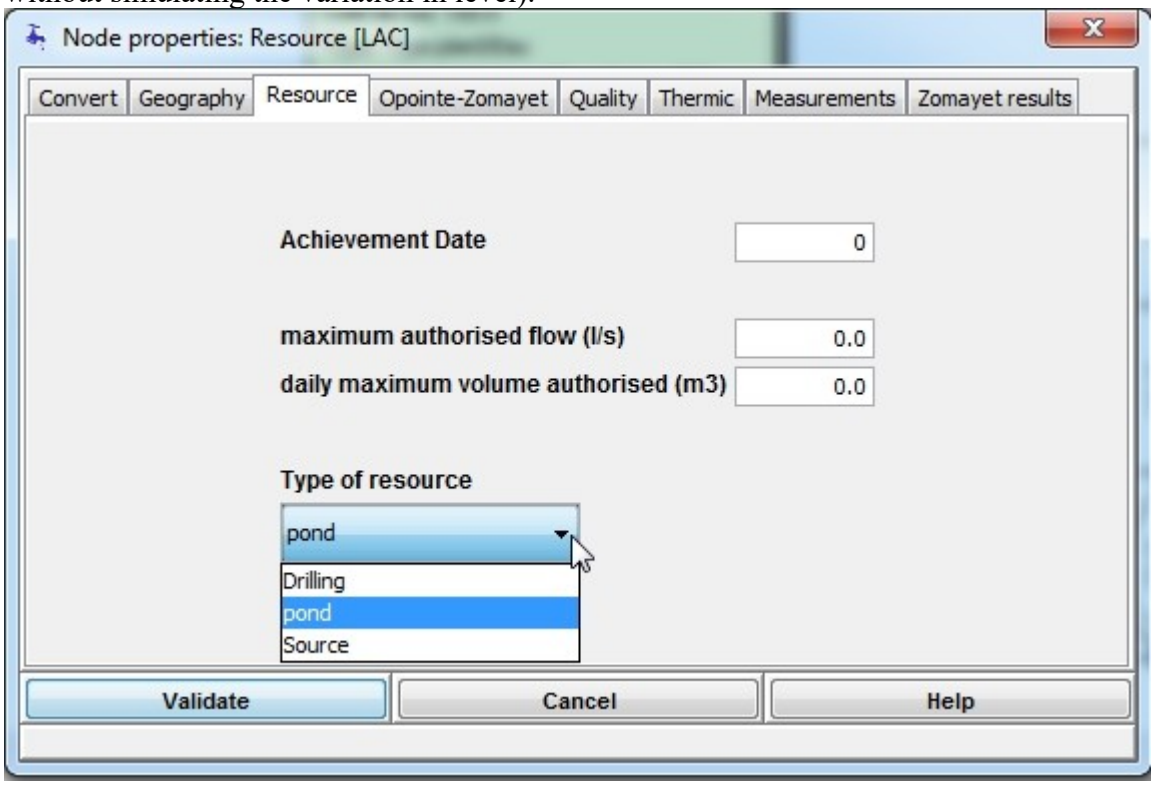
The choice **head** shows the evolution of water level simulated (or imposed). The graph is scaled using the minimum and maximum height levels chosen for the tank.
Above the graph are shown the extreme values reached during the simulation.



The choice **"Combined"** shows the cumulative volumes entering and leaving the tank, this allows from the sum of the two largest differences to calculate the volume used in the simulation.
Above the graph the numerical values for the balance are shown.

Resource Node

A resource node can be used to model a water resource, be it real (source, water table) or fictive (reservoir without simulating the variation in level).



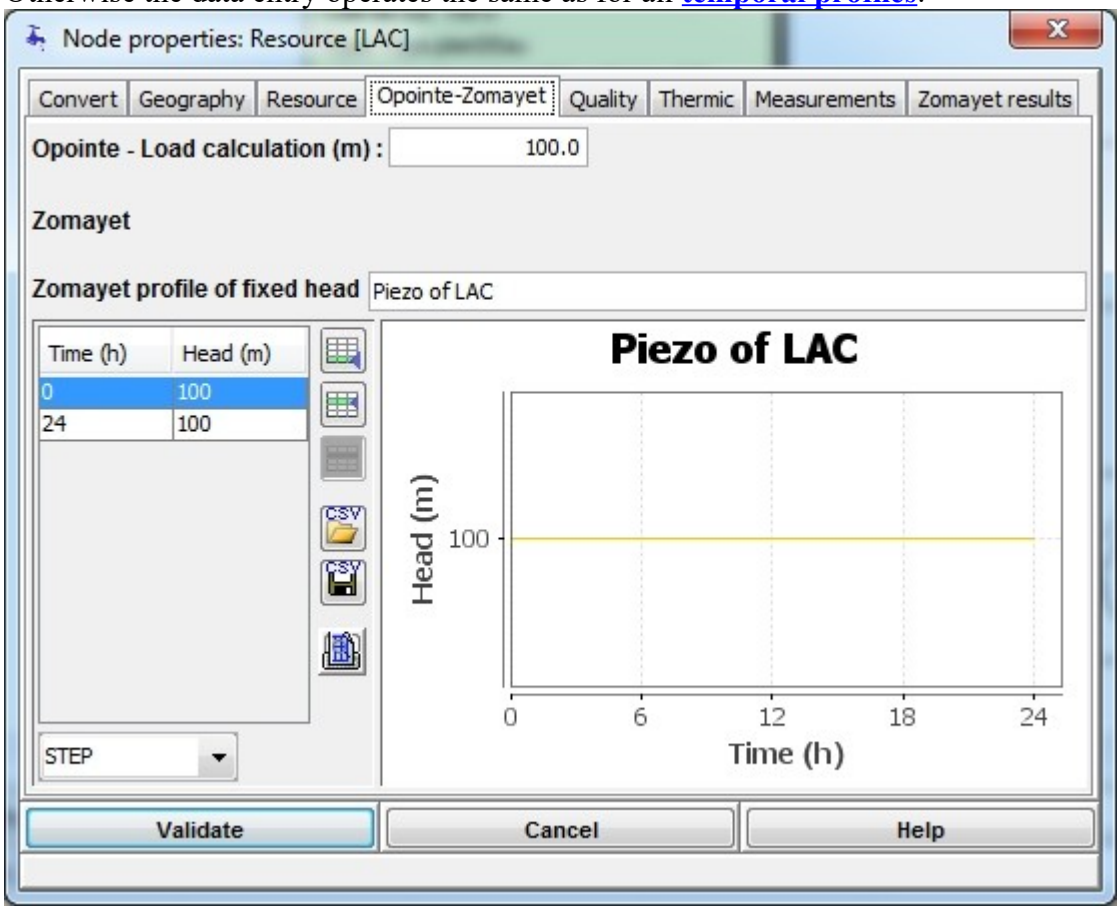
The "resource" tab enables data to be entered that is not used in the calculations but can help the user in his approach.

The **achievement date** field allows the construction year to be entered.
The **maximum authorised flow** field allows the threshold for the regulated authorised instantaneous withdrawal to be recorded.
The**maximum authorised daily volume** field allows the threshold for the regulated authorised daily withdrawal to be recorded.
The **type of resource** scroll list allows the type of resource to be modelled to be chosen.

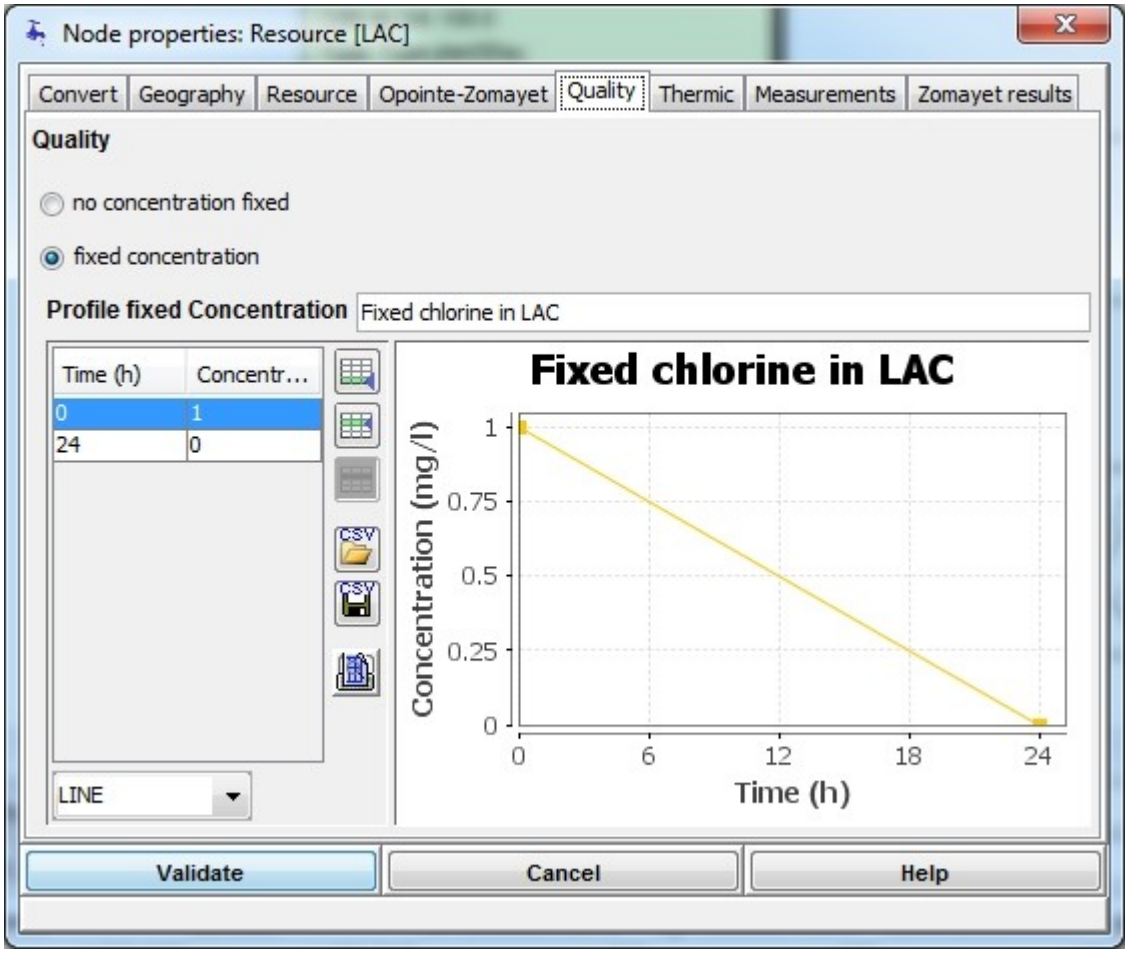
Opointe-Zomayet Tab

The conditions of initial head are fixed in the "Opointe-Zomayet" tab.
For Opointe the head for the calculation is entered in the **Opointe-calculation head** field.
For Zomayet, a graph of the evolution of the head as a function of time is entered as a profile.
To obtain a constant head, two values defining the simulation length are indicated.

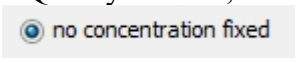
Otherwise the data entry operates the same as for all **temporal profiles**.



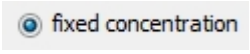
Quality Tab



For the Quality module, the conditions at the limits of the system must be indicated.

If the 

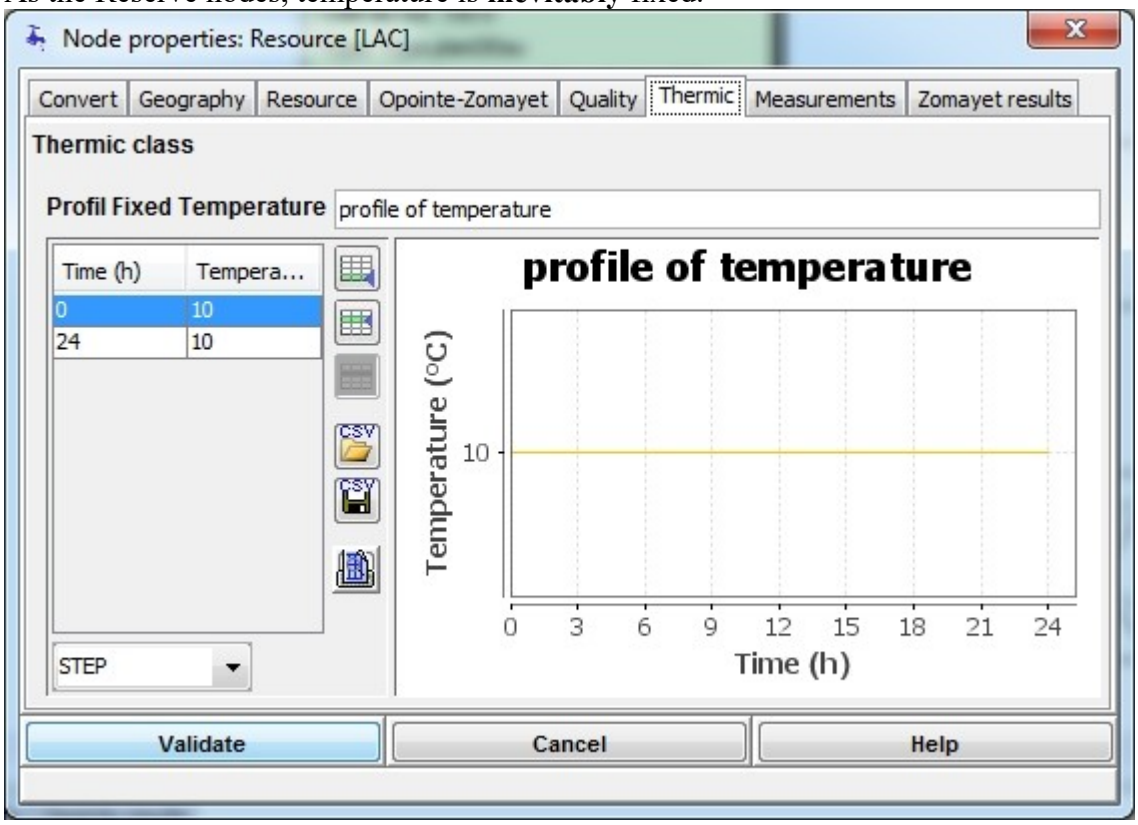
button is selected, the concentration is zero for all the simulation.

If the 

button is selected, a concentration profile describes the evolution of the concentration during the simulation. If the duration of the profile is lower than the duration of the simulation, it is reproduced from scratch as many times as necessary.

Onglet Thermic

As the Reserve nodes, temperature is **inevitably** fixed.

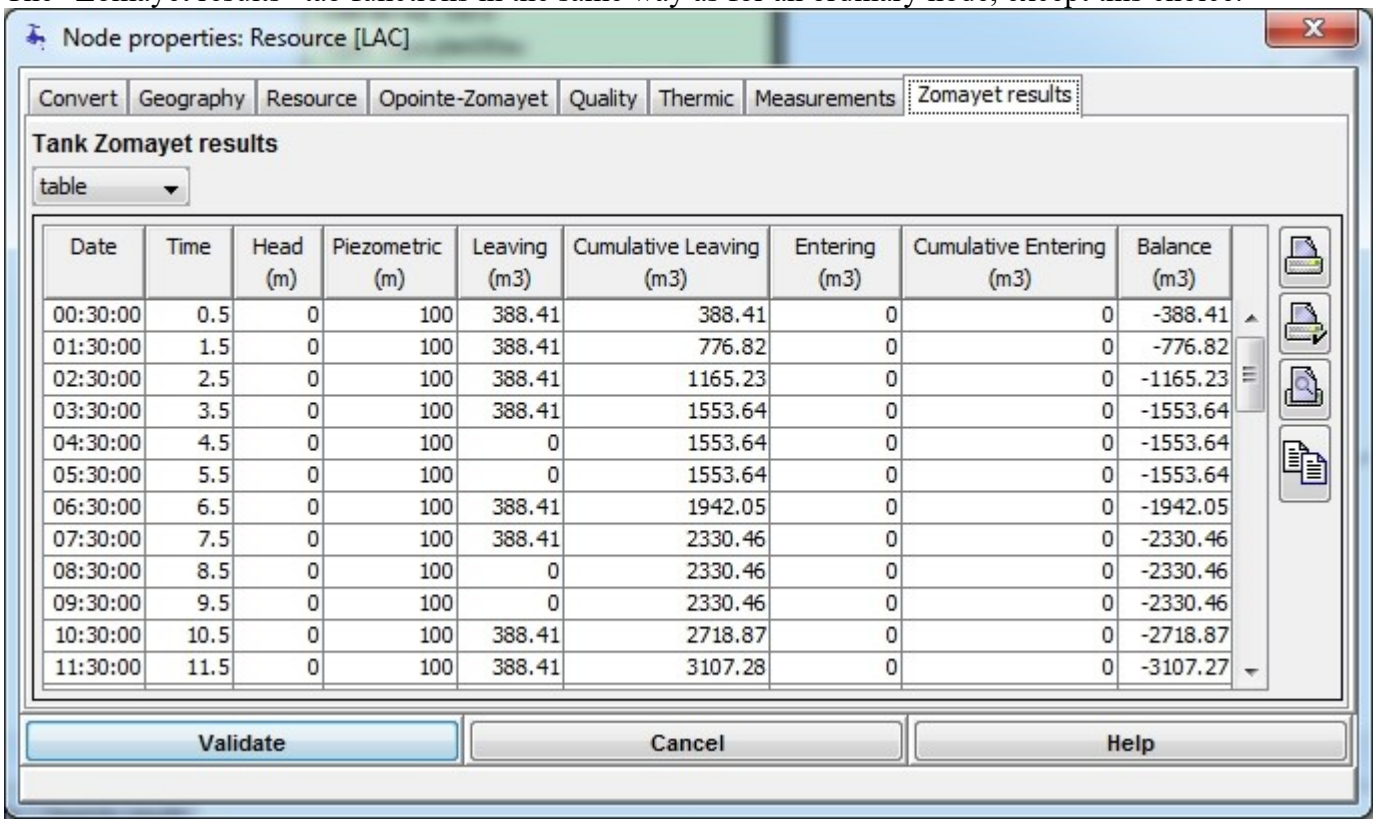


Measurements Tab

For information, a **concentration and/or temperature measurement** can be entered at a resource node.

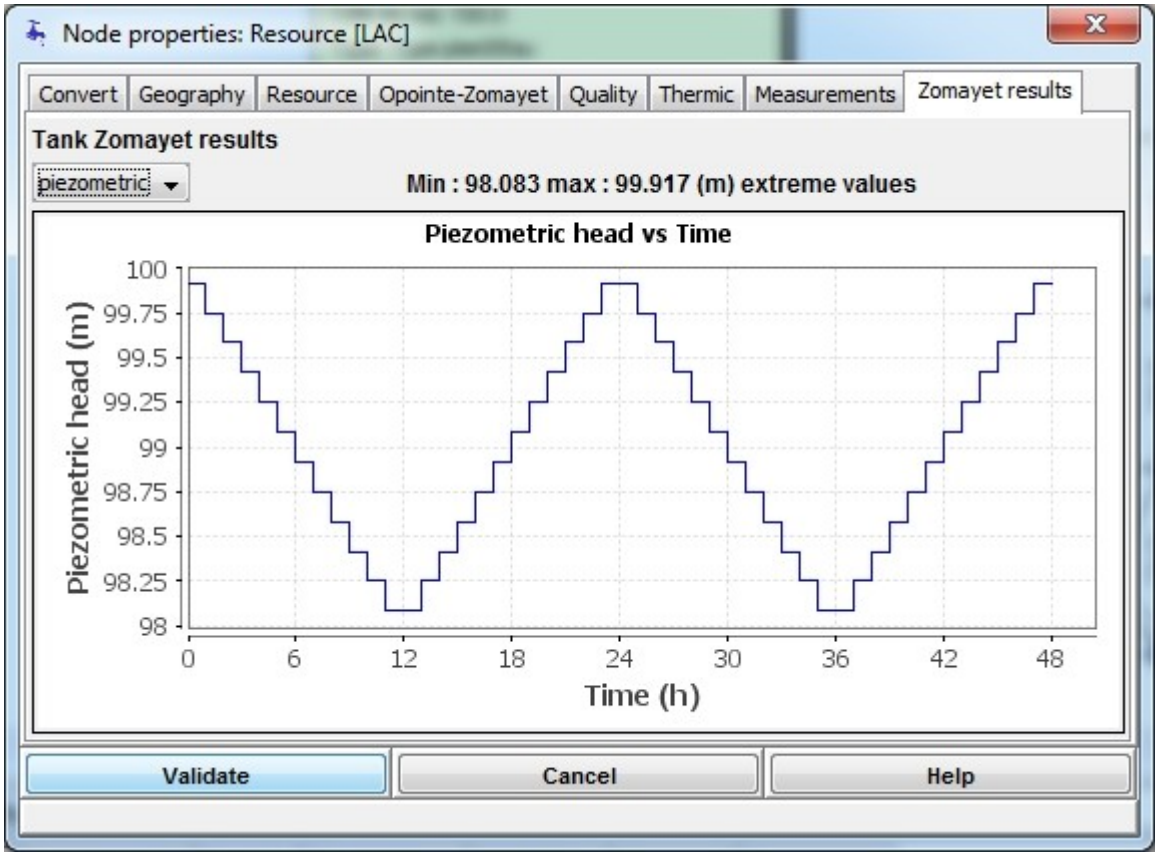
Zomayet Results Tab

The “Zomayet results” tab functions in the same way as for an ordinary node, except this choice.

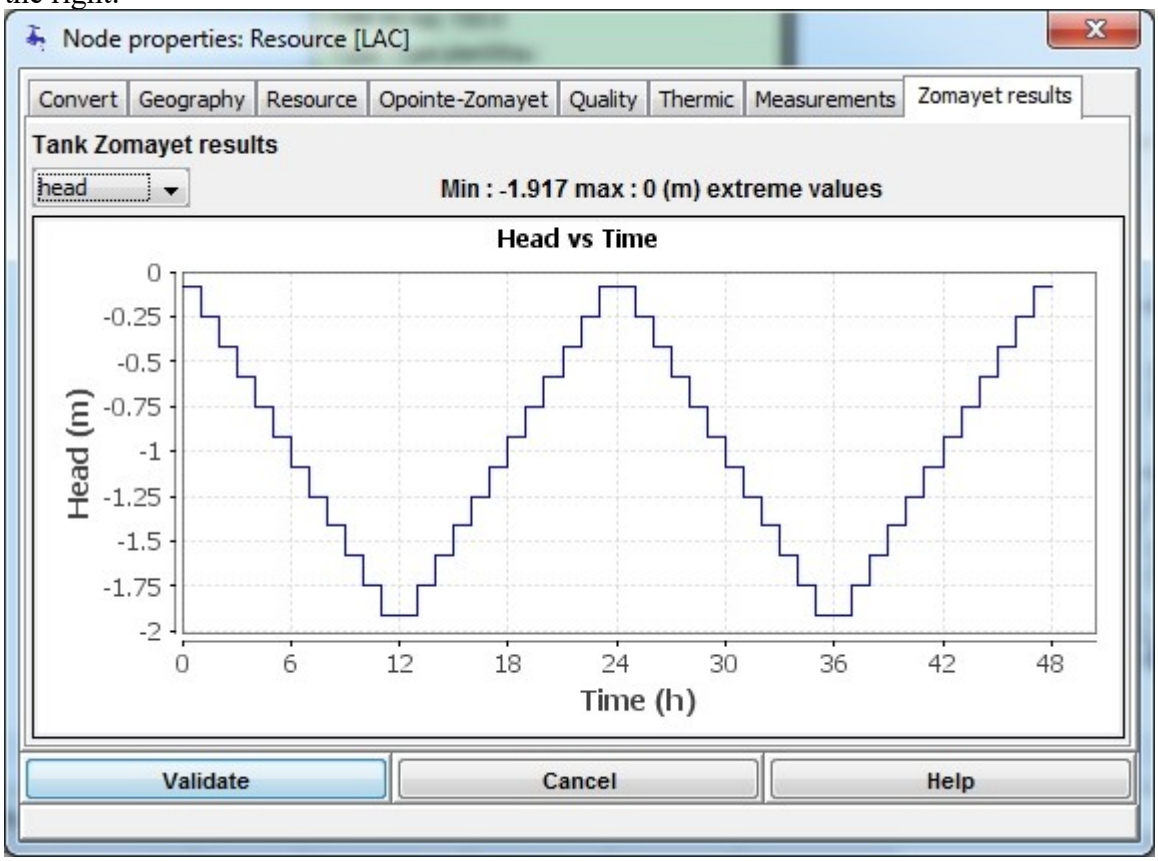


Date	Time	Head (m)	Piezometric (m)	Leaving (m3)	Cumulative Leaving (m3)	Entering (m3)	Cumulative Entering (m3)	Balance (m3)
00:30:00	0.5	0	100	388.41	388.41	0	0	-388.41
01:30:00	1.5	0	100	388.41	776.82	0	0	-776.82
02:30:00	2.5	0	100	388.41	1165.23	0	0	-1165.23
03:30:00	3.5	0	100	388.41	1553.64	0	0	-1553.64
04:30:00	4.5	0	100	0	1553.64	0	0	-1553.64
05:30:00	5.5	0	100	0	1553.64	0	0	-1553.64
06:30:00	6.5	0	100	388.41	1942.05	0	0	-1942.05
07:30:00	7.5	0	100	388.41	2330.46	0	0	-2330.46
08:30:00	8.5	0	100	0	2330.46	0	0	-2330.46
09:30:00	9.5	0	100	0	2330.46	0	0	-2330.46
10:30:00	10.5	0	100	388.41	2718.87	0	0	-2718.87
11:30:00	11.5	0	100	388.41	3107.28	0	0	-3107.27

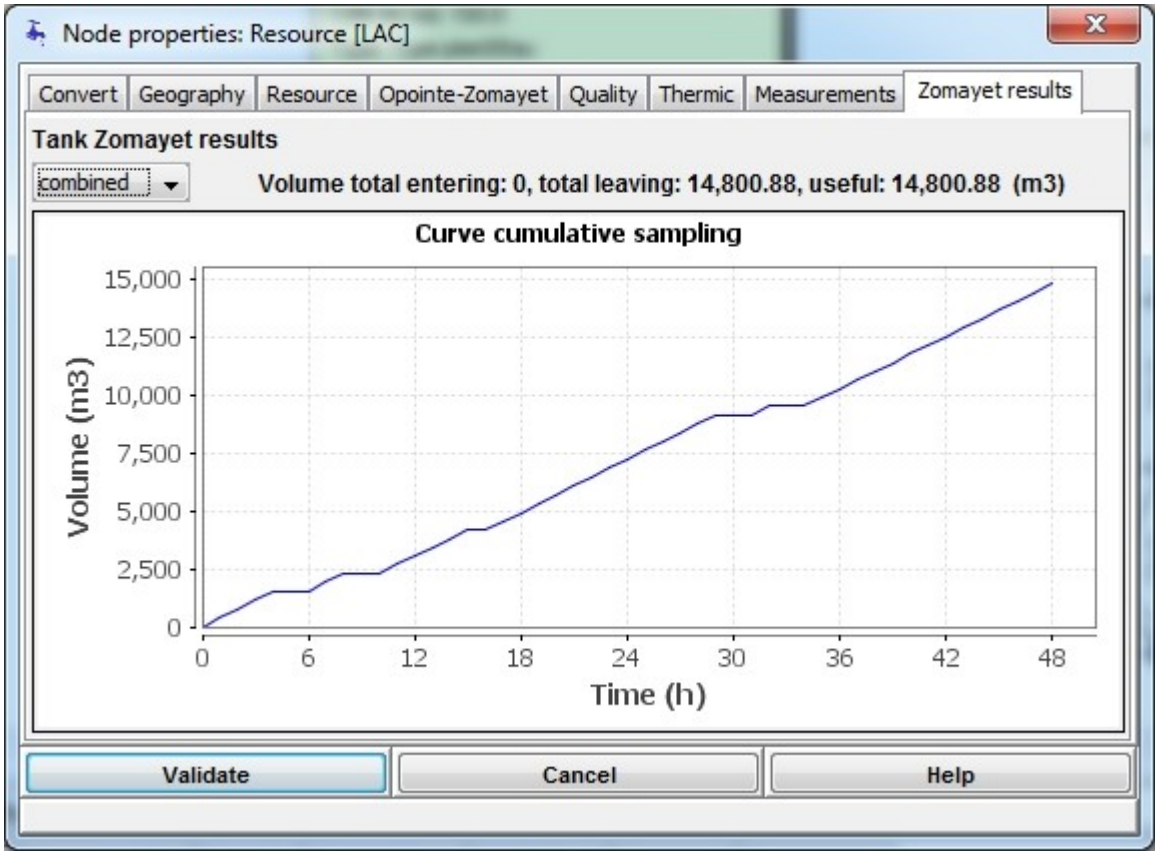
The **table** choice functions the same way as for a tank node.
The inflow has been left in the eventuality that a simulation would end in a return of water towards to the resource.



The **piezo** choice redraws the head level imposed by the data for which the ground level is represented by the right.



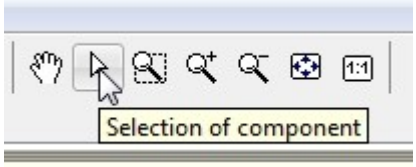
The **head** choice redraws the pressure level imposed calculated relative to the ground level.
Above the chart, extreme values reached during the simulation are displayed.



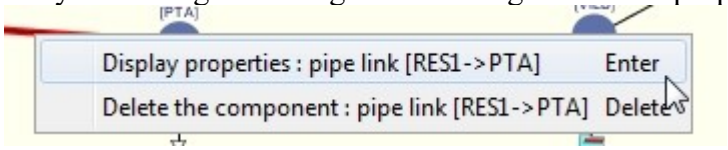
The **combined** choice functions as for a tank node and allows the inflow/outflow balance to be calculated for the node during the simulation.

Pipe link

This dialog box is accessible by double left clicking on a pipe segment in Schematic or Map View with the “Select” cursor:



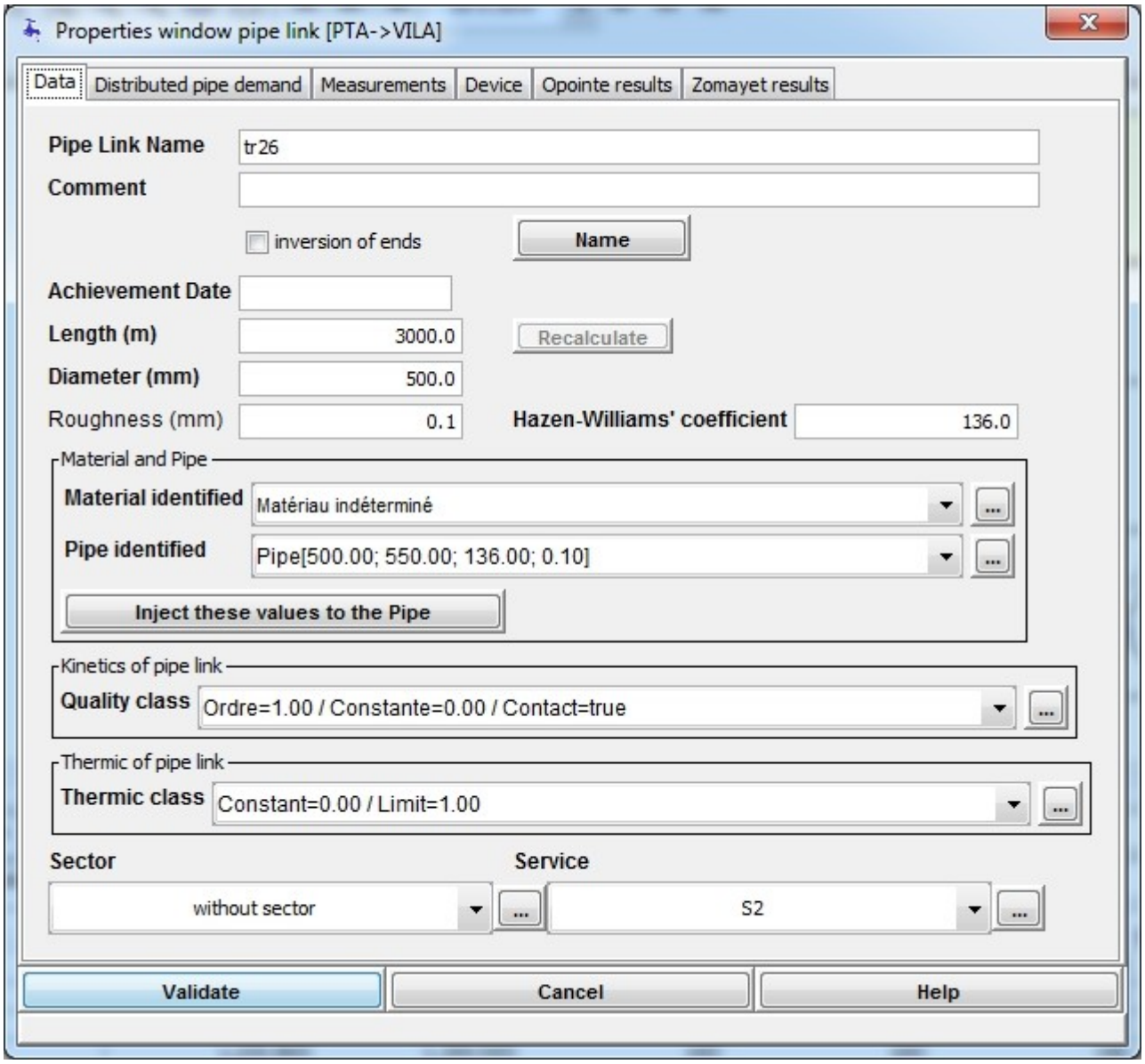
or by double right clicking and choosing the “Show properties: pipe […->…]” from the contextual menu:



The data entered at this level of the dialog box can be either essential, ie mandatory for the calculations, or supplementary, and not required for the calculation runs. The dialog box title states the name of the pipe by associating it with its two nodes.
The “Properties window: pipe […-> …]” starts with the “Data” tab.
Reminder: Table match the old 8 standard **roughnesses** of Porteau 2.

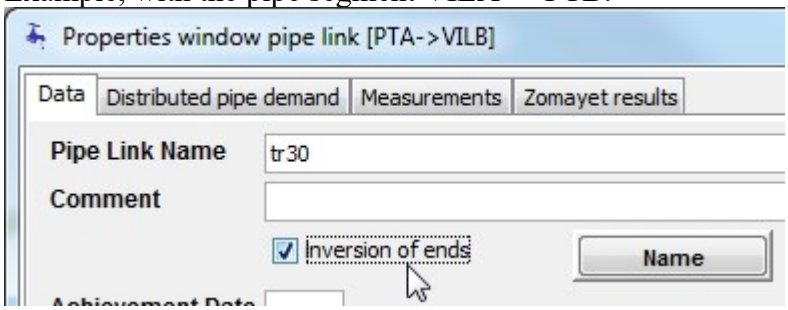
N° of roughness Porteau 2	Hazen-Williams	Colebrook (mm)
1	95	2.0
2	106	1.0
3	116	0.5
4	130	0.25
5	136	0.10
6	141	0.05
7	145	0.025
8	146.5	0

Data Tab

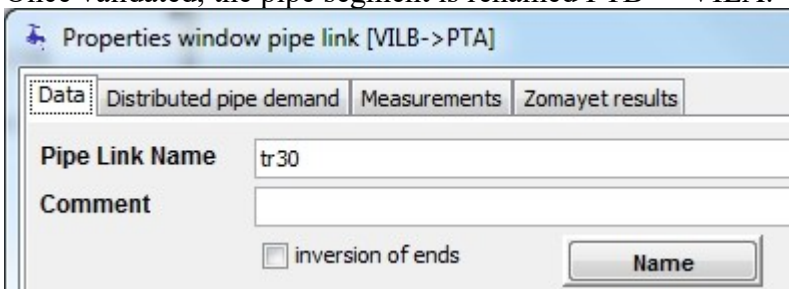


Fields : The **Pipe name** is a chain of characters enabling the pipe segment to be uniquely identified; this field cannot be the same for two segments and cannot be empty.
It can however be constructed from the names of its two nodes by clicking on the “name” button.The

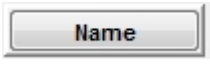
Comment field allows characters to be entered without control.The **Enabled section** checkbox allows to active or not section for simulations. **Currently** this option has no effect.It is possible to inverse the direction of an entered pipe segment.
Click on the **Inversion of ends** checkbox (you will be alerted if there are any devices on the segment).
Example, with the pipe segment VILA -> PTB:



Once validated, the pipe segment is renamed PTB -> VILA:

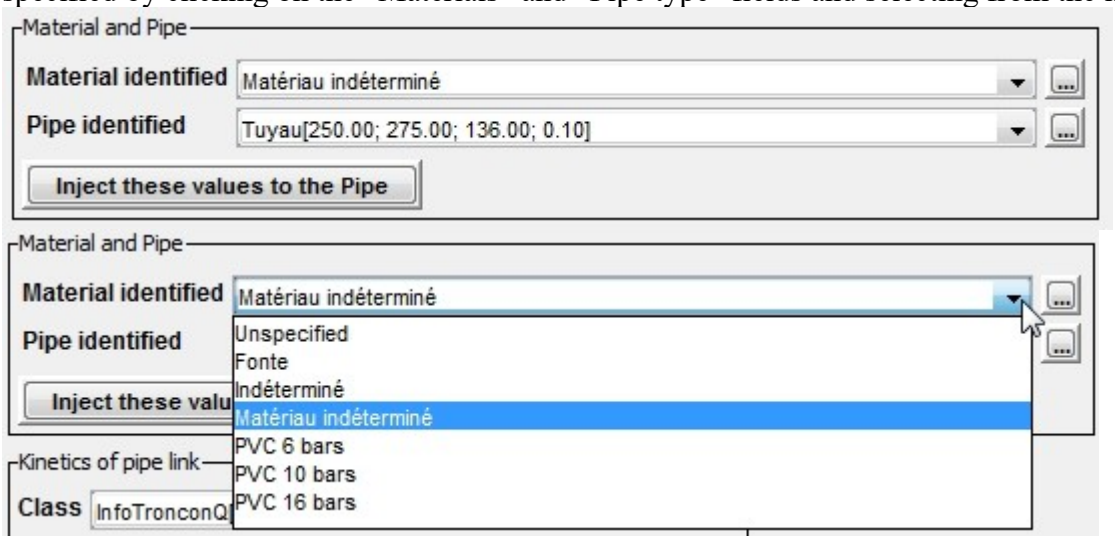


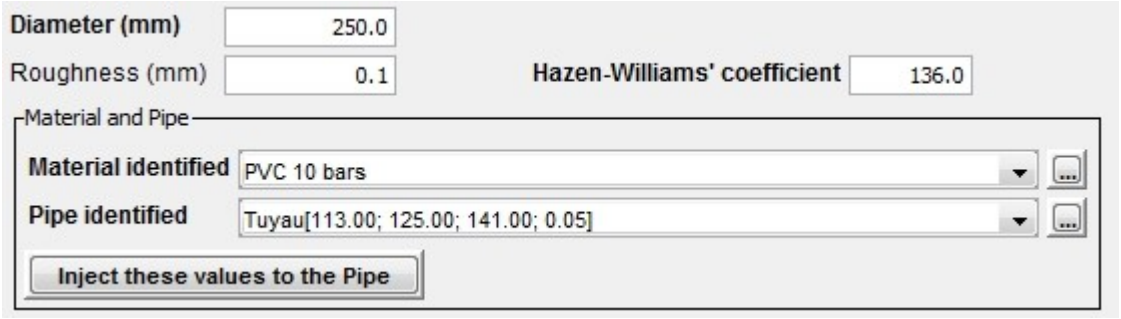
Take care its name is not changed, to rebuild the name with end nodes names, click on the



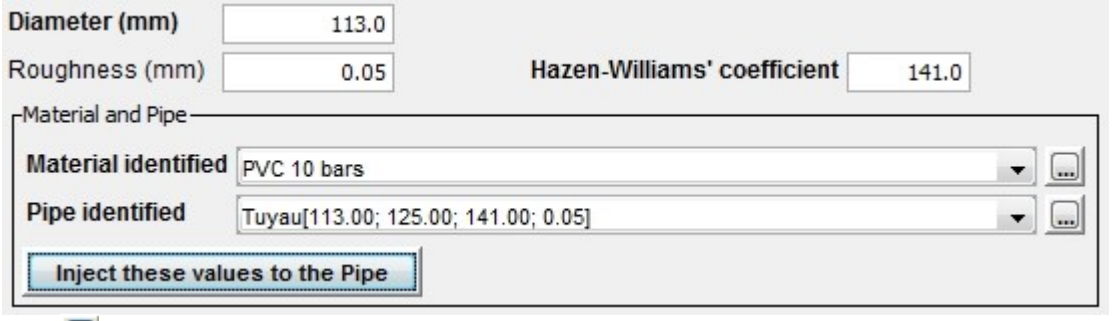
button.
The **Achievement Date** field allows any string to be entered without control.The **Diameter** field allows a numerical value to be entered; its default value is that of the default pipe type chosen for the project. The default unit is the millimetre.The **Hazen-Williams coefficient** field allows a non-dimensional numerical value to be entered.

Material and Pipe type : The “Material and Pipe type” frame allows materials and pipe types that are different from the defaults to be chosen for the pipe segment. New materials and pipe types can be specified by clicking on the “Materials” and “Pipe type” fields and selecting from the lists.

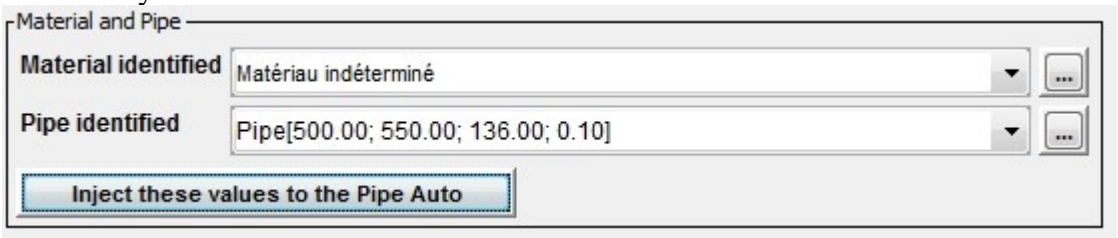




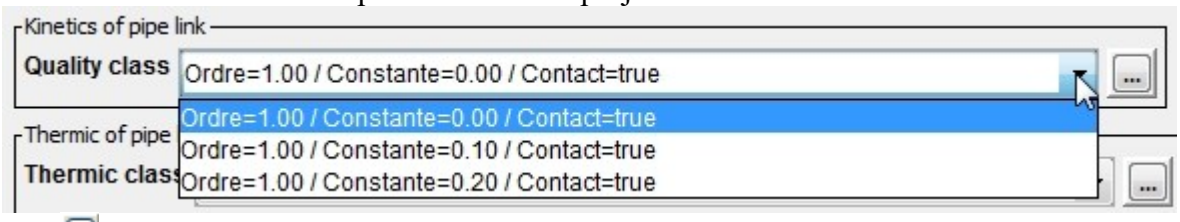
By clicking the box “Assign these values to the pipe”, the “Diameter”, “Roughness” and “Hazen-Williams coefficient” fields are updated with the associated values.



The  buttons access the **dialog boxes** for entering the possible materials and pipe types.
The combination of the click on the Inject button and the pushed-down Shift key switches the function of injection mode : automatic mode or not, the automatic mode automatically injects the changes in choice of material and/or pipe in the local data for internal diameter, roughness and Hazen-Williams coefficient if necessary.

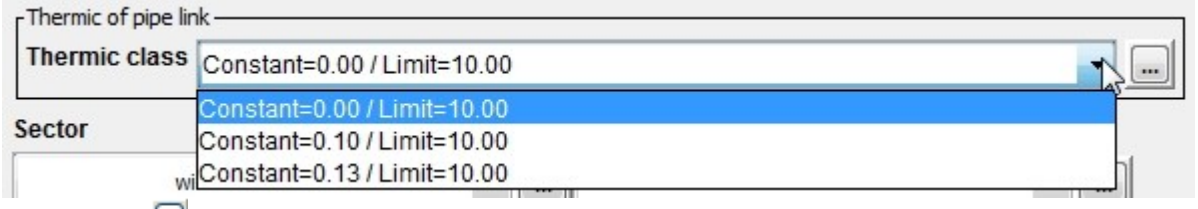


Pipe segment kinetics : The “Pipe kinetics” frame allows a kinetic order and constant to be chosen for the pipe segment. **If the order is equal to 1, the unit of the constant is [h-1].**
A triplet of "Order/Constant/Contact" values can be specifief by clicking on the “Class” field by selecting from the list. The default triplet is that of the project default.



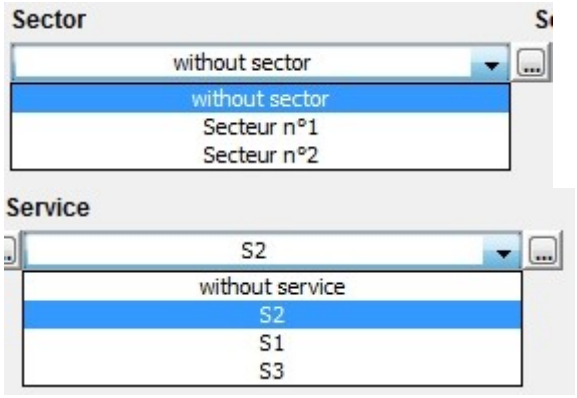
The  button access the **dialog box** for entering the possible kinetics.

Thermic section : The box "Thermic section" allows to choose a thermic exchange coefficient and an extern limit temperature for the section.
We can affect a couple "Constant/Limit" by clicking the field "Thermic" which allows to choose in the proposed list to which class the section belongs. The class by default is the one chosen for the project.



The button  allows to access the **input dialog box** of the suggested list of thermic classes.

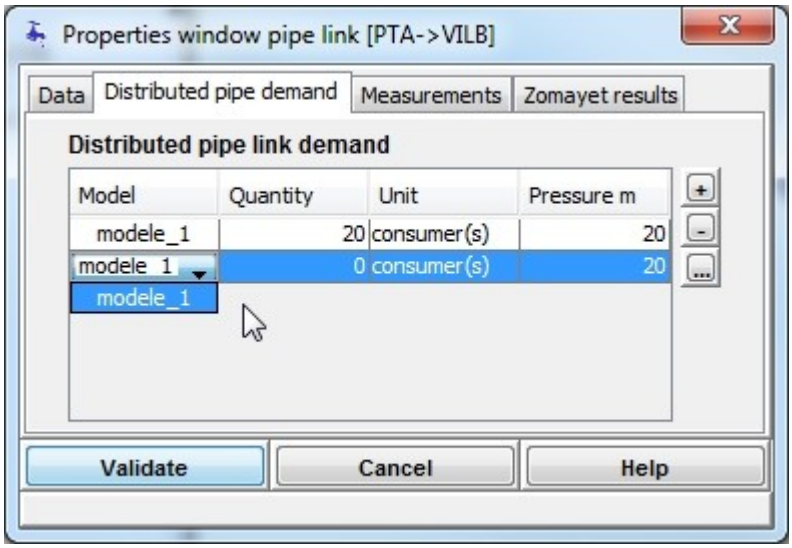
Secteur / Service :



The “Sector” and “Service” fields propose a list of the sectors and services which can be specified for each pipe segment; if none are chosen, the choice stay as “without sector” and/or “without service”.

The button access the **dialog box** for entering the possible sectors and services.

Distributed demand tab



The entering of distributed demand data functions in the same way as for the **consumption at an ordinary node**.

However, **only** the domestic model type can be specified for a pipe segment.

The “Distributed demand” table contains pairs of “consumption model / quantity consumed”. To add a pair click on the button. To delete a pair, select the line and click on the button.

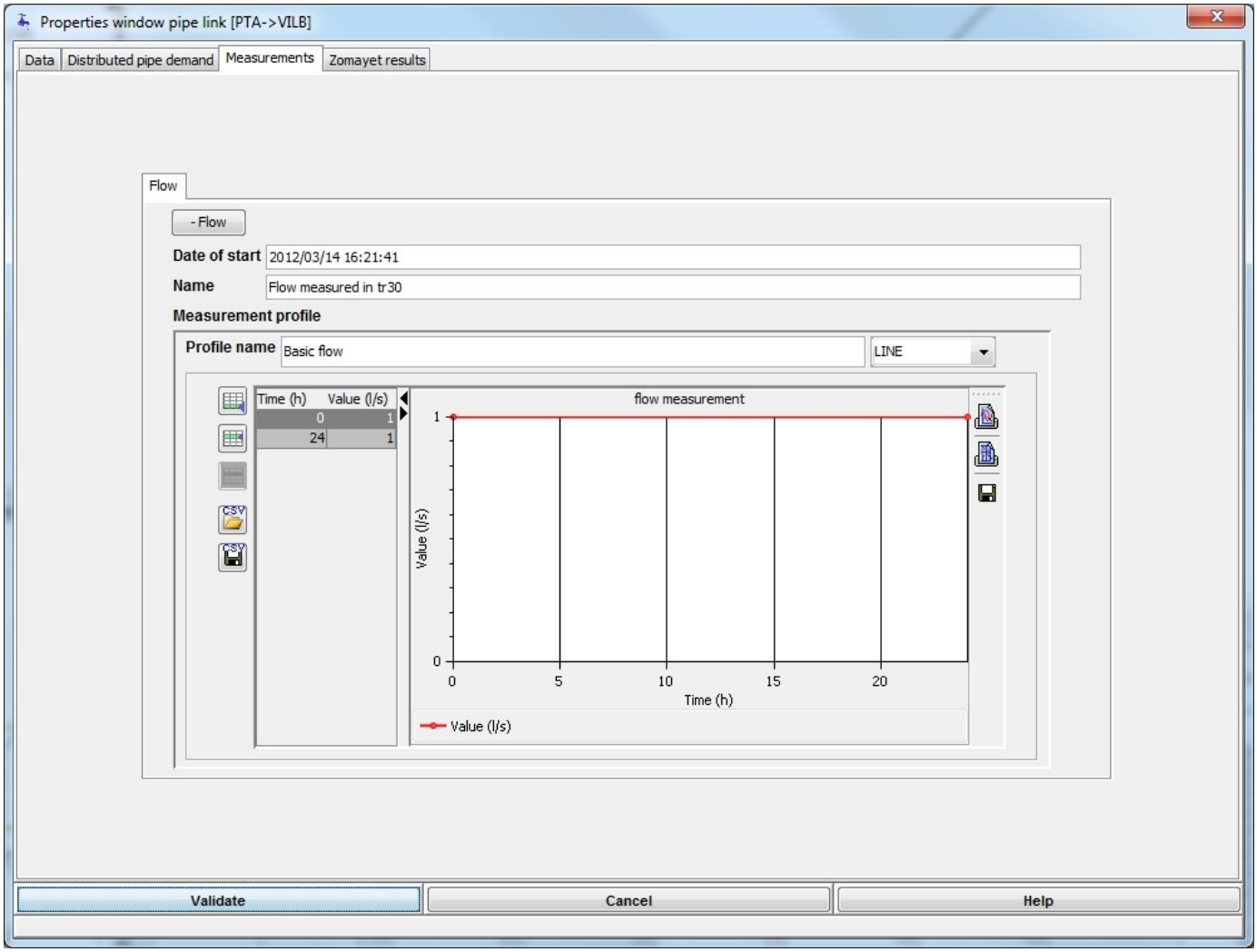
Click on the button to access the “consumption models” dialog box.

To modify the model of the pair click on its line in the “Model” column; the scroll list proposes all the models available. In the example, one type of possible model is illustrated. As a function of the model type chosen, the “Quantity” and “Unit” fields are updated. For domestic models, the quantity must be provided by the user and the unit is the consumer.

Measurements tab

It is possible to enter measurement values for a pipe segment. These will be superimposed on to the simulation results. By default, no measurement is included at the creation of a pipe segment.

By clicking the + Flow button, the dialog box for entering a flow measurement is shown.



To delete a flow measurement, click on the - Flow button.

Devices Tab (Pipe segment hydraulic device)

A pipe segment can carry one or more hydraulic devices. By default, none are present. Only one device by type can be attached to a pipe segment.
To add a device, select the button of the desired device type in the device toolbar and click on a pipe segment.

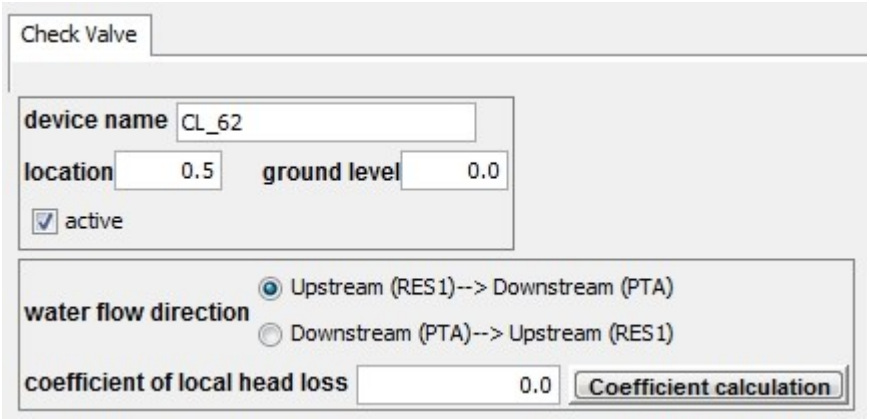
To delete an existing device, click on the button.

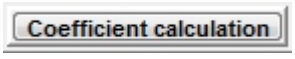
Maximum **three** devices on a pipe segment. Not twice the same type of equipment on a pipe segment.
Usage Condition:

- **Overflow inlet**: with a reserve node at end node of pipe segment, position > 50% of length
- **Float Valve**: with a reserve node at end node of pipe segment, position > 50% of length
- **Float Valve-Overflow Combination**: with a reserve node at end node of pipe segment, position > 50% of length

Pipe link hydraulic devices

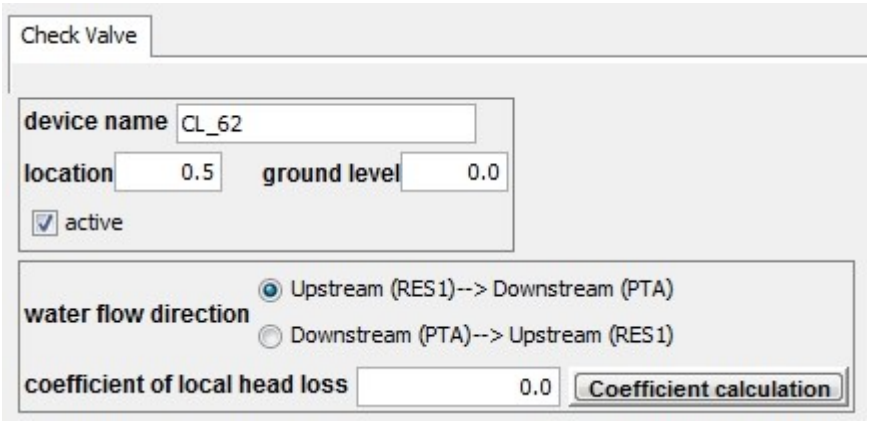
Common Datum for devices



The **device name** field is composed of XX_nnn, with XX the first letters of type of device and nnn the index number of network object representing the device.
The **location** field is the position of the device on the pipe link.
The **ground level** field allows a numerical value to be entered; its default unit is the metre.
The **active** checkbox allows the device to be rendered active or not for a calculation.
The button 

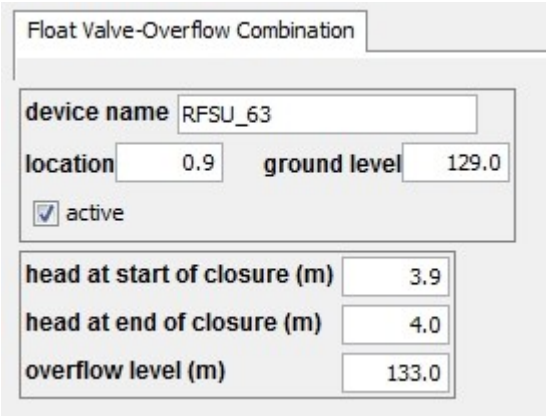
helps to calculate the head loss coefficient from a value of water level and a flow value,when this type of device is present.

Check Valve



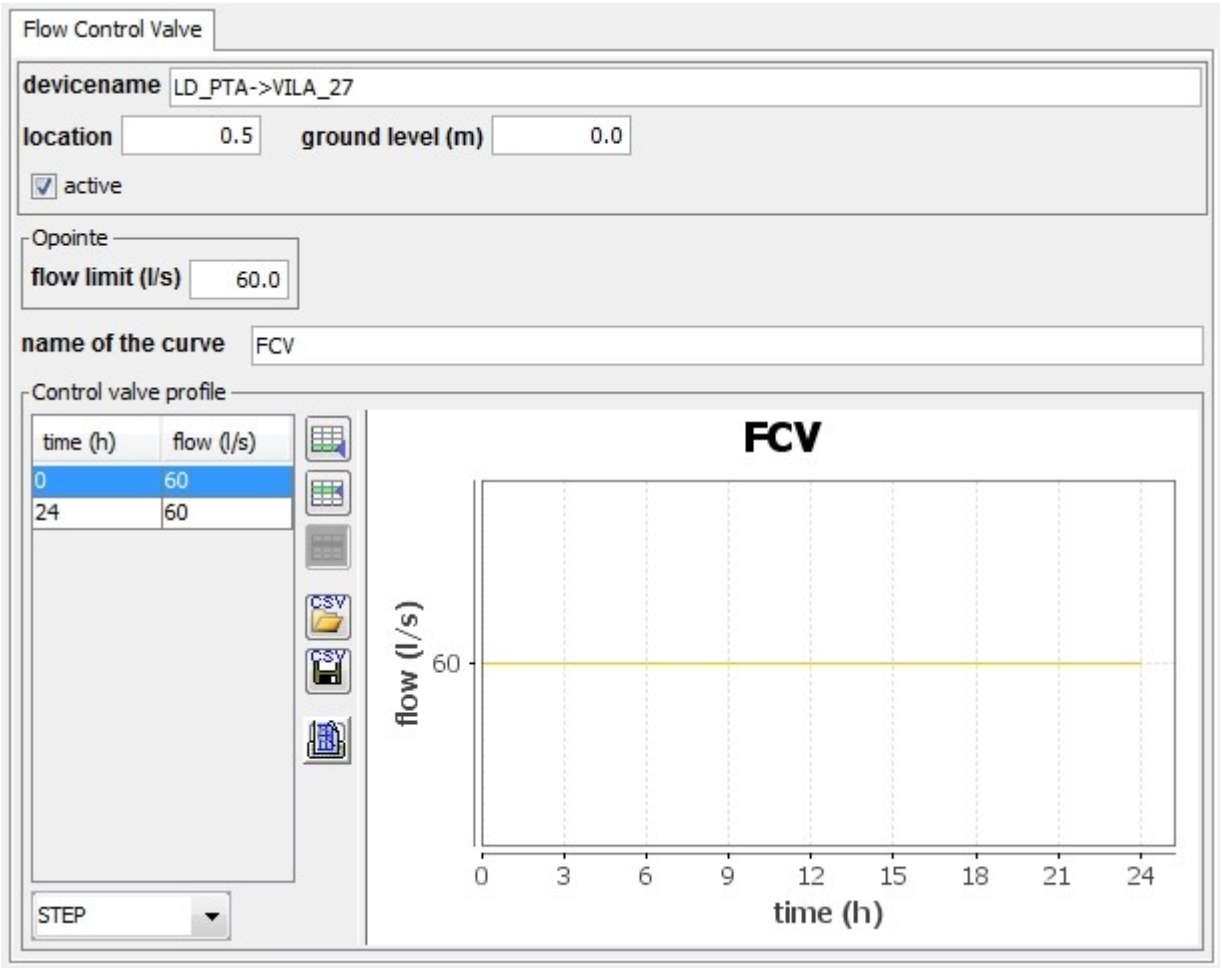
The **flow direction** field allows the upstream-downstream flow direction to be chosen using the radio buttons, "upstream-->downstream" or "downstream-->upstream".
The **local head loss coefficient** field allows a non-dimensional numerical value to be entered.

Float Valve-Overflow Combination



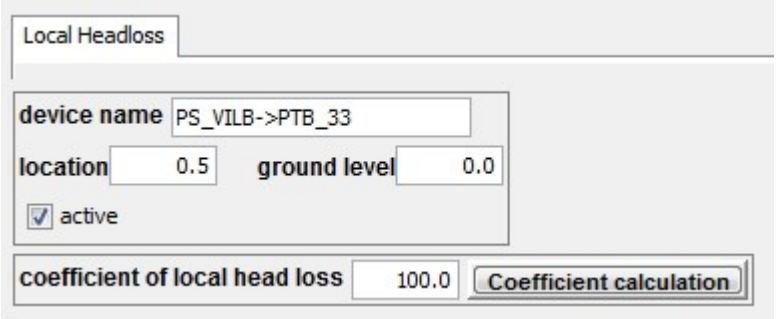
The **heigth at start of closure** field allow a numerical value to be entered; its default unit is the meter.
The **heigth at end of closure** field allows a numerical value to be entered; its default unit is the meter.
The **overflow head** field allows a numerical value to be entered; its default unit is the meter.

Flow Control Valve



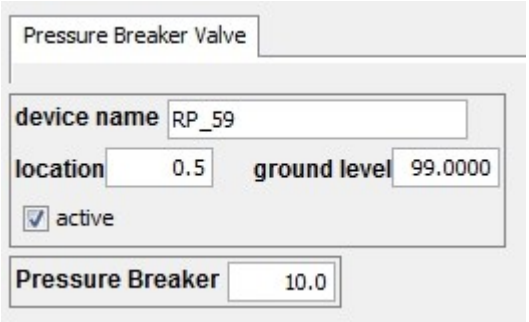
The **flow limit** field allows a numerical value to be entered for the Opointe calculation target; its default unit is the litre per second.
It is possible to enter a **temporal profile** for the flow. The way to enter this profile is identical to for **other profiles**.

Local headloss



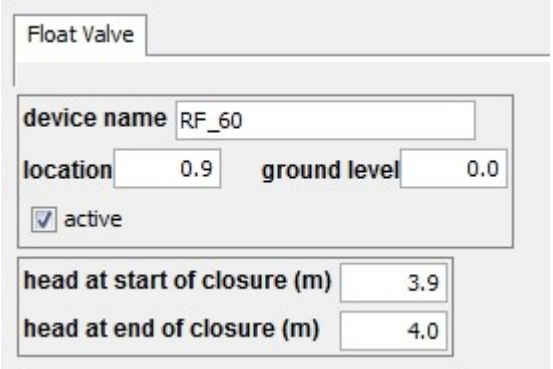
The **local head loss coefficient** field allows a non-dimensional numerical value to be entered.

Pressure Breaker Valve



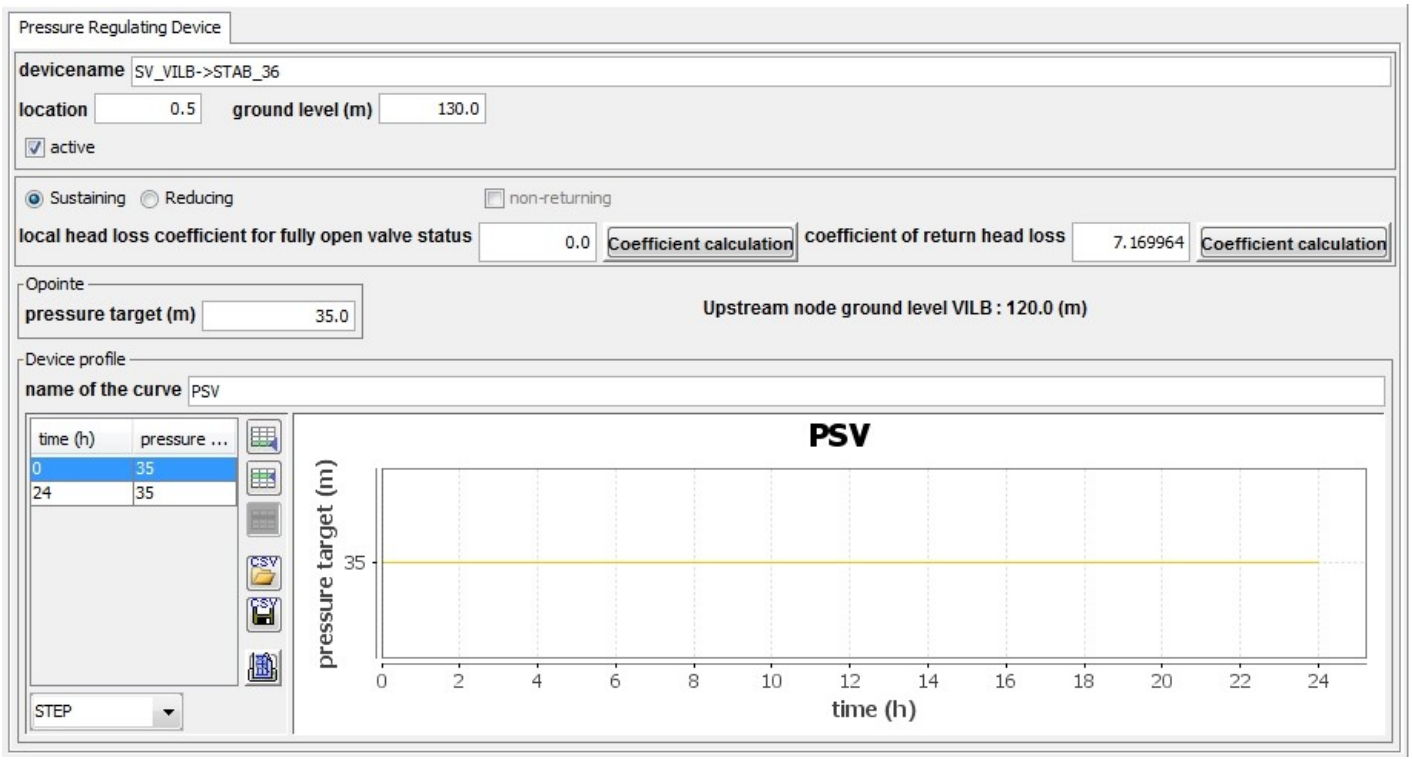
The **Pressure Breaker** field allows a reducing pressure imposed on the calculation to be entered.

Float Valve



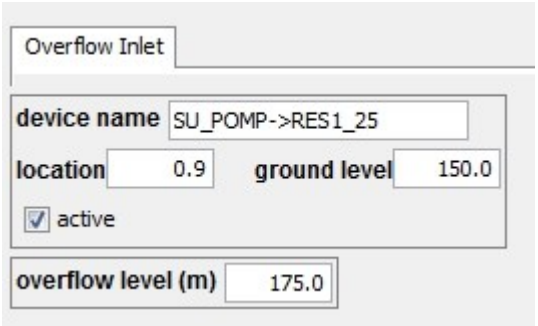
The **heigth at start of closure** field allow a numerical value to be entered; its default unit is the meter.
The **heigth at end of closure** field allows a numerical value to be entered; its default unit is the meter.

Pressure Valve (Sustaining or Reducing)



The **Sustaining** - **Reducing** radio buttons allow the choice of the type of the valve.
The **non-returning** checkbox allows the device to be function as non-return or not.
The **fully-opened head loss coefficient** field allows a non-dimensional numerical value to be entered.
The **return head loss coefficient** field allows a non-dimensional numerical value to be entered.
The **pressure target** field allows a numerical value to be entered for the Opointe calculation target; its default unit is the meter.
A temporal profile for Zomayet pressure must be entered. The way to enter this profile is identical to for **other profiles**.

Overflow Inlet



The **overflow level** field allows a numerical value to be entered; its default unit is the meter.

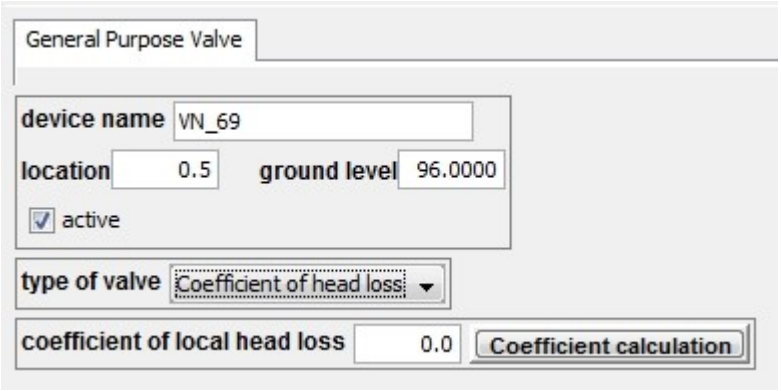
General Purpose Valve

3 types of valve exist with different data for each one:

- **Head loss coefficient type valve**
- **Closed valve**
- **Head loss profile type valve**

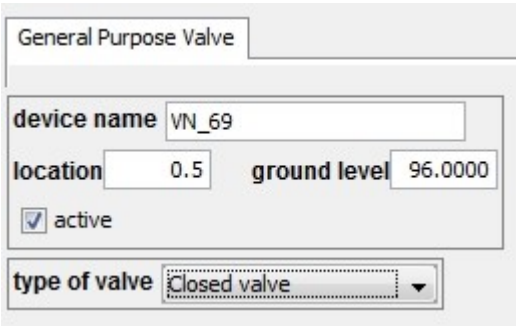
Motorised valves are a different equipment with its own type and treated separately.

Head loss coefficient type valve



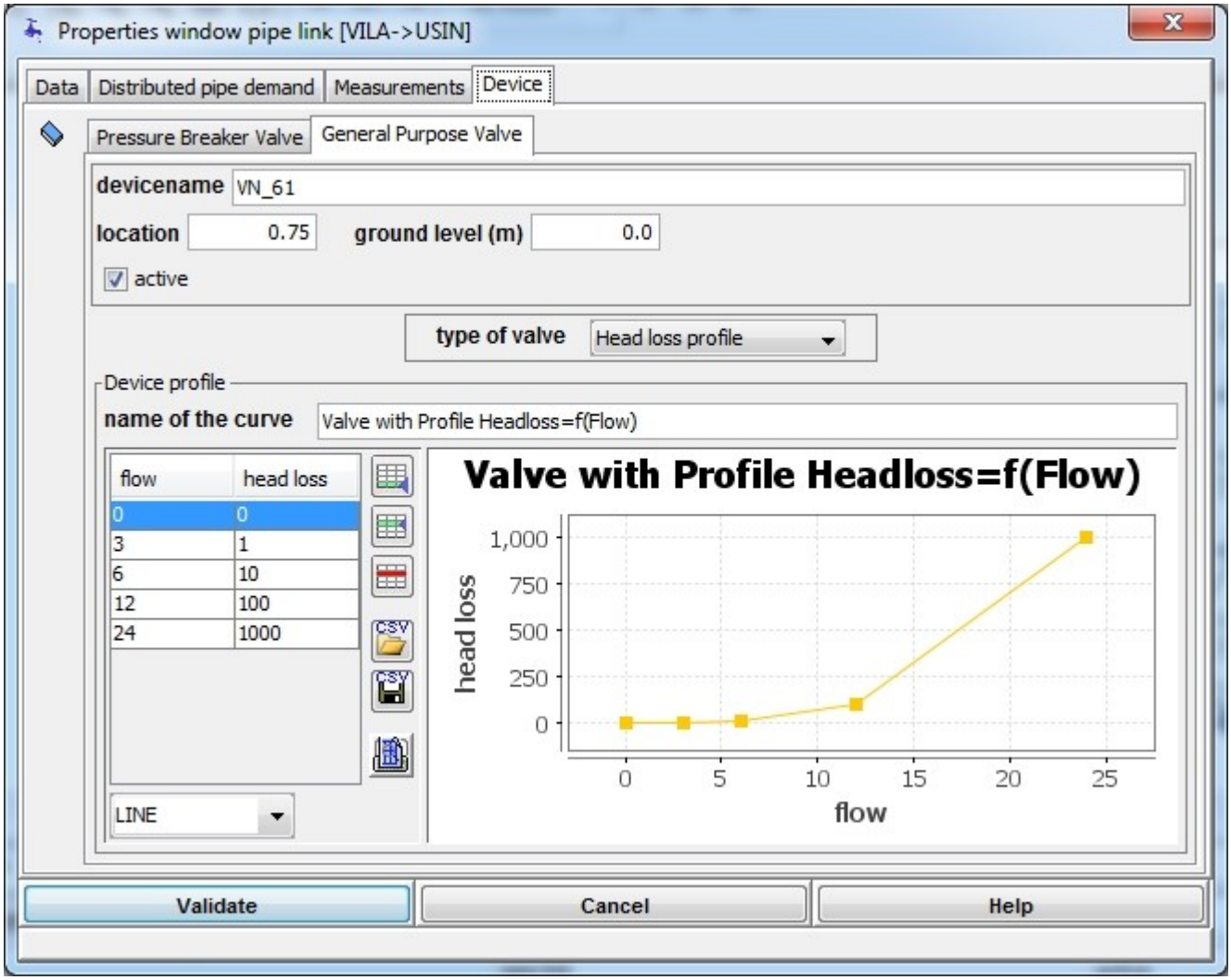
The **coefficient of local head loss** field allows a non-dimensional numerical value to be entered.

Closed Valve



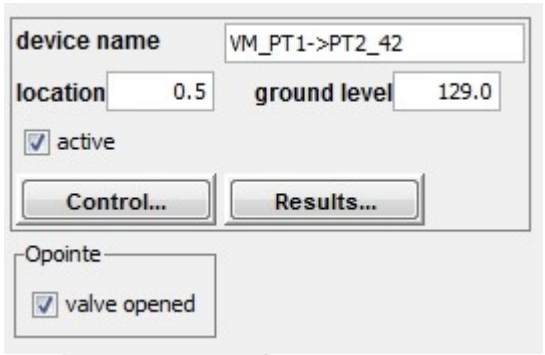
No more data for this type of valve.

Valve of head loss profile



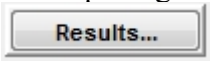
For a valve of this profile type, it is possible to enter a **head loss profile** as a function of flow.

Motorised Valve



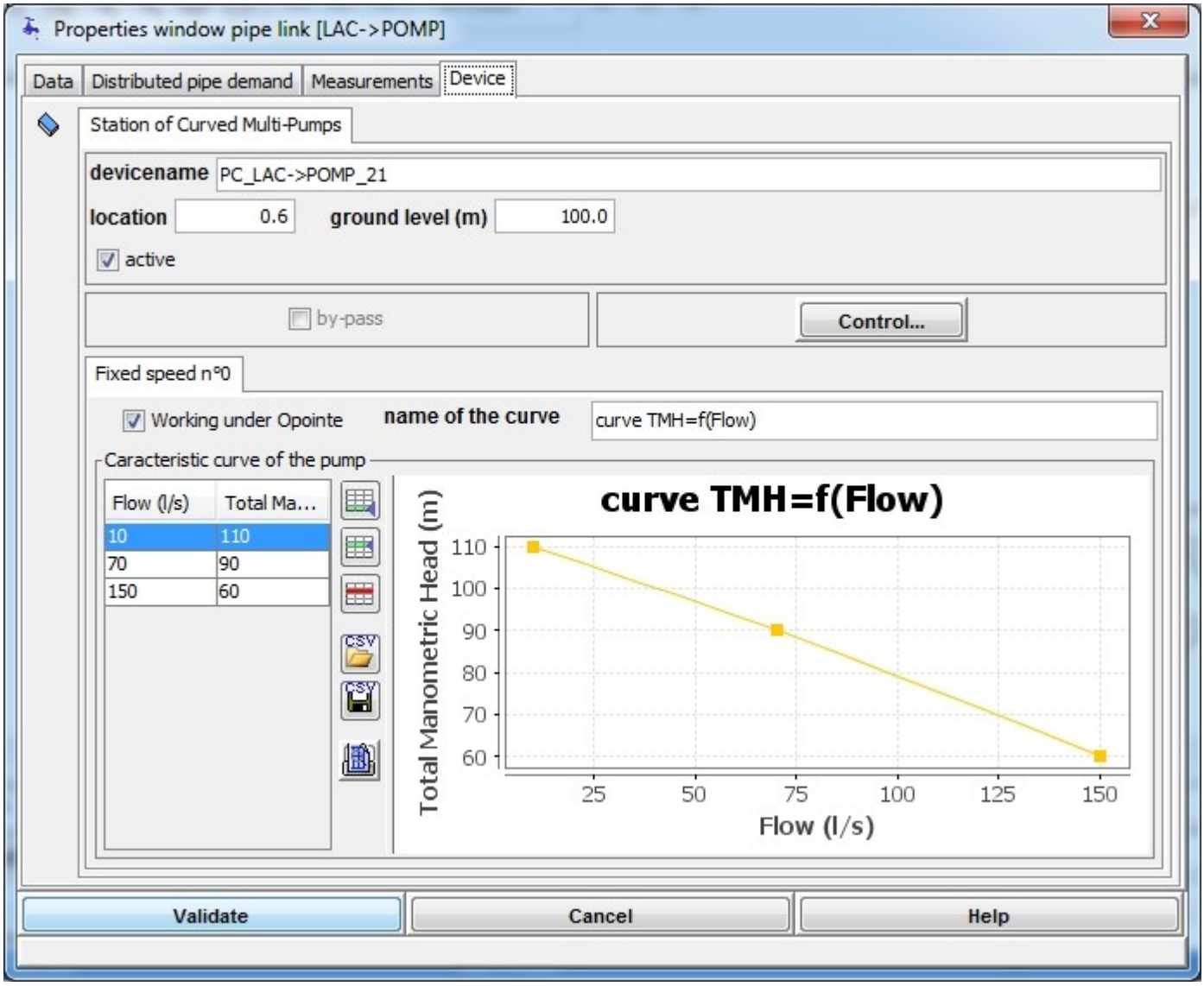
The 

button accesses the **dialog box for the regulation** of the valve.
The **valve opened** checkbox allows the opening or closing of the valve to be imposed under Opointe.

If Zomayet results are present, the 

button allows these to be accessed for the regulation of the valve under Zomayet.

Pump



The **bypass** checkbox allows the pumping station to be bypassed with its exit functioning as a nonreturn valve.
The Control... button alowws accesses the **dialog box for the regulation** of the pump.
The **Working under Opointe** checkbox imposes the starting and stopping of the pump under Opointe.
The lower part of the dialog box allows the pump characteristic curve to be entered. It works in the same way as **entering a profile**.
Warning, the characteristic curve must contain exactly three points. If Zomayet results are present, the button Results... allows these to be accessed for the regulation of the pumping station by Zomayet.

Regulation

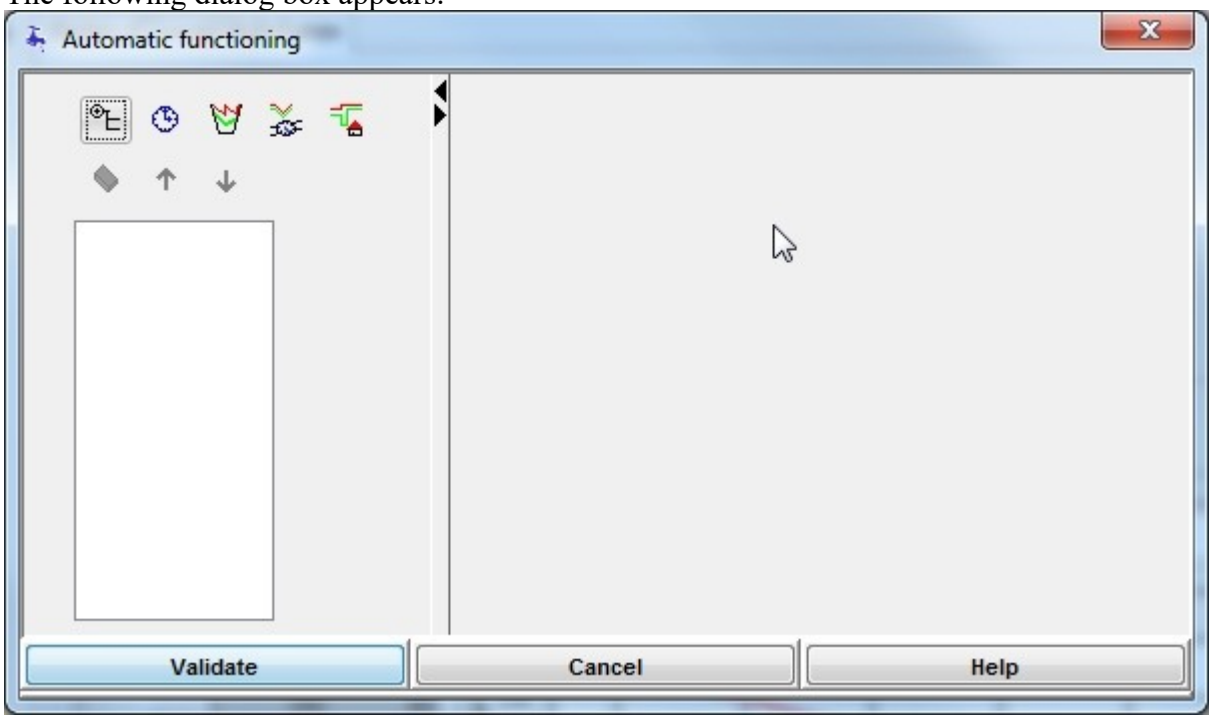
Common datum

This part is common to all types of “regulator”.

This dialog box is accessible from the dialog box for any pipe carrying a regulating device (e.g. pumping station or motorised valve). Point to the tab for the device, then click on the Control... button.

During construction, **no** regulation is created by default; the state of the device remains either **off** for a pump or **closed** for a valve if nothing is entered.

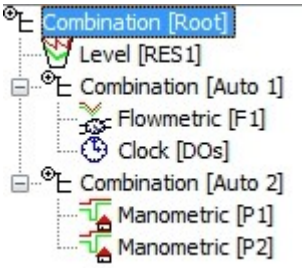
The following dialog box appears:



It consists of three zones:

- The buttons for the management of the regulation sequence.

- A tree-menu showing the assembly of regulators; here is a complex example that has already been entered.



  Editing of a regulator is made by selecting from the tree-menu.
- The selected regulator then appears in the window on the right - in this example a combined regulator.

The button bar for the management of the regulation sequence contains the following buttons:

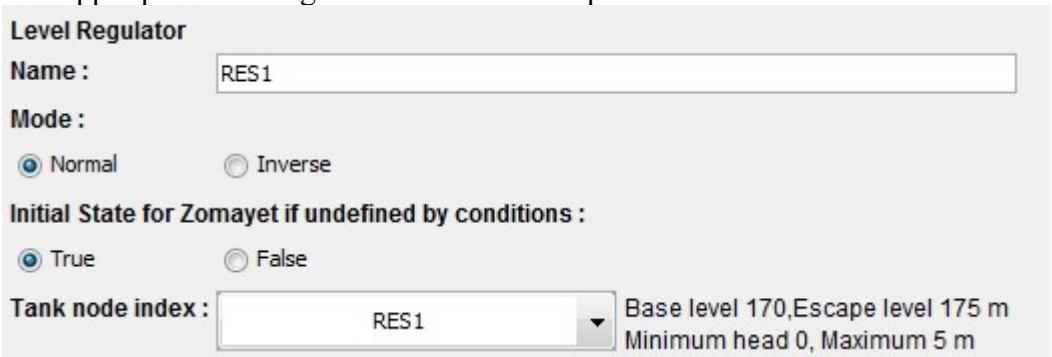
- add a combined regulator
- add a clock
- add a level
- add a flowmeter
- add a manometer
- delete the selected line
- up the selected line in the tree menu
- down the selected line in the tree menu.

In order to show the possible actions, the buttons are active or inactive depending on the line selected. The regulators simply calculate the state as a unique function of level, flow, pressure or time. Here, only level is described but the other types of regulator function according to the same principles.

Level regulator

This dialog box appears either by selecting a level regulator from the tree-menu or by adding a new one.

The upper part of the right-hand window is represented below:



The **name** field allows a particular name to be attributed to the regulator to better identify it.
The **Mode** field has a choice of two radio buttons.

For a normal regulation, the Normal button should be selected. In this case, the regulation is set as true (1), if the reference variable (tank level) goes below or reaches the start-up level at the instant of the calculation and set to false (0) if the reference variable goes above or reaches the stop level. The goal is to fill the tank.

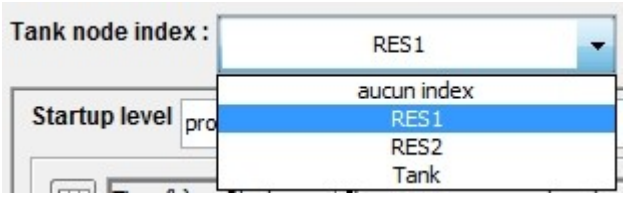
For an inverse regulation, the Inverse button should be selected. In this case, the regulation is set as true (1), if the reference variable (tank level) goes above or reaches the start-up level at the instant of the calculation and set to false (0) if the reference variable goes under or reaches the stop level. The goal is to empty the tank. Cf an example of an **inverse-regulated flowmeter**.

The **Zomayet if start-up condition is undetermined** field allows the state of the regulator to be fixed in the case where the initialisation of the calculation needs to know what happened before the simulation time begins. For example, if the reference variable at the start of Zomayet calculations is between the start-up and stopping level, the state of the regulator cannot be determined without knowing what happened before. This field allows a value to be fixed.

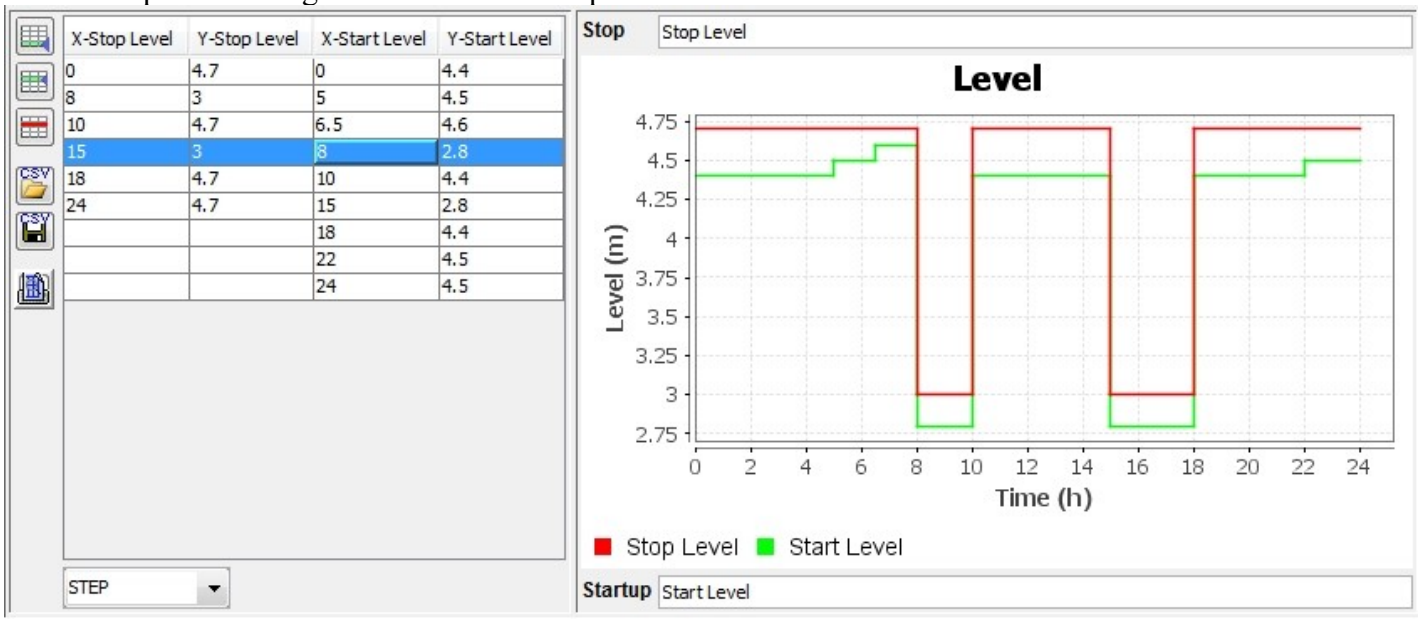
To fix this field in the true state (1), the True button should be selected.
To fix this field in the false state (0), the False button should be selected.

The **Tank node index** field designates the tank for which the flow measurement serves as the variable for calculating the state of the regulator as a function of time. It is necessary to select the one concerned from

a list of tanks.

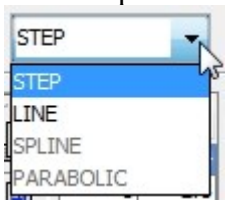


The lower part of the right-hand window is represented below:



This part of the window functions in an identical fashion to the **window for entering profiles**.

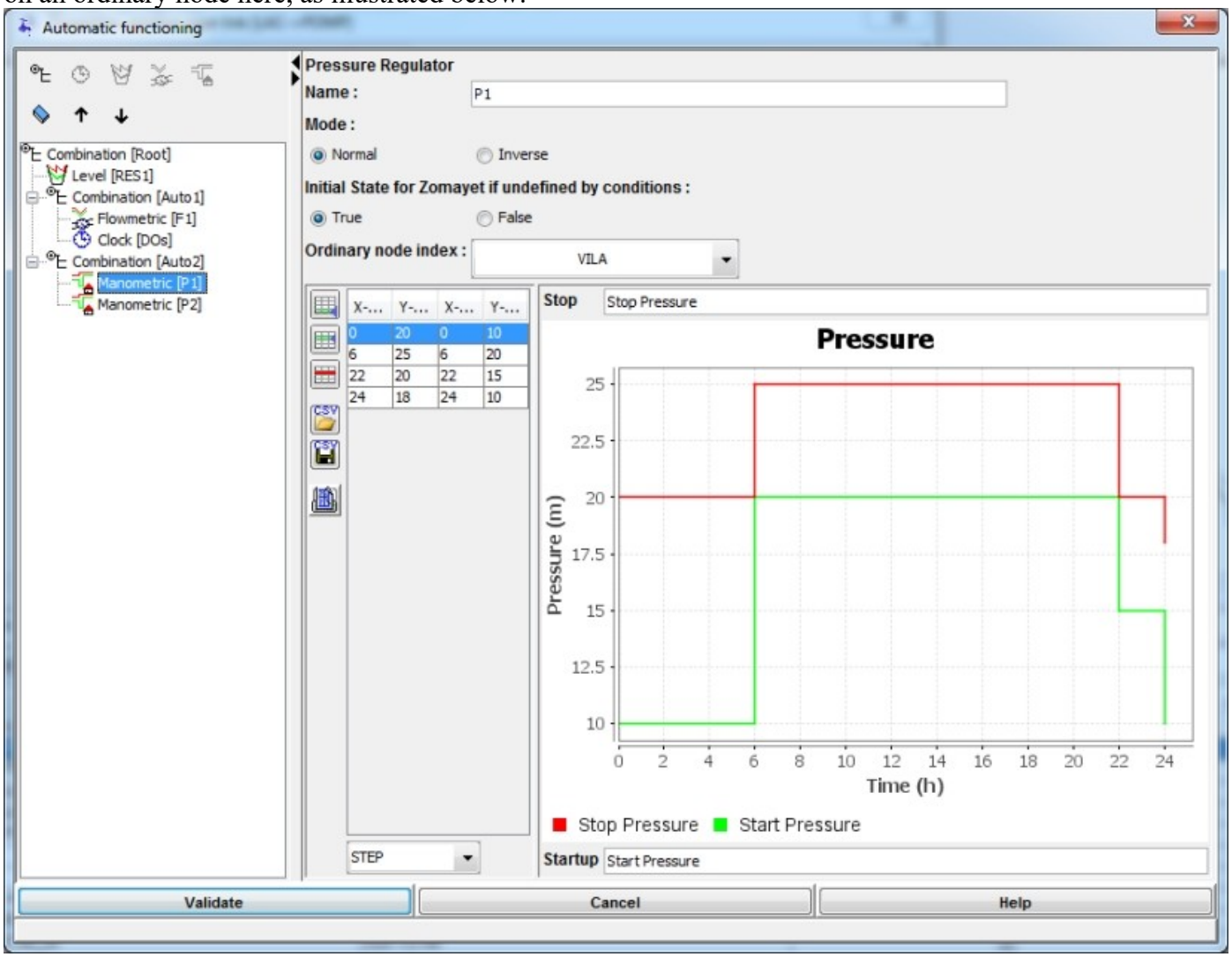
In addition, having entered the profile for the start-up level as a function of time, a second profile is shown for the stopping level.
In **Normal** mode, the table associated with the former is to the right of the graph and the table of the latter, to the left and visa versa for **Inverse** mode.
The two profiles are obligatory of the same type, either STEP or LINE.



They do not need to contain the same number of points though.
For simulations longer than 24 hours, for which the regulation must be repeated, a value of 24 hours **must be** used for both profiles, otherwise the last value will be applied until the end of the simulation time.

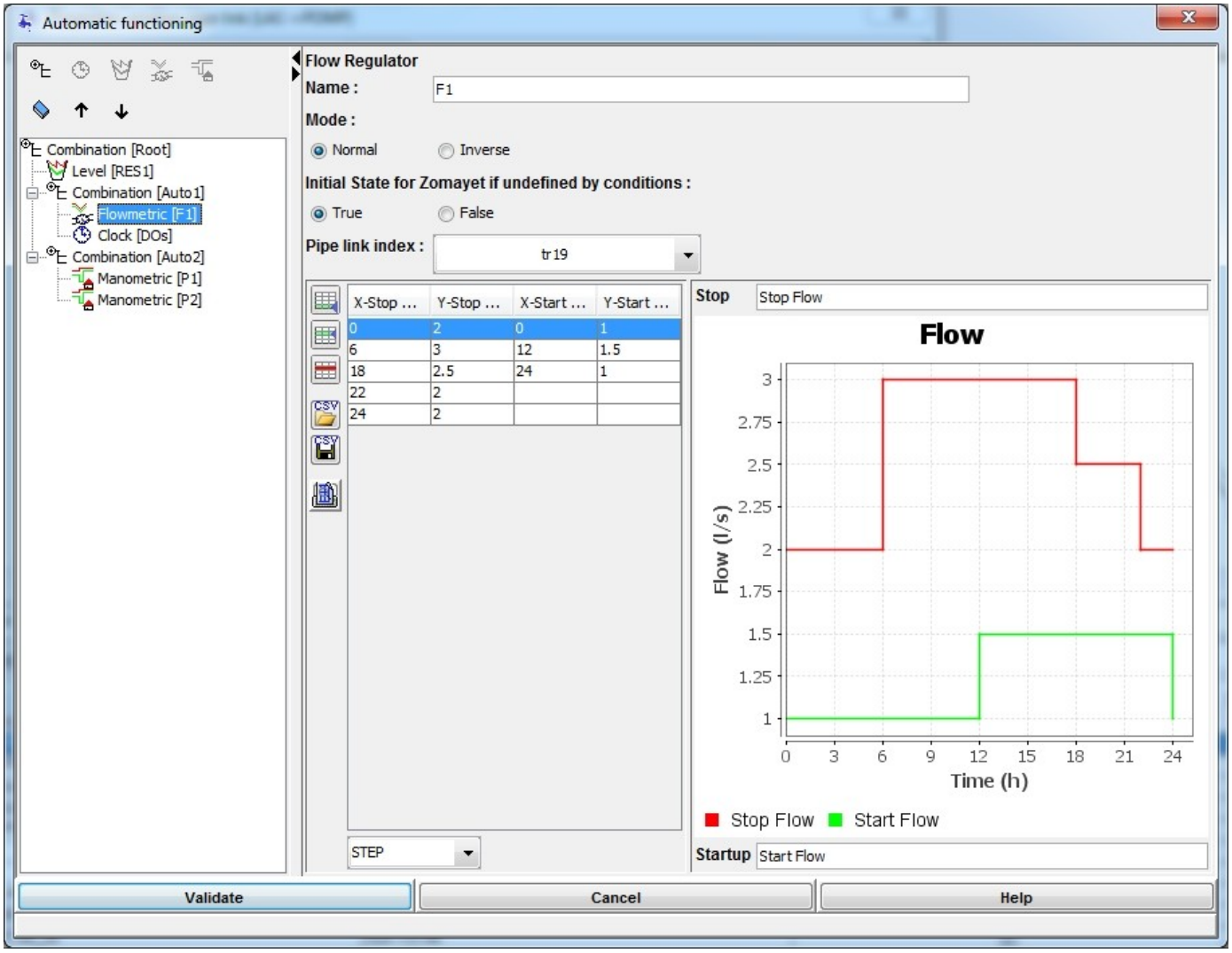
Pressure regulator

The only difference with the **level regulator** is the data value for the reference variable that is a pressure on an ordinary node here, as illustrated below.

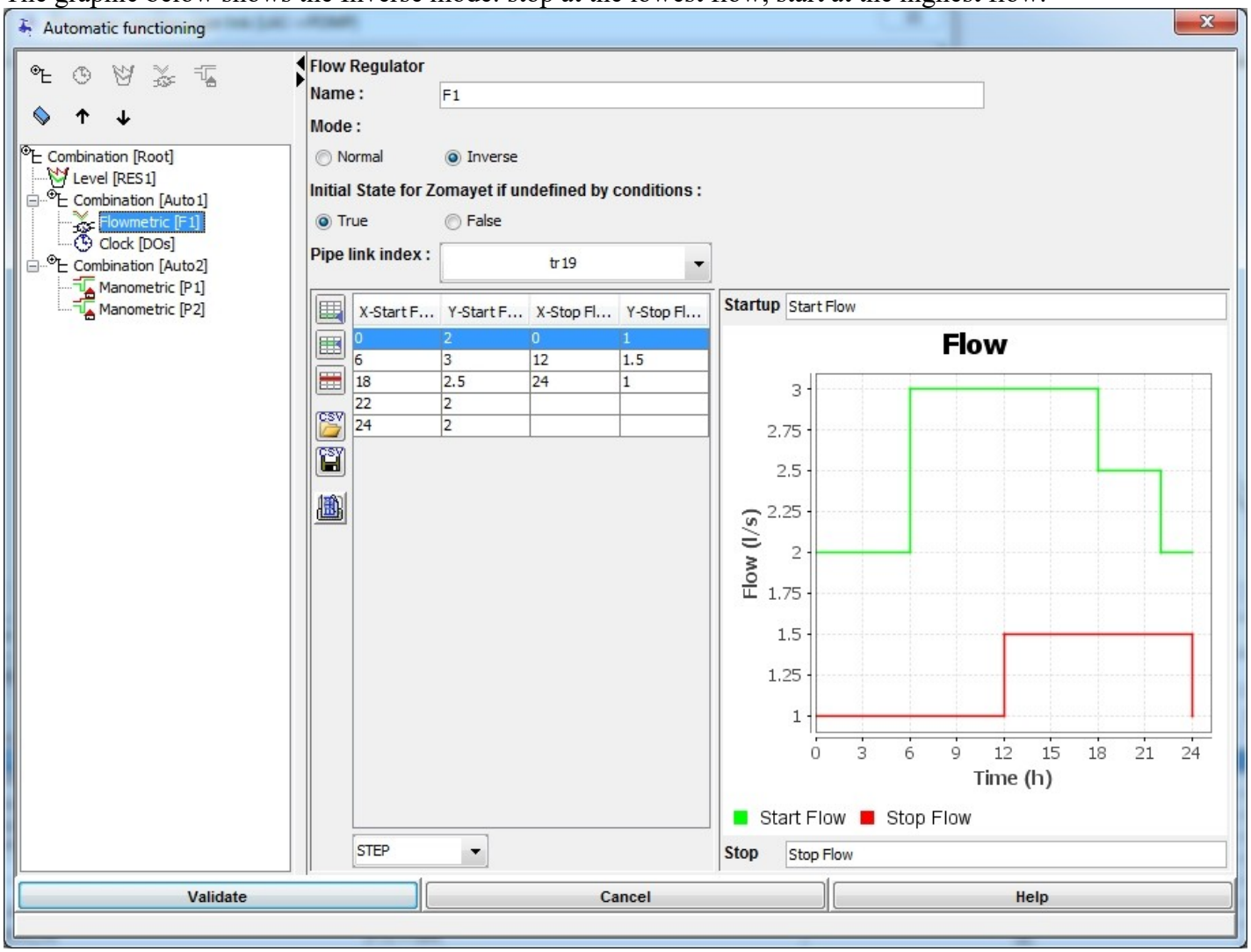


Flow regulator

The only difference with the **level regulator** is the data value for the reference variable that is a flow through a pipe segment here, as illustrated below.

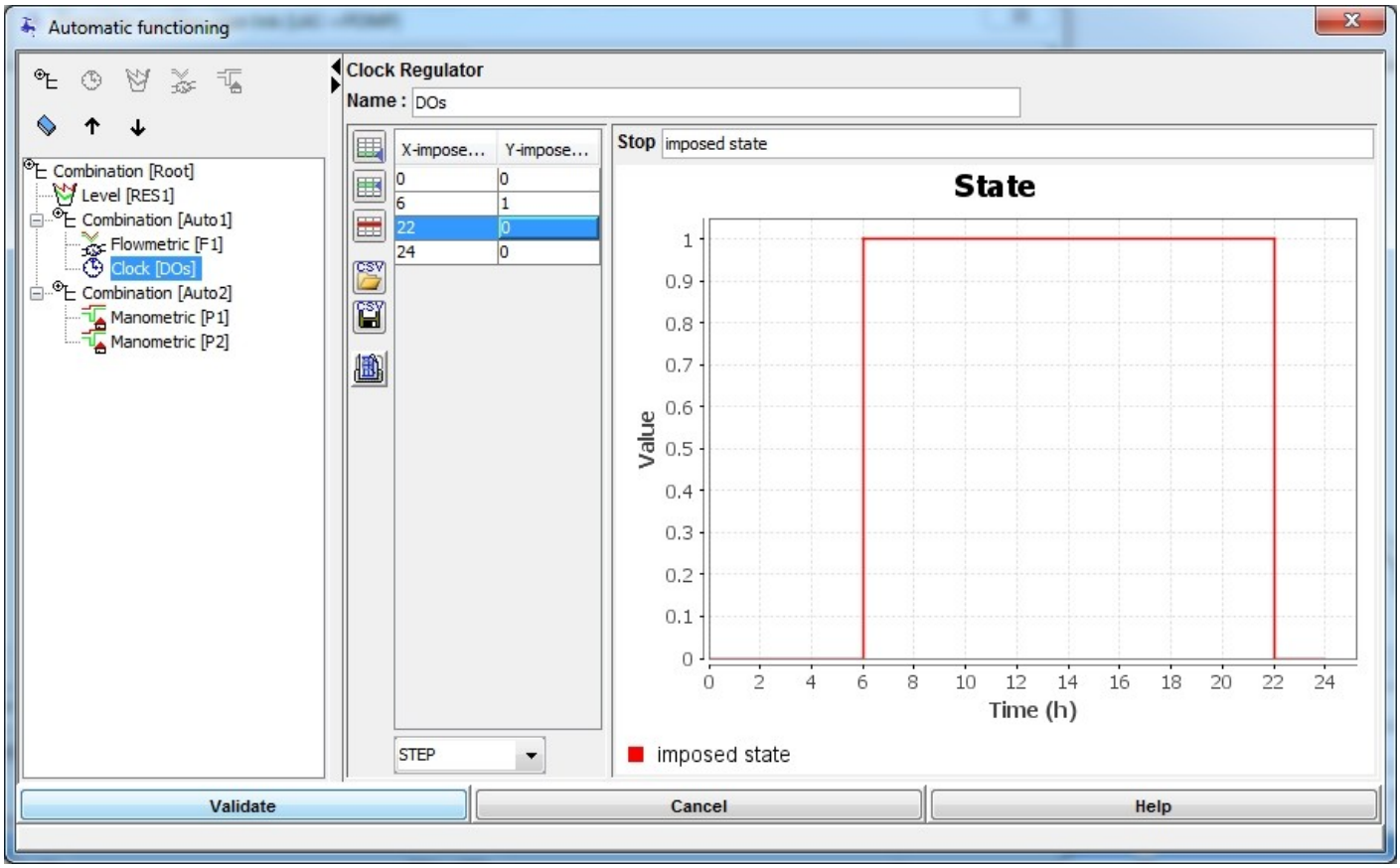

Automatic functioning
Combination [Root]
Level [RES1]
Combination [Auto1]
Flowmetric [F1]
Clock [DOs]
Combination [Auto2]
Manometric [P1]
Manometric [P2]
Flow Regulator
Name :
F1
Mode :
Normal
Inverse
Initial State for Zomayet if undefined by conditions :
True
False
Pipe link index :
tr19
X-Stop ...
Y-Stop ...
X-Start ...
Y-Start ...
0 2 0 1
6 3 12 1.5
18 2.5 24 1
22 2
24 2
STEP
Stop
Stop Flow
Flow
Flow (l/s)
Time (h)
Stop Flow
Start Flow
Startup
Start Flow
Validate
Cancel
Help

The graphic below shows the Inverse mode: stop at the lowest flow, start at the highest flow.



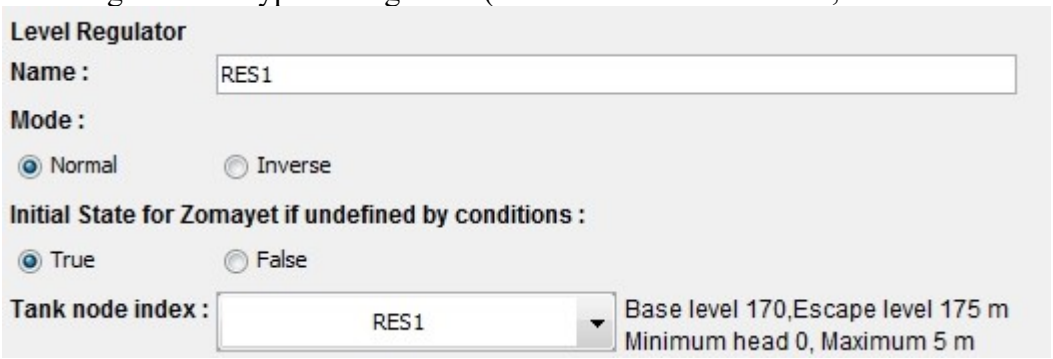
Clock regulator

This is the simplest of regulators, it is only necessary to enter the state (**0 or 1**) as a function of time as illustrated below:

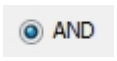


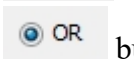
Combined regulator

For complex regulation - dependent on several tanks, for example, it is necessary to combine the state of several single regulators (e.g. two tanks require water to turn on a pump).
The “Combined” is used for this. It allows as many regulators as necessary to be associated together, including different types of regulator (combined AND combined, combined OR level, etc...).



The “name” field allows the combined regulator to be identified by a chain of characters.
Two logical operators for combination are possible:

- Select the 

button to associate the states of the regulators with the AND operator,
- Select the 

button to associate the states of the regulators with the OR operator.

Example :

AND : (level 1) AND (level 2) :

- if (state1 = true) AND (state2 = true) then true (start, open)
- if (state1 = false) AND (state2 = true) then false (stop, close)
- if (state1 = false) AND (state2 = false) then false (stop, close)
- if (state1 = true) AND (state2 = false) then false (stop, close)

OR : (level 1) OR (level 2) :

- if (state1 = true) OR (state2 = true) then true (start, open)
- if (state1 = false) OR (state2 = true) then true (start, open)
- if (state1 = false) OR (state2 = false) then false (stop, close)
- if (state1 = true) OR (state2 = false) then true (start, open)

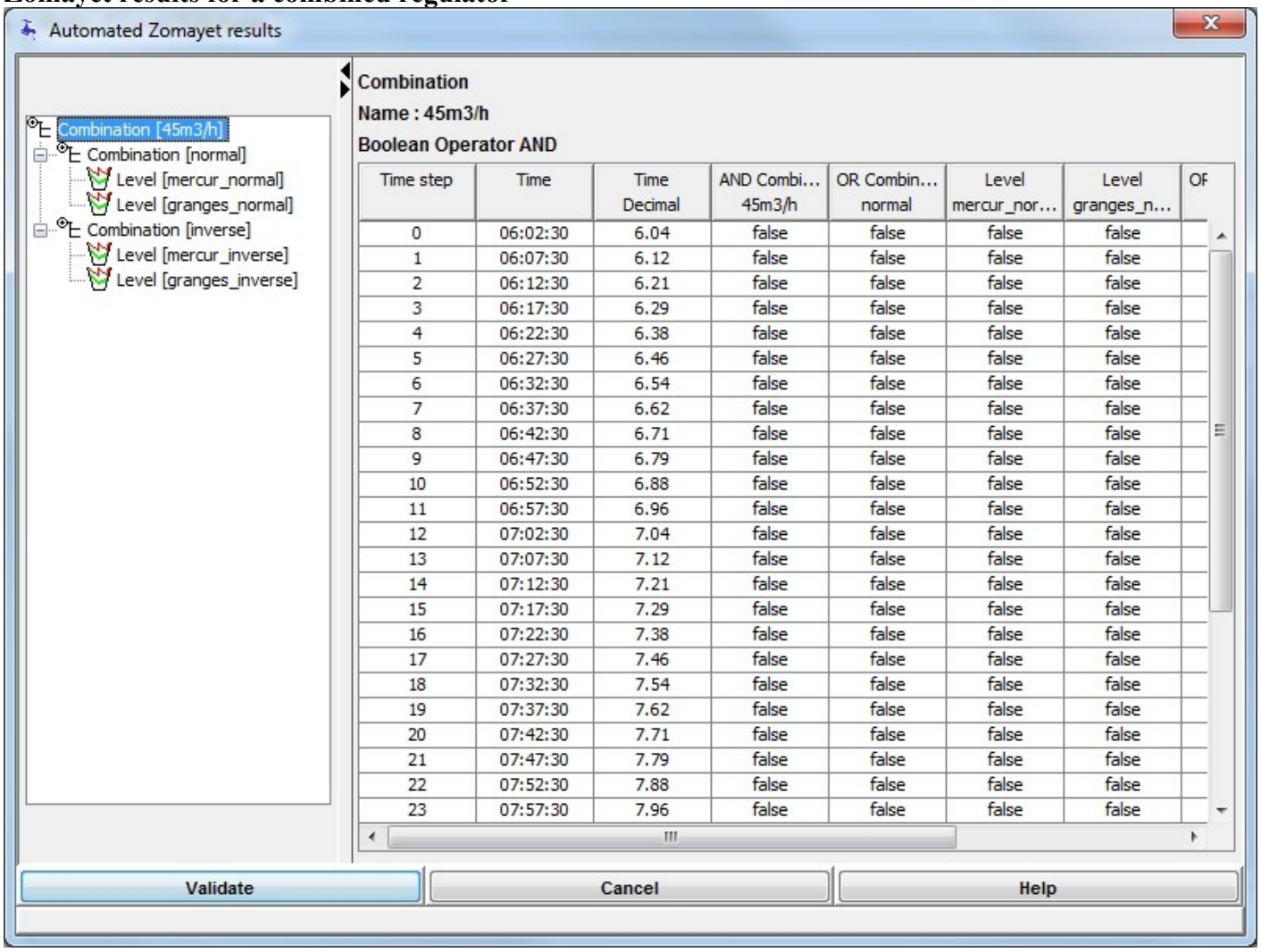
Results of regulation

Zomayet results for a level regulator

The graph above represents results of a Zomayet calculation over 24 hours. We find both curves of start

and stop (green, red). The level calculated as time goes by is represented in blue. The state of the command under binary shape (here a pump) is represented in dotted line (0 for stop, 1 for start), its Y axis is on the right. This graph can be printed or exported in vectoriel file.

Zomayet results for a combined regulator



Time step	Time	Time Decimal	AND Combi... 45m3/h	OR Combin... normal	Level mercur_nor...	Level granges_n...	OF
0	06:02:30	6.04	false	false	false	false	
1	06:07:30	6.12	false	false	false	false	
2	06:12:30	6.21	false	false	false	false	
3	06:17:30	6.29	false	false	false	false	
4	06:22:30	6.38	false	false	false	false	
5	06:27:30	6.46	false	false	false	false	
6	06:32:30	6.54	false	false	false	false	
7	06:37:30	6.62	false	false	false	false	
8	06:42:30	6.71	false	false	false	false	
9	06:47:30	6.79	false	false	false	false	
10	06:52:30	6.88	false	false	false	false	
11	06:57:30	6.96	false	false	false	false	
12	07:02:30	7.04	false	false	false	false	
13	07:07:30	7.12	false	false	false	false	
14	07:12:30	7.21	false	false	false	false	
15	07:17:30	7.29	false	false	false	false	
16	07:22:30	7.38	false	false	false	false	
17	07:27:30	7.46	false	false	false	false	
18	07:32:30	7.54	false	false	false	false	
19	07:37:30	7.62	false	false	false	false	
20	07:42:30	7.71	false	false	false	false	
21	07:47:30	7.79	false	false	false	false	
22	07:52:30	7.88	false	false	false	false	
23	07:57:30	7.96	false	false	false	false	

The table for the assembly of regulators allows time step by time step to find the calculation of the complete regulation. The vertical scroll allows exploration of the time until the end of the simulation. This table can be copied (after selecting using <Ctrl>+A) as tabulated text.

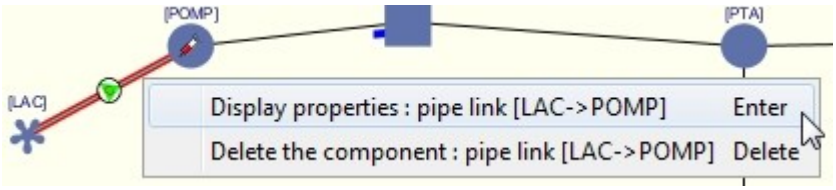
Access to the pipe simulation results

Introduction

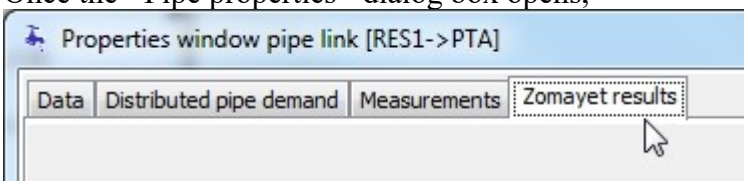
The access to the pipe simulation results for individual pipe segments firstly requires the pipe itself to be selected and of course on there being calculations results available to consult. Note, even after the modification of the data used for the OPOINTE or ZOMAYET calculations, these results remain present until another calculation replaces them.

To recap, there are three ways to access in Map View or Schematic View:

- Double left click on the selected pipe here in red, LAC-RES1
- Double right click on the selected pipe then select the first line from the contextual menu by either right click or pressing <ENTER> on the keyboard.
- Double click on the table of pipes on the line of the selected pipe.



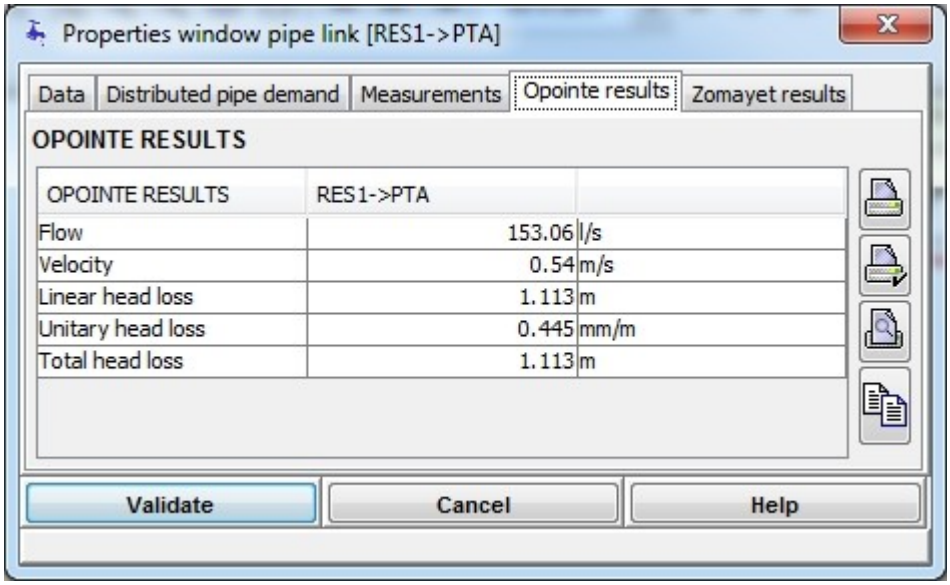
Once the <Pipe properties> dialog box opens,



Click on the "<Opointe results> or <Zomayet results> tabs according to your need.

Opointe

Only three result figures are shown: the flow in l/s, the velocity of the water in the pipe in m/s and the linear head loss in m.



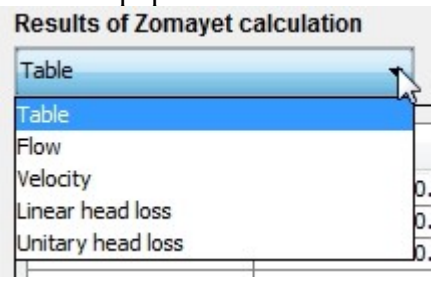
Zomayet

There is access to the results of a Zomayet simulation of one of the pipe segments in the network as a function of time.

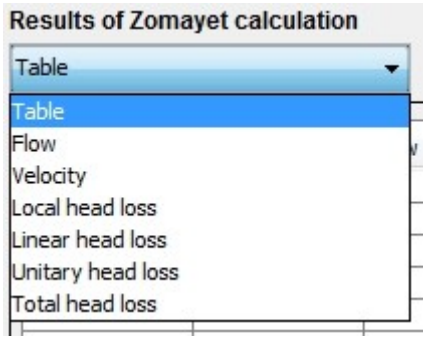
The tab shows a table giving different results according to whether the pipe supports a distributed demand or not and/or if it is equipped with a hydraulic device or not.

The scrolling list allows the choice of information to be shown, with 4 variations according to whether the pipe has (a) no particularity; (b) has a hydraulic equipment; (c) has a distributed demand or (d) has both hydraulic equipment and a distributed demand.

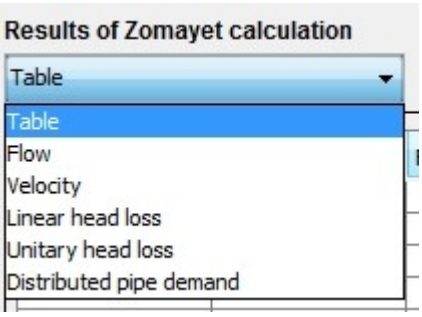
a)



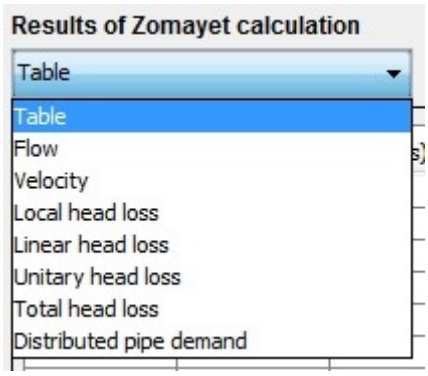
b)



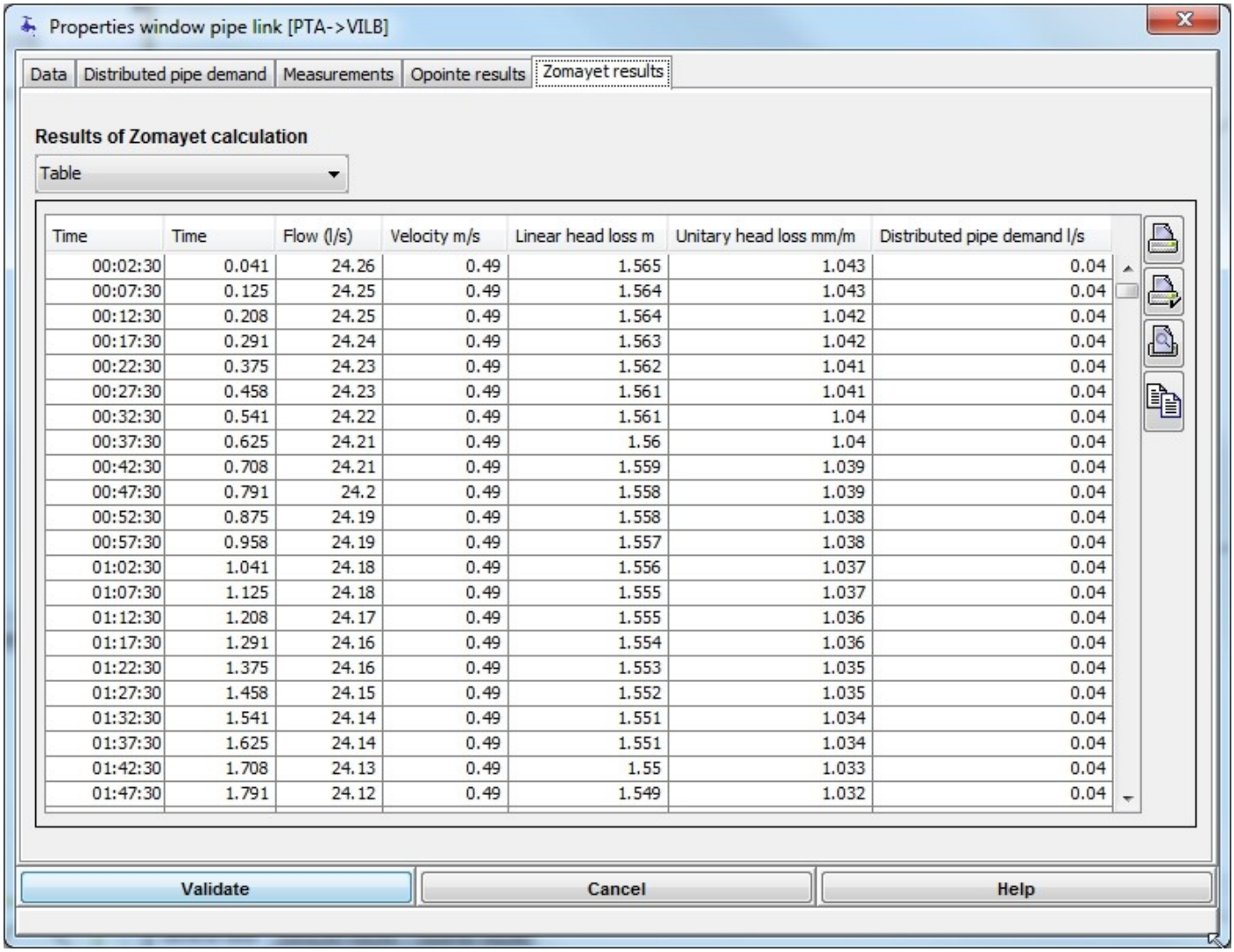
c)



d)



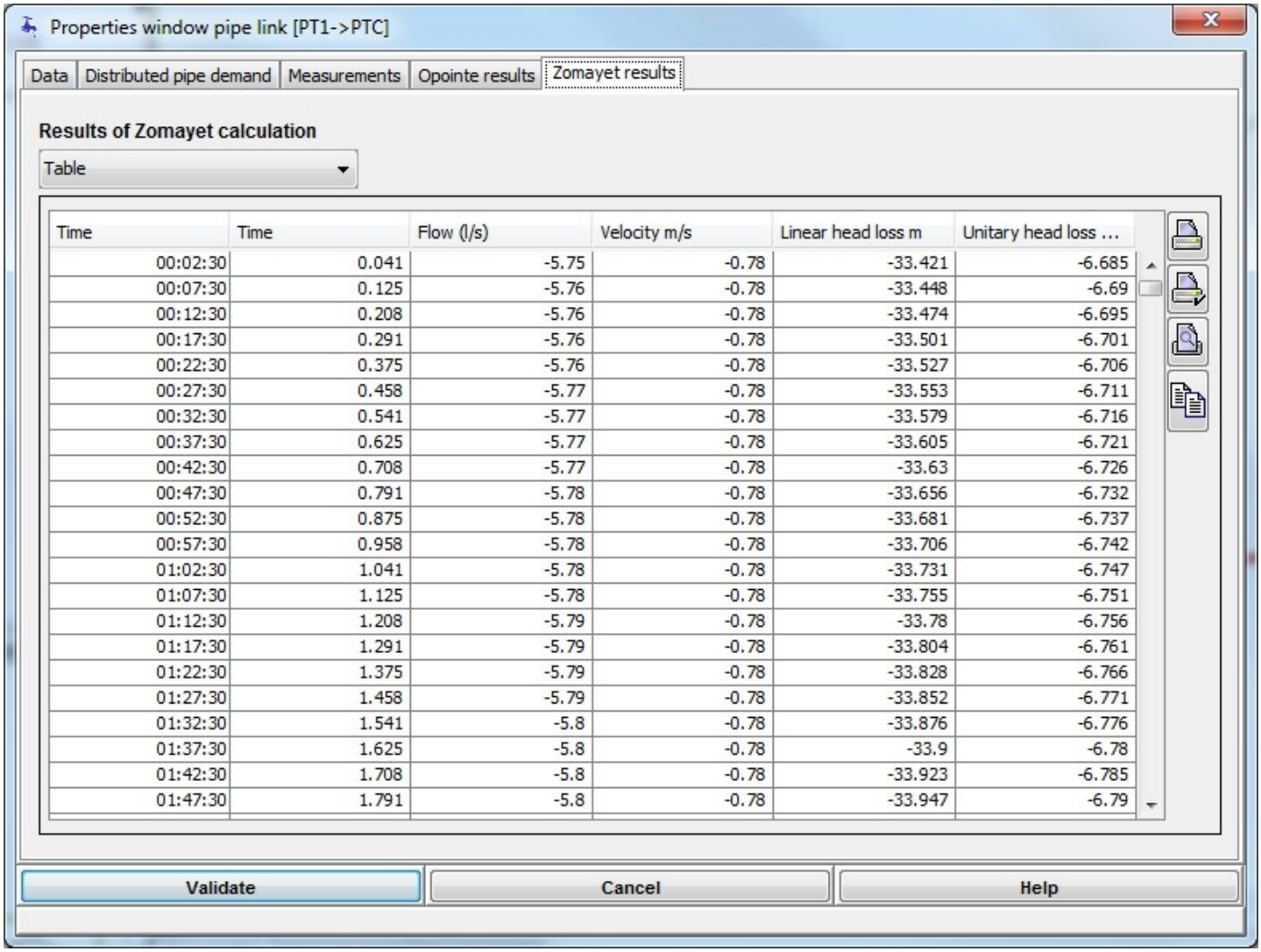
The full table of values is shown as follows (here for case d):



Time	Time	Flow (l/s)	Velocity m/s	Linear head loss m	Unitary head loss mm/m	Distributed pipe demand l/s
00:02:30	0.041	24.26	0.49	1.565	1.043	0.04
00:07:30	0.125	24.25	0.49	1.564	1.043	0.04
00:12:30	0.208	24.25	0.49	1.564	1.042	0.04
00:17:30	0.291	24.24	0.49	1.563	1.042	0.04
00:22:30	0.375	24.23	0.49	1.562	1.041	0.04
00:27:30	0.458	24.23	0.49	1.561	1.041	0.04
00:32:30	0.541	24.22	0.49	1.561	1.04	0.04
00:37:30	0.625	24.21	0.49	1.56	1.04	0.04
00:42:30	0.708	24.21	0.49	1.559	1.039	0.04
00:47:30	0.791	24.2	0.49	1.558	1.039	0.04
00:52:30	0.875	24.19	0.49	1.558	1.038	0.04
00:57:30	0.958	24.19	0.49	1.557	1.038	0.04
01:02:30	1.041	24.18	0.49	1.556	1.037	0.04
01:07:30	1.125	24.18	0.49	1.555	1.037	0.04
01:12:30	1.208	24.17	0.49	1.555	1.036	0.04
01:17:30	1.291	24.16	0.49	1.554	1.036	0.04
01:22:30	1.375	24.16	0.49	1.553	1.035	0.04
01:27:30	1.458	24.15	0.49	1.552	1.035	0.04
01:32:30	1.541	24.14	0.49	1.551	1.034	0.04
01:37:30	1.625	24.14	0.49	1.551	1.034	0.04
01:42:30	1.708	24.13	0.49	1.55	1.033	0.04
01:47:30	1.791	24.12	0.49	1.549	1.032	0.04

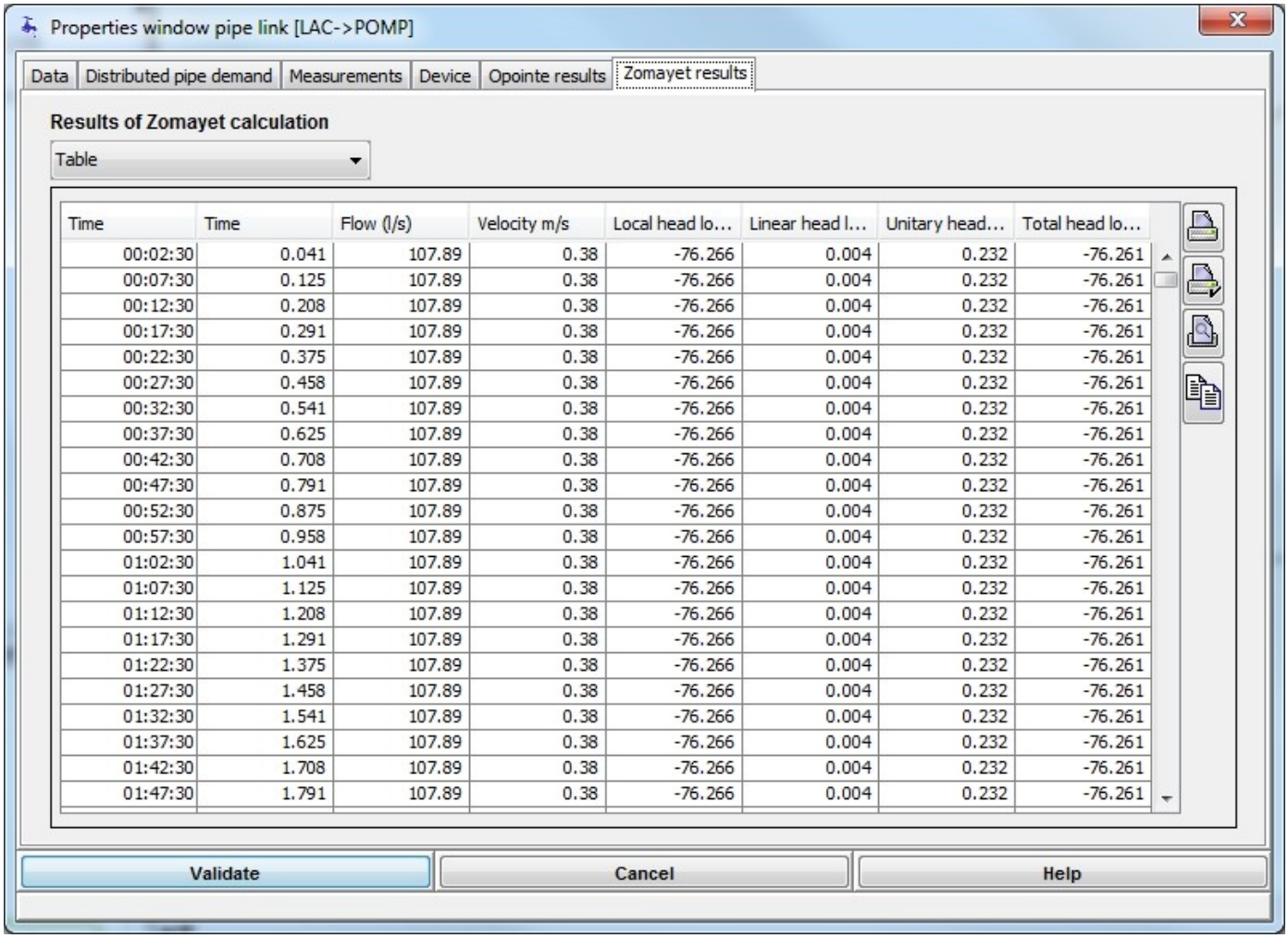
There are two time displays, a) in h:m:s ; and b) in fraction of hours, with following calculation results: the flow in l/s , the velocity in m/s, the linear head loss in m, the singular head loss in m, the total head loss in m and the distributed demand in l/s if applicable. It is possible to copy part or all (<Ctrl>+A) of the data to tabular applications using the shortcut <Ctrl>+C.

There are only the first five columns

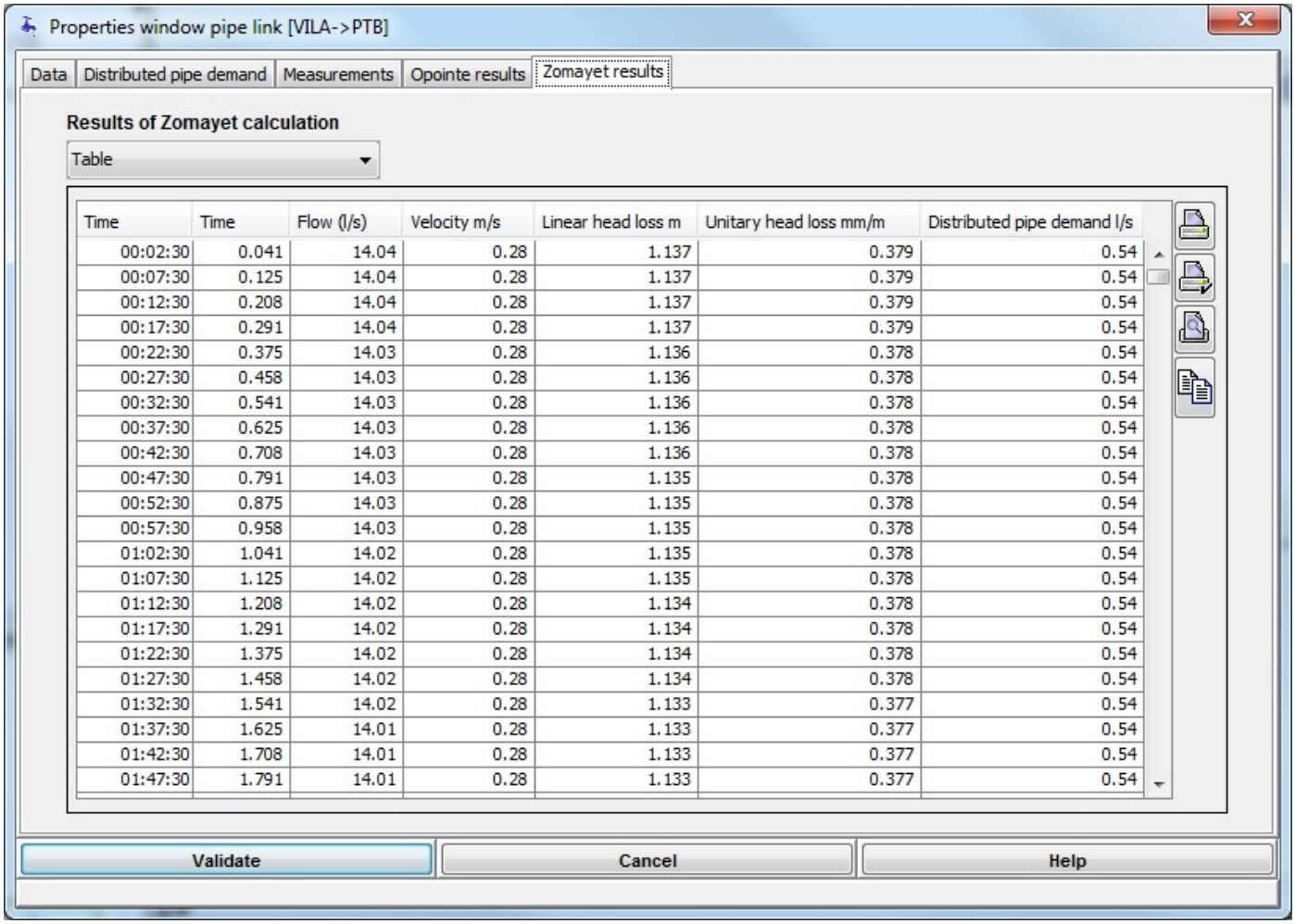


Time	Time	Flow (l/s)	Velocity m/s	Linear head loss m	Unitary head loss ...
00:02:30	0.041	-5.75	-0.78	-33.421	-6.685
00:07:30	0.125	-5.76	-0.78	-33.448	-6.69
00:12:30	0.208	-5.76	-0.78	-33.474	-6.695
00:17:30	0.291	-5.76	-0.78	-33.501	-6.701
00:22:30	0.375	-5.76	-0.78	-33.527	-6.706
00:27:30	0.458	-5.77	-0.78	-33.553	-6.711
00:32:30	0.541	-5.77	-0.78	-33.579	-6.716
00:37:30	0.625	-5.77	-0.78	-33.605	-6.721
00:42:30	0.708	-5.77	-0.78	-33.63	-6.726
00:47:30	0.791	-5.78	-0.78	-33.656	-6.732
00:52:30	0.875	-5.78	-0.78	-33.681	-6.737
00:57:30	0.958	-5.78	-0.78	-33.706	-6.742
01:02:30	1.041	-5.78	-0.78	-33.731	-6.747
01:07:30	1.125	-5.78	-0.78	-33.755	-6.751
01:12:30	1.208	-5.79	-0.78	-33.78	-6.756
01:17:30	1.291	-5.79	-0.78	-33.804	-6.761
01:22:30	1.375	-5.79	-0.78	-33.828	-6.766
01:27:30	1.458	-5.79	-0.78	-33.852	-6.771
01:32:30	1.541	-5.8	-0.78	-33.876	-6.776
01:37:30	1.625	-5.8	-0.78	-33.9	-6.78
01:42:30	1.708	-5.8	-0.78	-33.923	-6.785
01:47:30	1.791	-5.8	-0.78	-33.947	-6.79

There are only the first seven columns

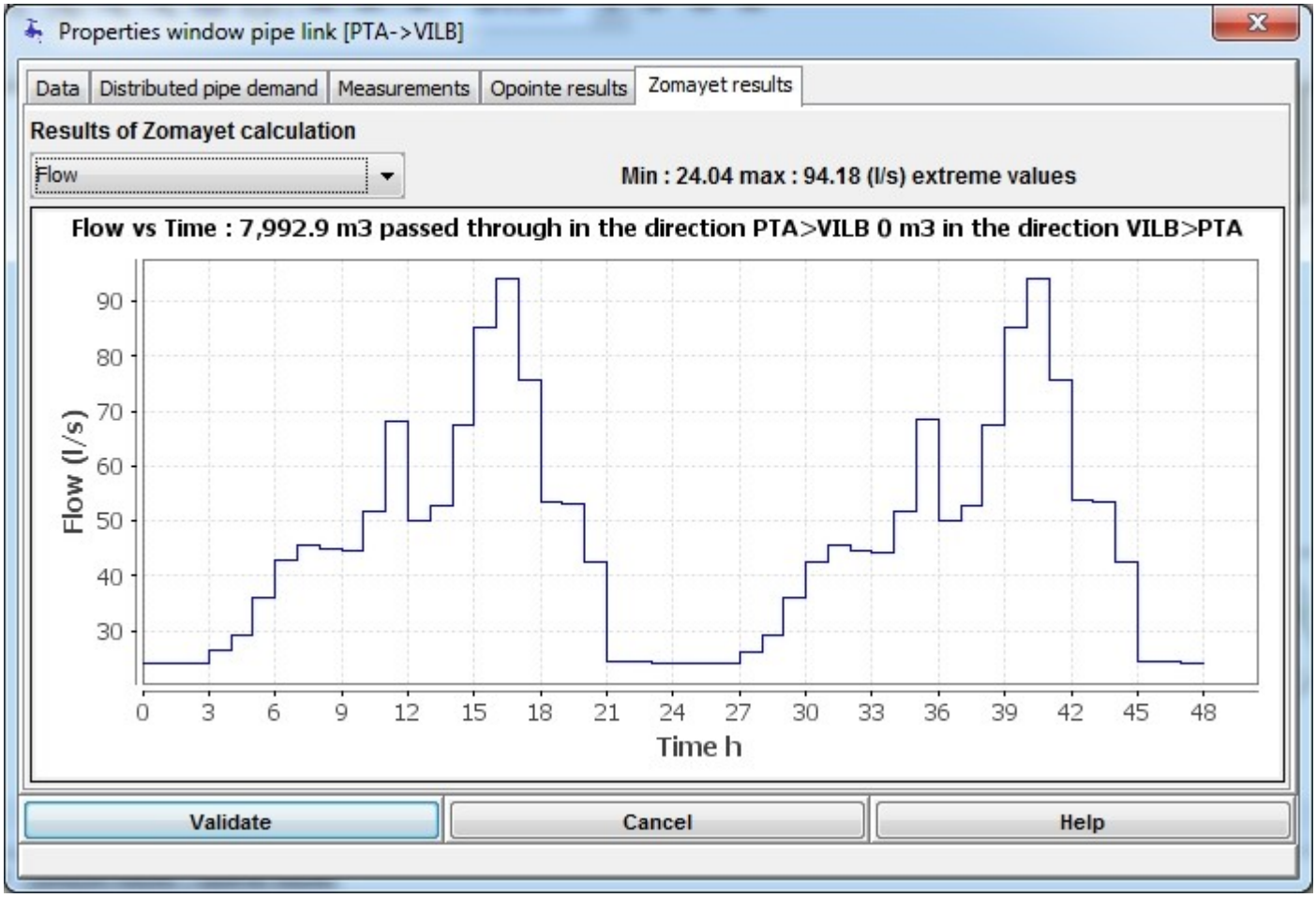


There are only the first five columns and the eighth

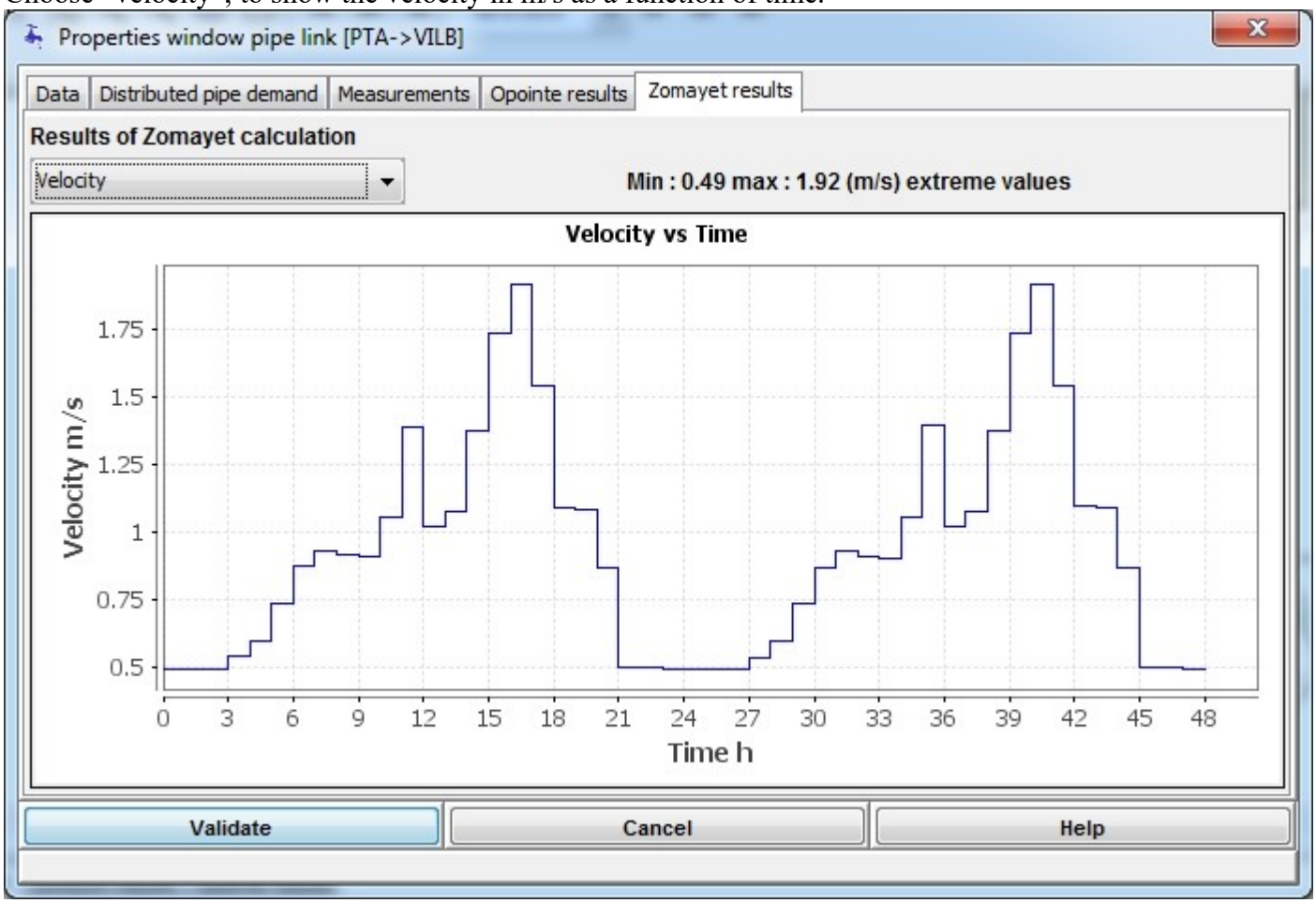


The following are the graphical representations that can be printed and exported in svg format (see the serie of buttons on the right, already presented elsewhere). A header for each graph shows the extreme values during the simulation period.

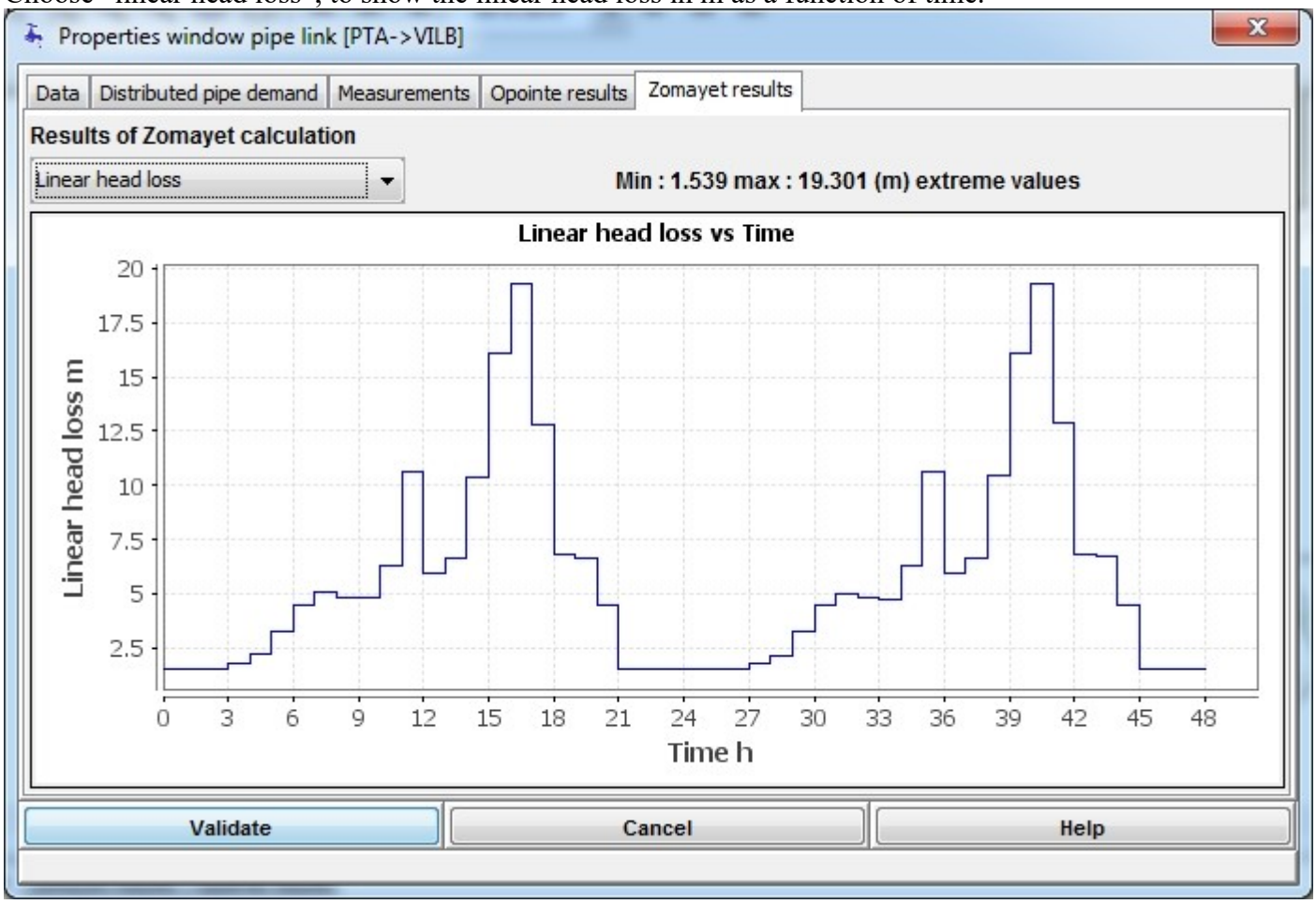
Choose “flow” to show the flow in l/s as a function of time.



Choose “velocity”, to show the velocity in m/s as a function of time.

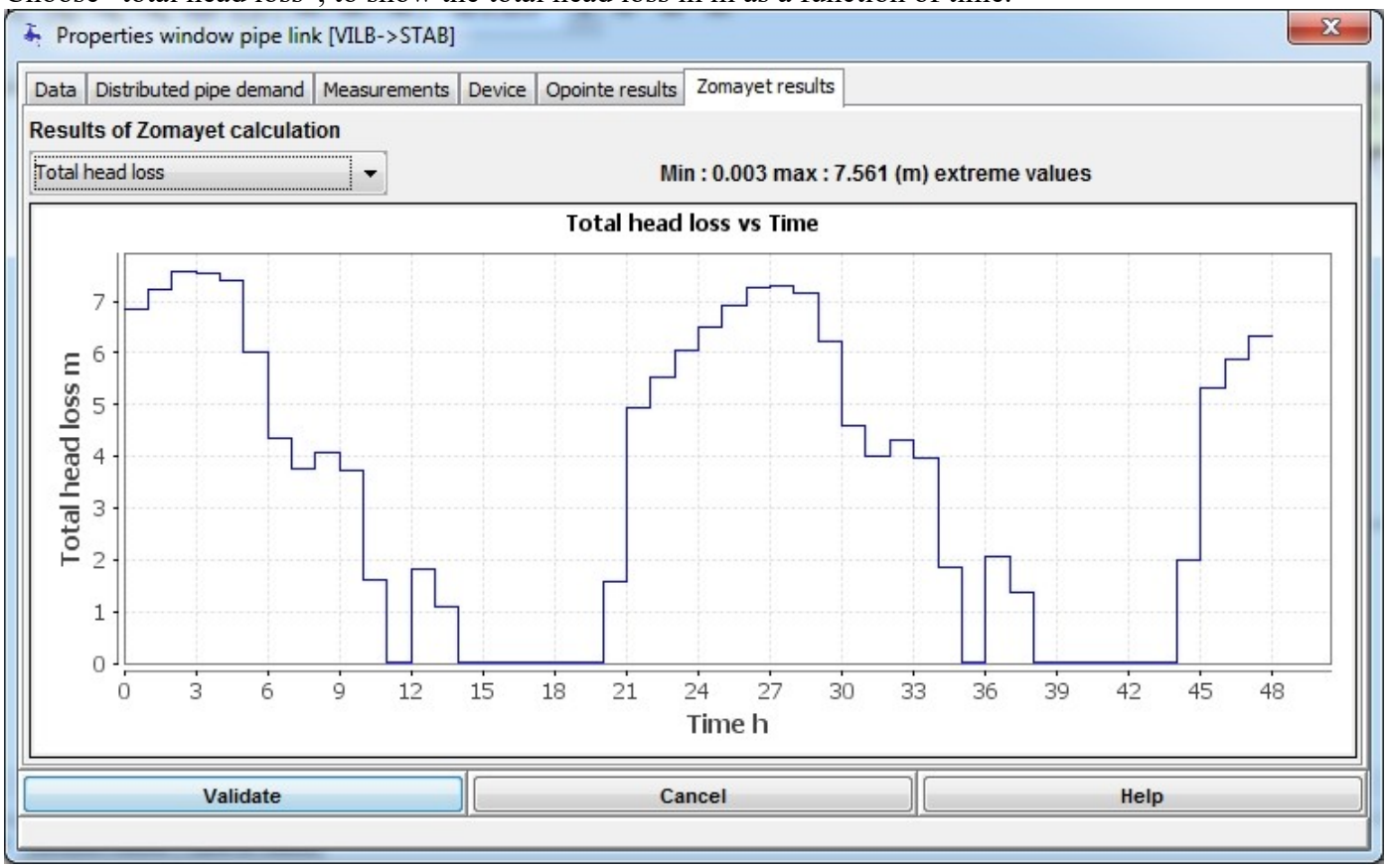


Choose “linear head loss”, to show the linear head loss in m as a function of time.

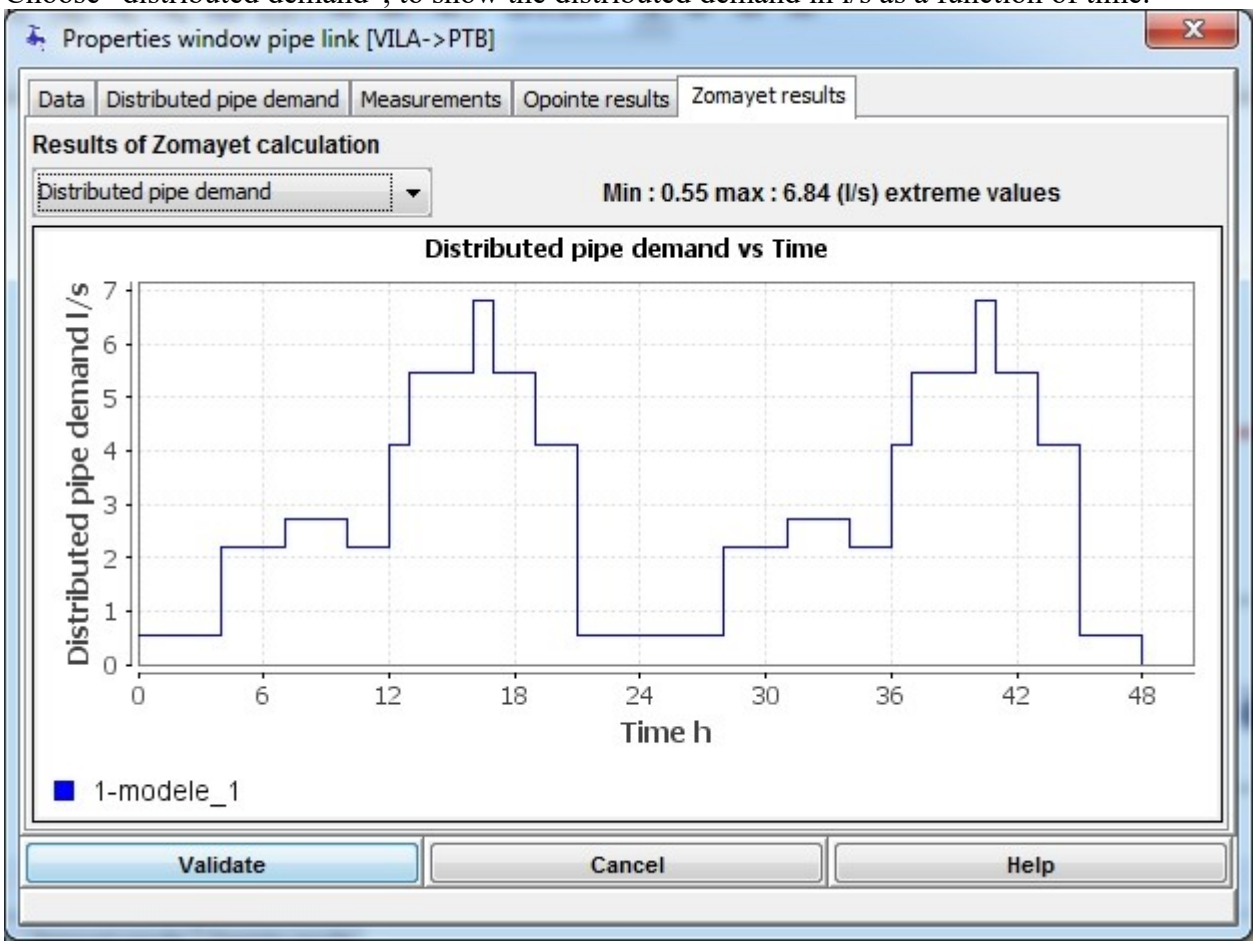


Choose “singular head loss”, to show the singular head loss in m as a function of time.

Choose “total head loss”, to show the total head loss in m as a function of time.



Choose “distributed demand”, to show the distributed demand in l/s as a function of time.

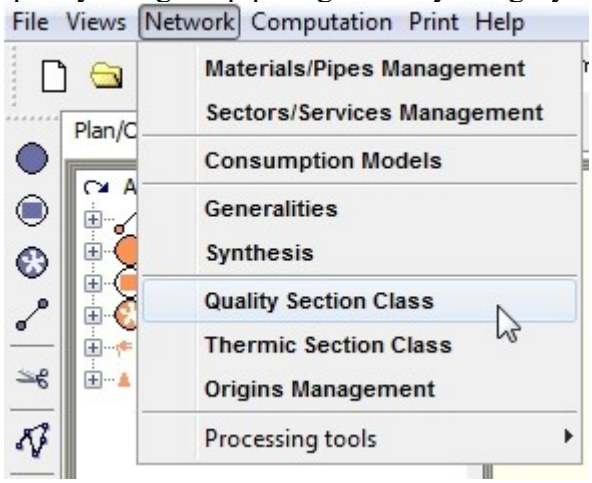


Remember, whenever regulation is applied to a hydraulic device, the results associated to this regulation are explained by following this **link**.

Quality

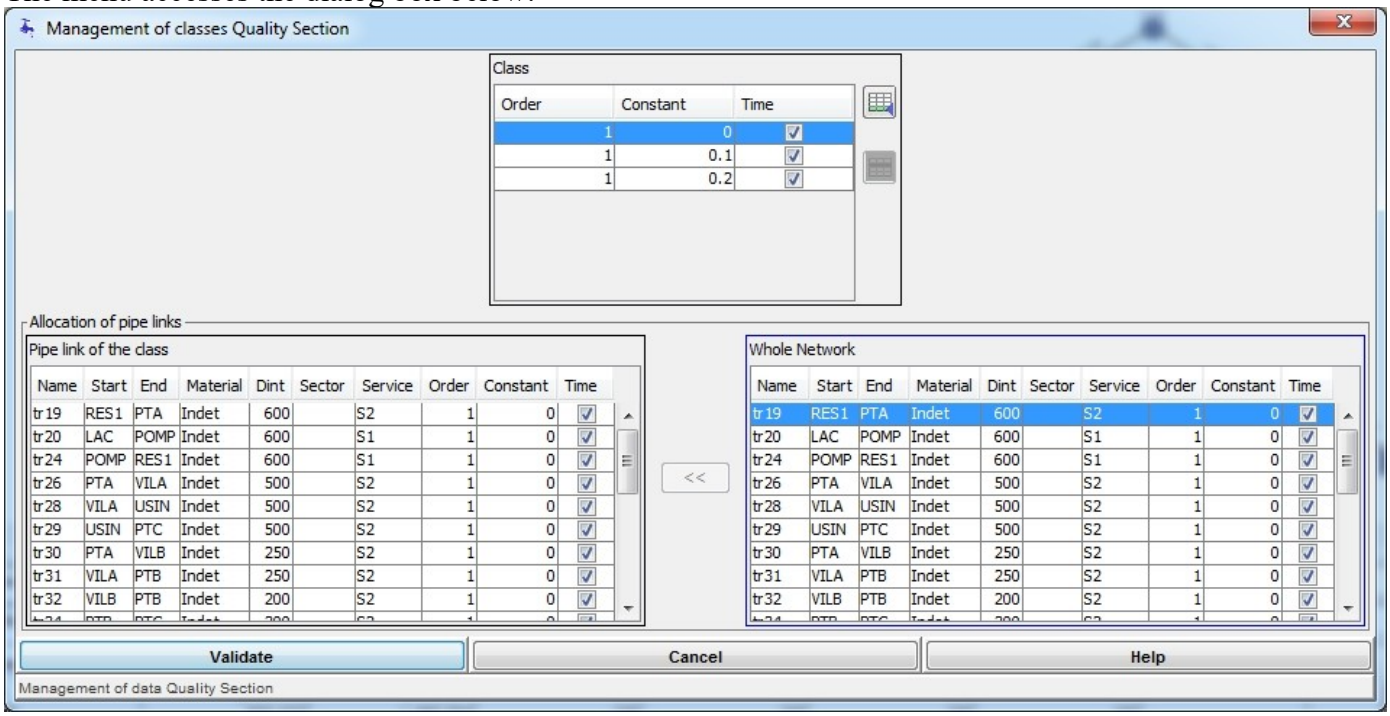
Managing Quality classes for the pipe segments

The menu below allows “quality” data to be specified allowing the simulation of the evolution of the quality along the pipe segments by category.



Each pipe is given parameters to simulate a kinetic law along the route.

The menu accesses the dialog box below:



Select a line in the table of classes (at the top), the example here has a unique class.
The values of order and kinetic constant can be modified. **If order is equal to 1, the unit of the kinetic constant is [h-1].**
If the checkbox of the time column is checked, the sections of the class participate in the ageing of the water, otherwise the age of the water does not change in these sections.
In the “Allocation of pipe segments” table appears a list of pipe segments allocated to this class.

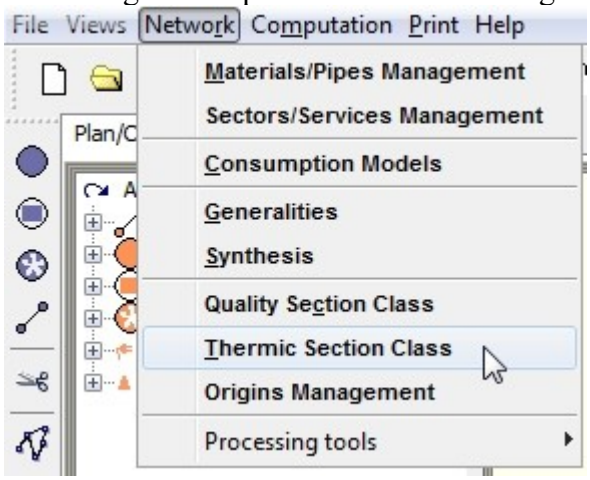
In the “Whole network” table appears a list of all pipe segments in the network.
To allocate the selected class to a pipe section in the network, select the line of the pipe in this table then click on the button, the pipe is transferred to the table on the left.

To add a quality class, click on the button.

To delete a quality class, click on the button. Warning, no other pipe segment must be allocated to a class for it to be deletable.

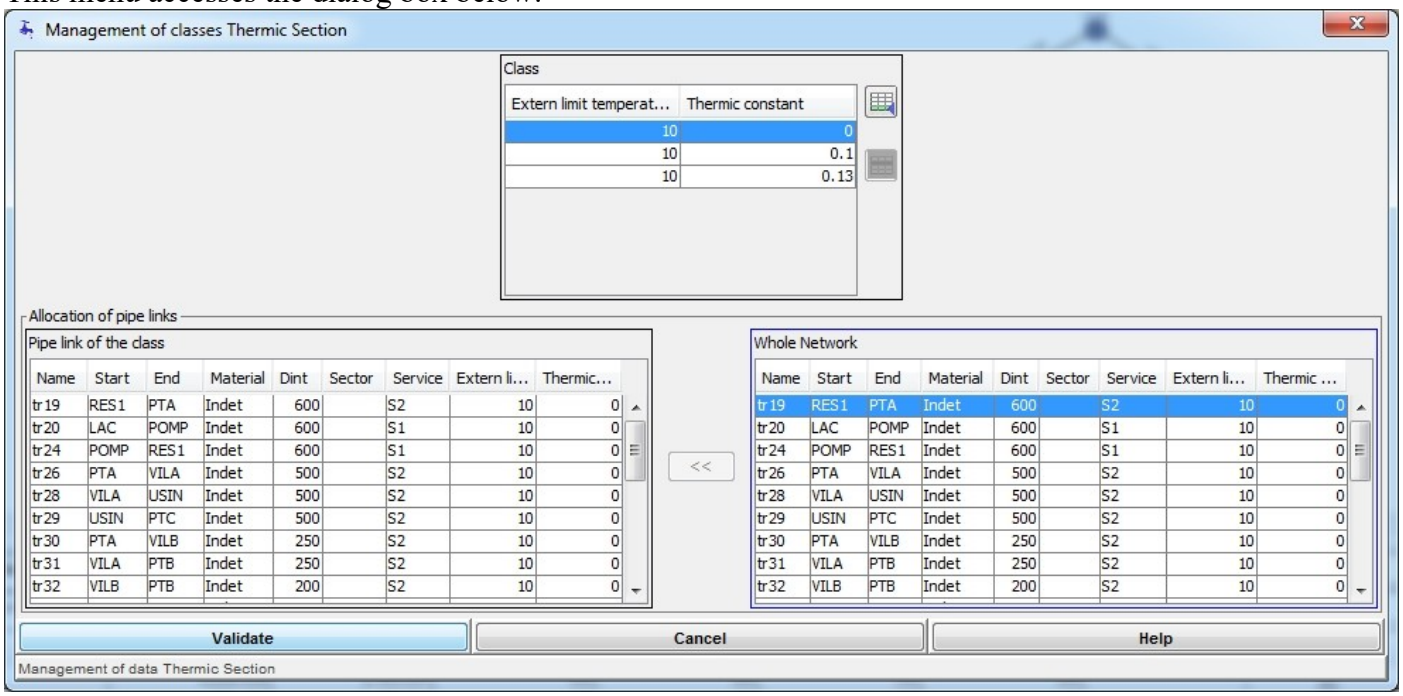
Thermic tronçons

The "Management of Thermic Class for Sections" menu below allows to specify "thermic" data simulating the temperature evolution along the pipes by family.



Each pipe is given parameters to simulate a law of thermic exchange with the outside along its way.

This menu accesses the dialog box below:



Select a line in the table of classes (at the top), the example here has a unique class.
The values of the exchange constant and of the extern limit temperature can be modified.
In the “Allocation of pipe segments” table appears a list of pipe segments allocated to this class.

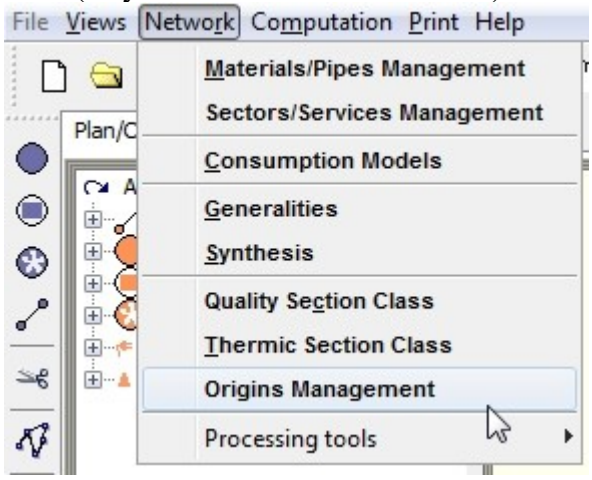
In the “Whole network” table appears a list of all pipe segments in the network.
To allocate the selected class to a pipe section in the network, select the line of the pipe in this table then click on the button, the pipe is transferred to the table on the left.

To add a thermic class, click on the button.

To delete a thermic class, click on the button. Warning, no other pipe segment must be allocated to a class for it to be deletable.

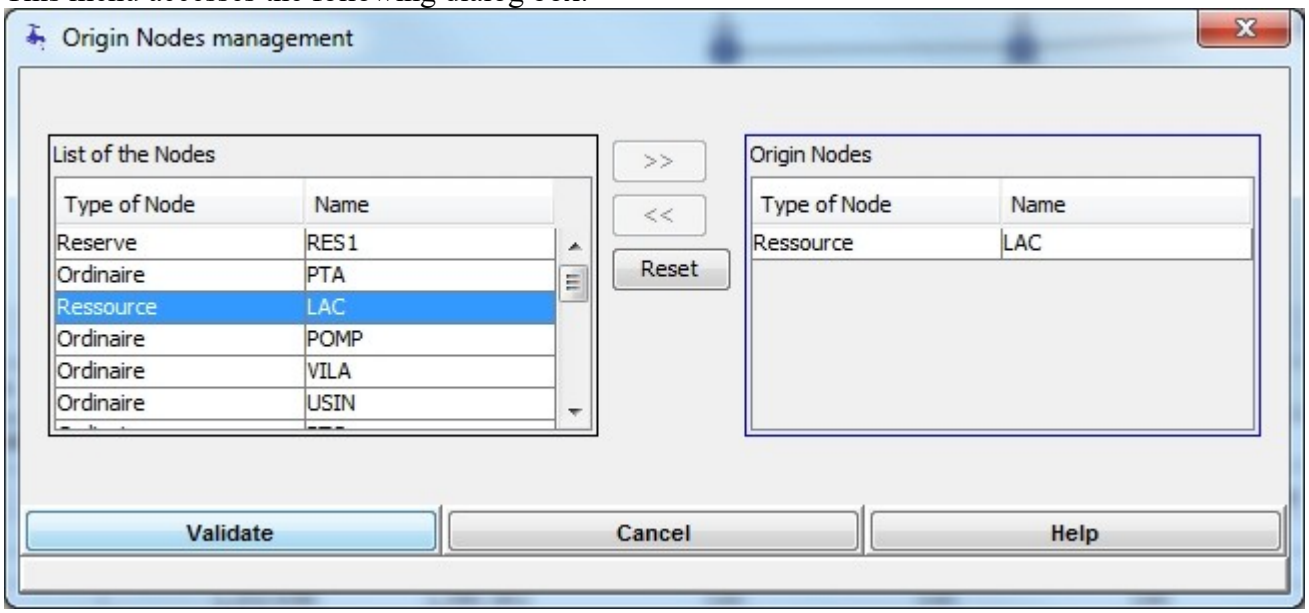
Management of source nodes

The menu below allows “quality” data to be specified allowing the tracking of the arrival of water at a node (Keyboard shortcut: <Alt>+R+P).



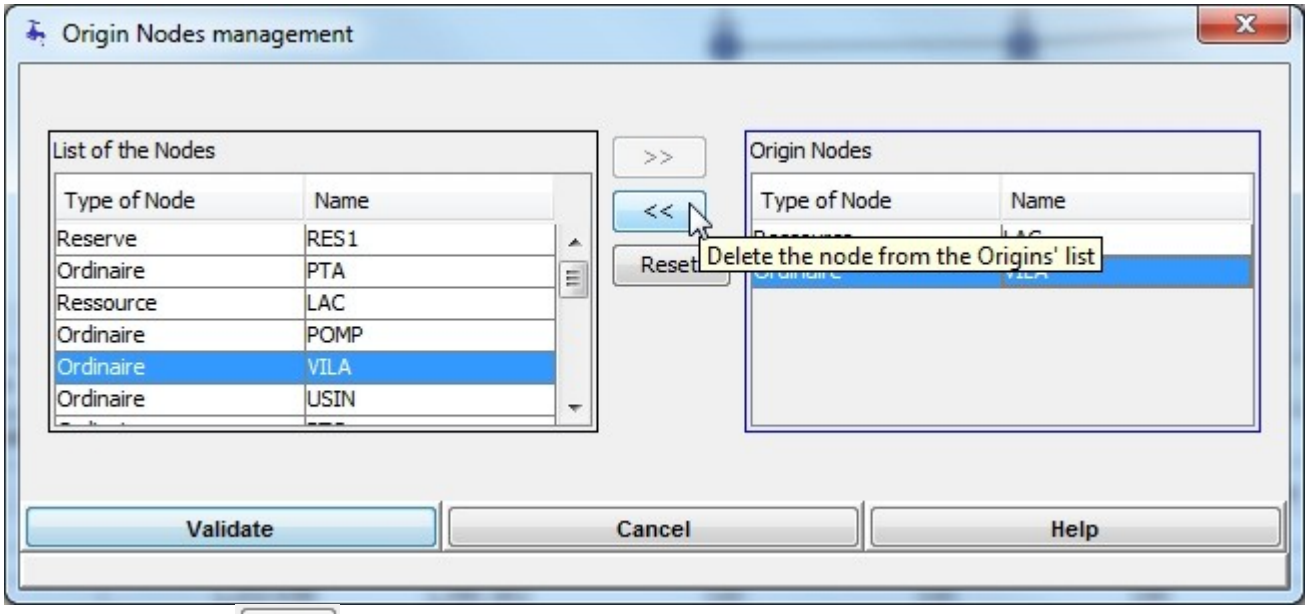
It concerns the identification of nodes from which for all network nodes you wish to know the percentage of water arriving from these designated nodes.

This menu accesses the following dialog box:



Select a node from the “List of nodes” in the network (the left hand table), in this example, the node GRATELOU. To pass it to the “Source nodes” list (the right hand table), click on the button.

To remove a node from the “Source nodes” list, select it as below for the VESVRES node.



Then click on the << button.

To delete all the nodes in the “Source nodes” list, click on the Reset button.
At the bottom of the dialog box, there are three buttons:

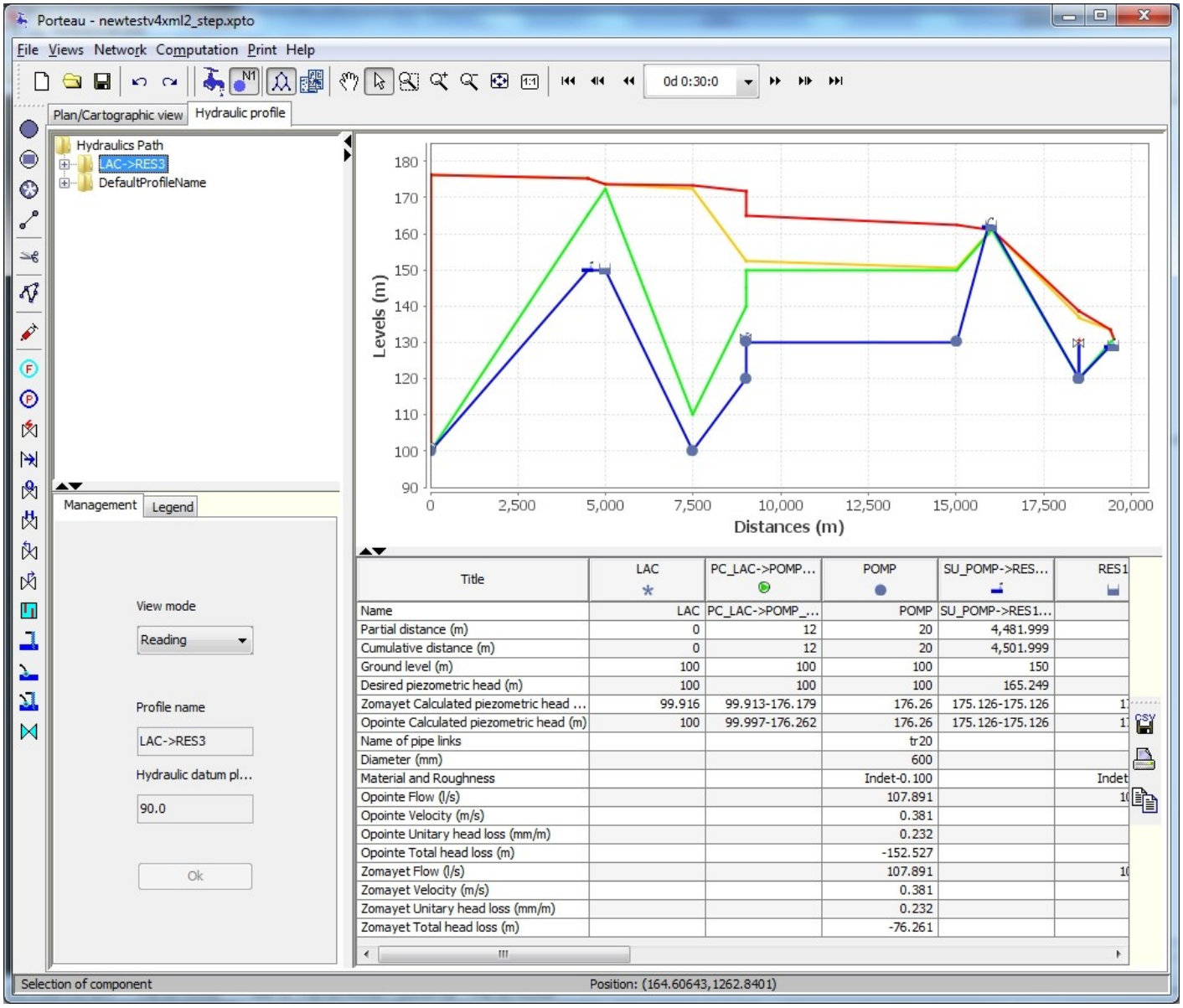
Validate to save and exit the dialog box.

Cancel to cancel and exit the dialog box.

Help to obtain help.

The detailed profile

Presentation
The detailed profiles are edited from the “Detailed profile” tab. The list of existing profiles is found in the navigator at the top left. This list may be empty. If no profile is selected, the basic window appears as below:



Title	LAC	PC_LAC->POMP...	POMP	SU_POMP->RES...	RES1
Name	LAC	PC_LAC->POMP_...	POMP	SU_POMP->RES1...	
Partial distance (m)	0	12	20	4,481.999	
Cumulative distance (m)	0	12	20	4,501.999	
Ground level (m)	100	100	100	150	
Desired piezometric head (m)	100	100	100	165.249	
Zomayet Calculated piezometric head ...	99.916	99.913-176.179	176.26	175.126-175.126	1
Opointe Calculated piezometric head (m)	100	99.997-176.262	176.26	175.126-175.126	1
Name of pipe links			tr20		
Diameter (mm)			600		
Material and Roughness			Indet-0.100		Indet
Opointe Flow (l/s)			107.891		1
Opointe Velocity (m/s)			0.381		
Opointe Unitary head loss (mm/m)			0.232		
Opointe Total head loss (m)			-152.527		
Zomayet Flow (l/s)			107.891		1
Zomayet Velocity (m/s)			0.381		
Zomayet Unitary head loss (mm/m)			0.232		
Zomayet Total head loss (m)			-76.261		

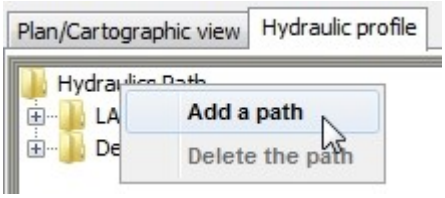
NB: we notice the following windows:

1. at the top left: Browser of detailed profiles
2. at the top right: Graphs or curves of the selected detailed profile
3. at the bottom left:
    1. Parameters tab of the selected detailed profile
    2. Legend tab of the graph or the curves of the selected detailed profile
4. at the bottom right: Data and results table of the selected detailed profile

Select a detailed profile
A detailed profile is shown by selecting the name of the profile in the browser windows, and nodes and pipes are also selected by selecting their name, once the correct profile has been chosen.

Add a detailed profile
To add a detailed profile, click right on the root of the list of profiles and choose “add a list”.

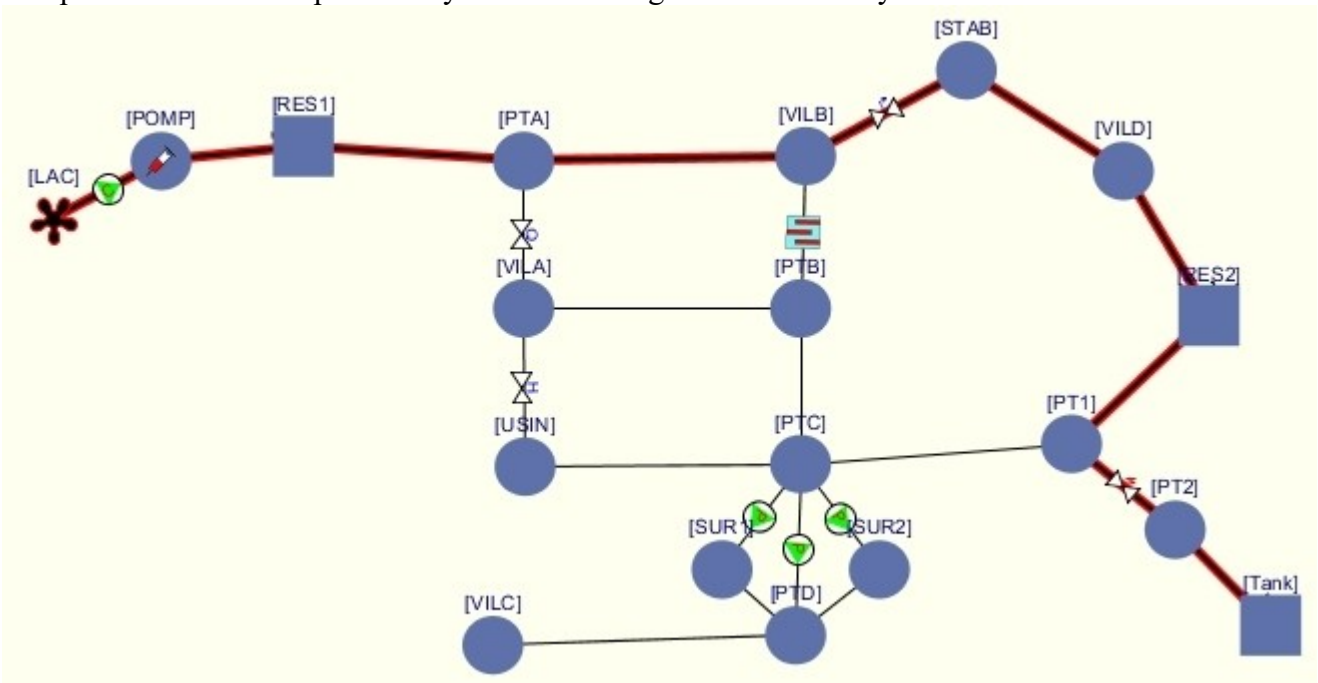
Delete a selected profile
To delete a detailed profile, click right on the profile of the list of profiles and choose “delete the list”.



Create a detailed profile

From an existing network, a detailed profile is created by first adding a list in the window of detailed profiles. Then to create the profile in the graphic tab, click on the hydraulic profile icon symbolised by  in the button bar at the left of the main window.

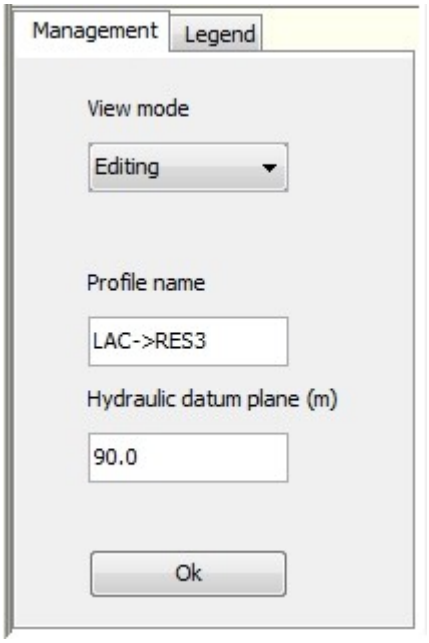
Then, in the graphics window, use the cursor  to position a source node for the network.
This node is the start of a path of pipe segments that is highlighted when clicked on.
The path of the detailed profile stays visible as long as the cursor stays active:



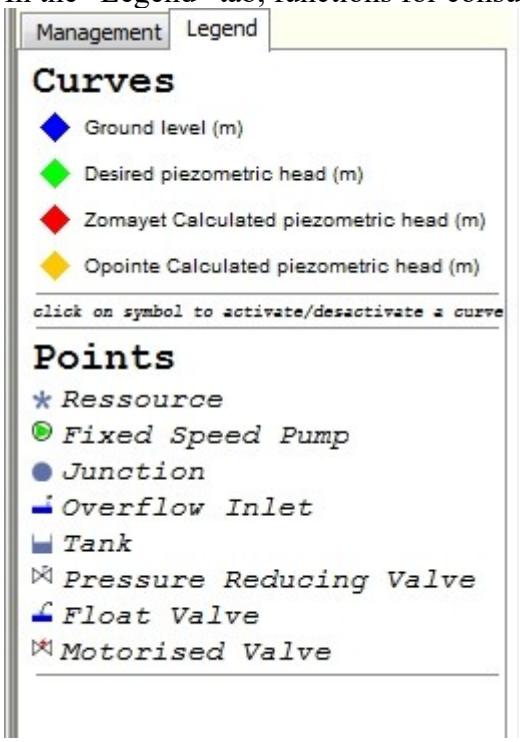
To deselect the active marker, click on the  icon in the main window.

Read/Modify parameters

In the “Management” tab, from the detailed profile window, the profile's parameters and consultation can be managed in the management/legend area.



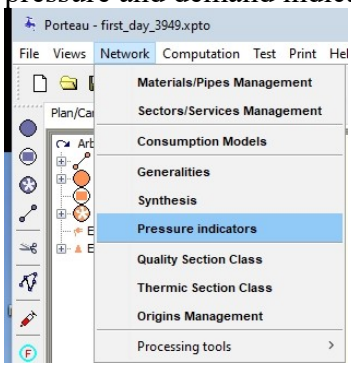
To modify the parameters of the displayed profile, choose a display mode: “Edit”, and validate changes.
The PHR head allows the origin heads of the graphic to be regulated.
In the “Legend” tab, functions for consulting the detailed profiles are shown:



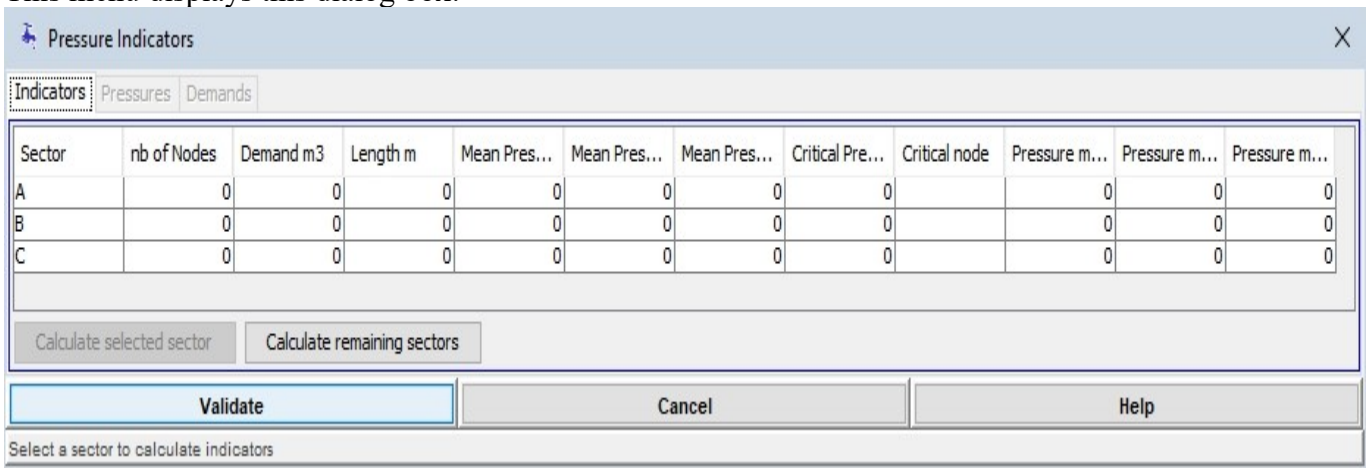
The legend for piezometric profiles for Opointe and Zomayet is shown only if the results of theses modules are present.

Pressure Indicators

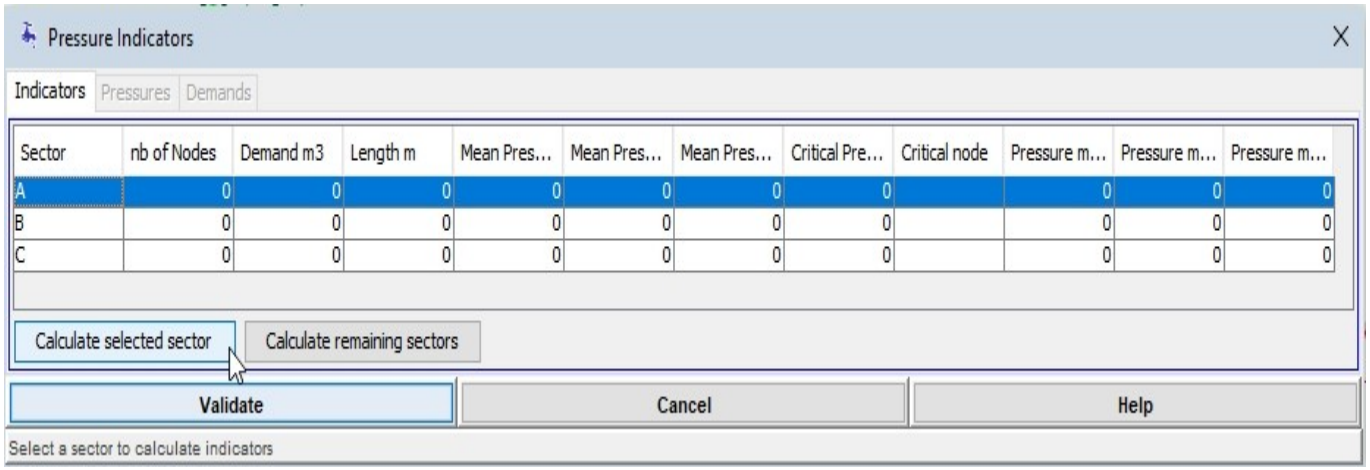
The menu below "Pressure Indicators" allows to compute sector by sector or for all of the network some pressure and demand indicators.

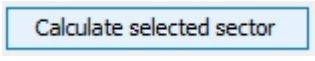


This menu displays this dialog box:

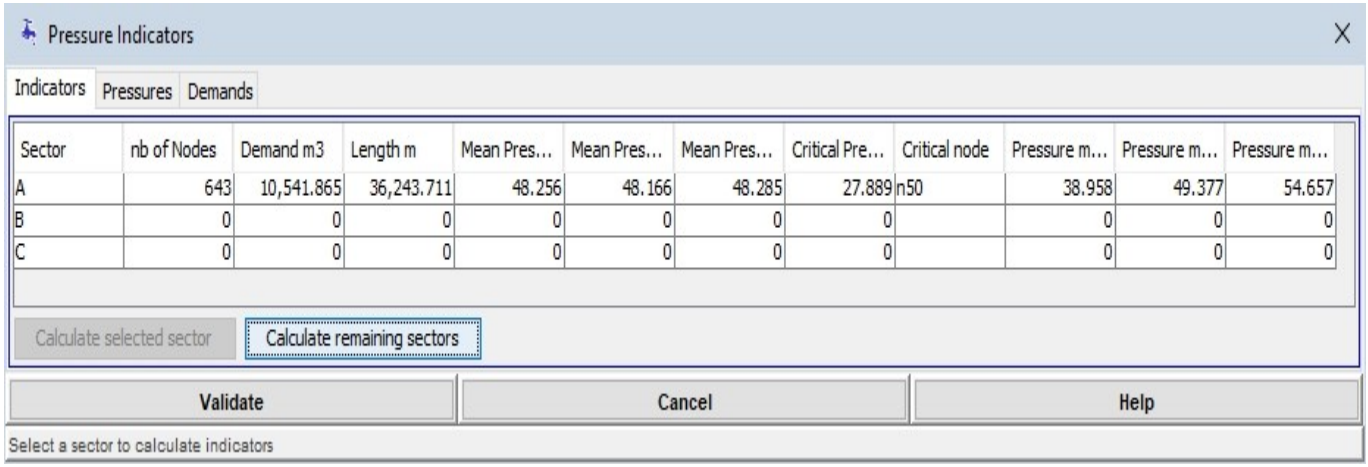


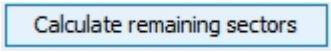
The dialog is composed of three tabs. The first tab gives a sector values for each line of the table.
If one line or more are selected (multiple selection allowed), the table displays the selection and the calculation buttons are activated.



Click on the button 

launches the computation of pressures and demands for the selected sectors. Results are stored in tables of the two tabs "Pressures" and "Demands", and each line of the table in the current tab is updated with the results of indicators done with Zomayet results.

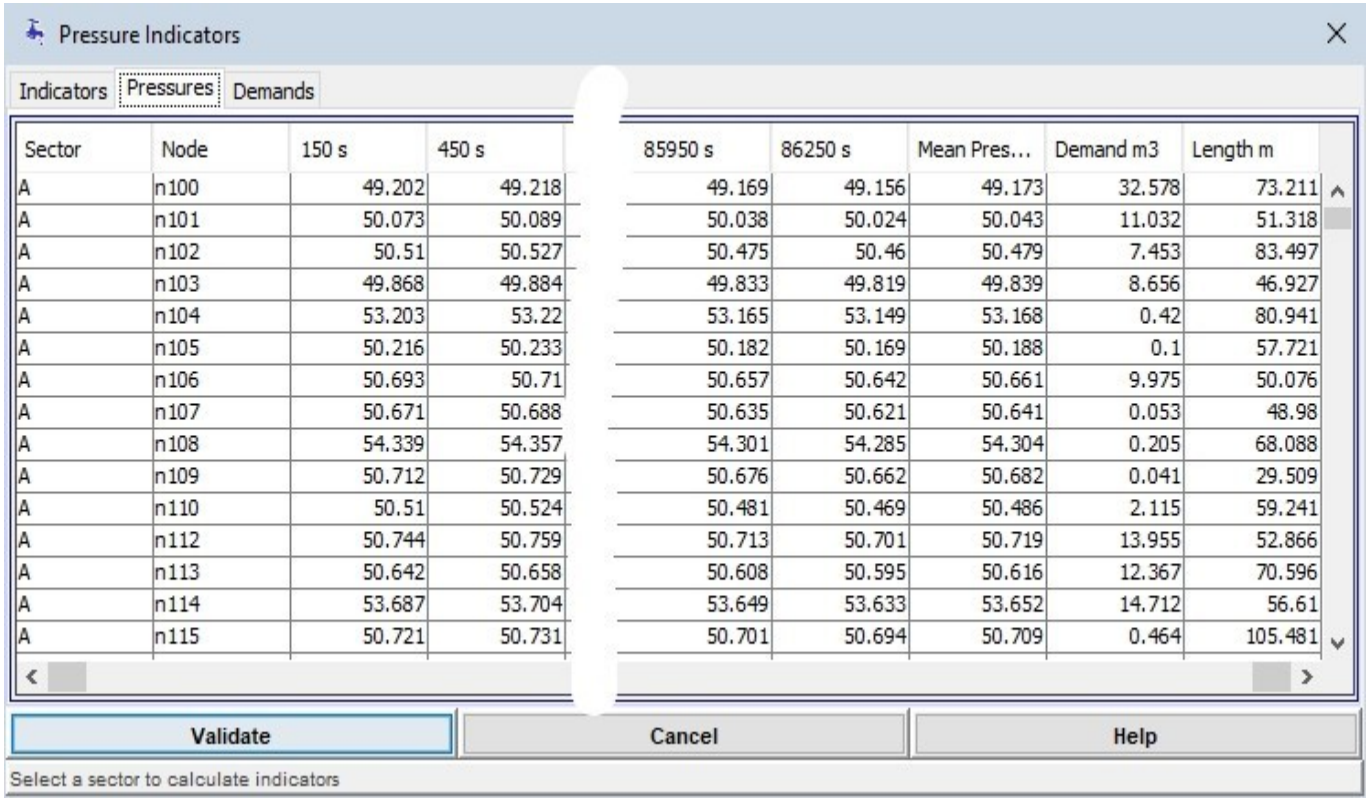


Click on the button 

launches the computation of indicators for the sectors not calculated before.
Indicators columns are:

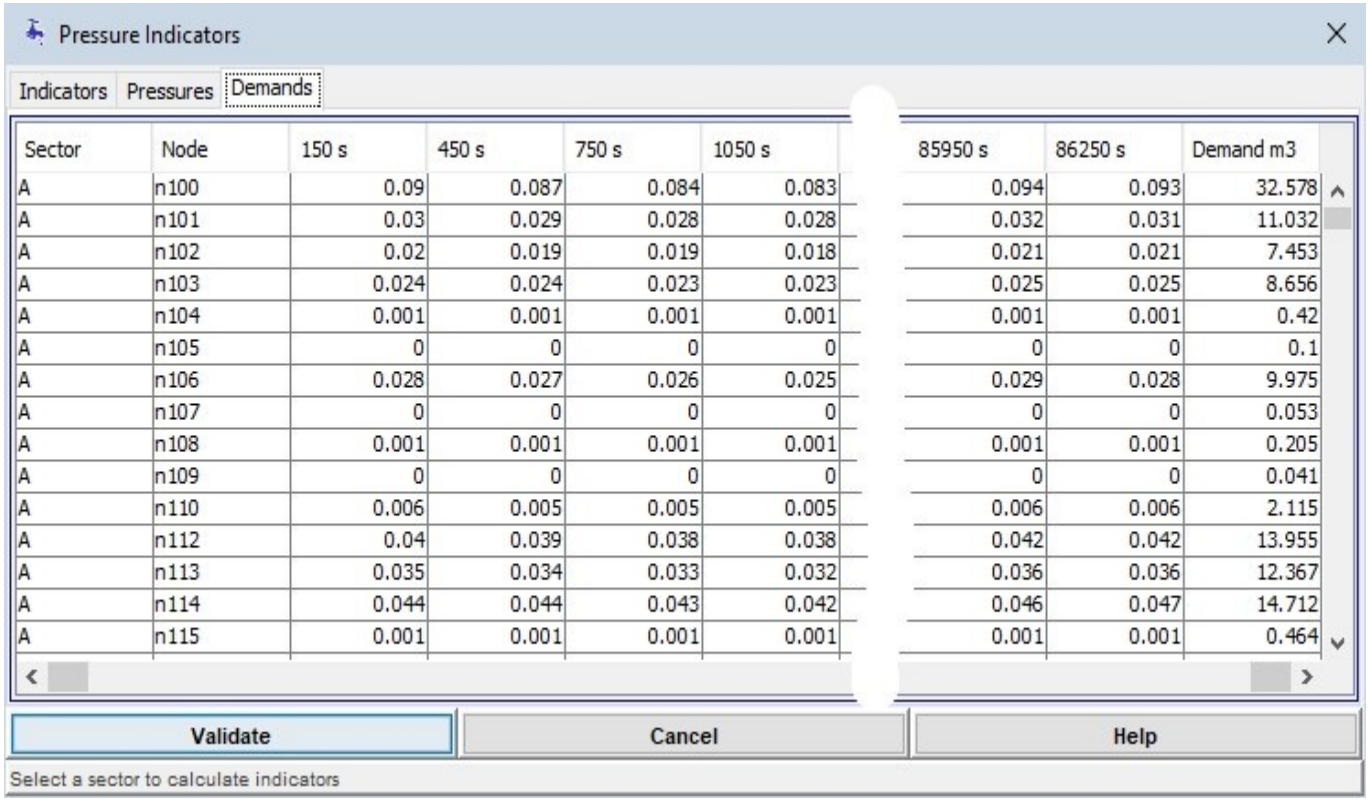
- Sector: the name of the sector
- nb de Nodes: the number of nodes of the sector
- Demand m3: the total demand of the sector calculated for the duration of Zomayet simulation in en m^3.
- Length m: the cumulated length for all links of the sector
- Mean Pressure m: the mean of pressure for all time steps of Zomayet and all nodes of the sector
- Mean Pressure / Length: the mean pressure weighted by the length of adjacent links of the node for the sector = sum for all nodes of the sector (mean pressure * 1/2 length of adjacent links of he node) / sum of length of links for the sector
  Caution: only pressure for nodes with demand are taken in account, but all the length are summed.
- Mean Pressure / Demand: the mean pressure weighted by the demand of adjacent links and the demande of the nodefor the sector = sum for all nodes of the sector (mean * (node + 1/2 adj link demand) / sum of all demands for the sector
- Critical Pressure m : the critical pressure (minimum of all Zomayet time steps) for the sector, only nodes with demands are considered
- Critical Node: Node where the critical pressure is measured
- Pressure max 10 %: the mean pressure for the first decile of sector mean pressures
- Pressure max 50 %: the mean pressure for the fifth decile of sector mean pressures
- Pressure max 90 %: the mean pressure for the ninth decile of sector mean pressures

The "Pressure" Tab shows for each node of calculated sectors one line with all pressures for all time steps of Zomayet simulation and at the end of the line values of "Mean Pressure", "Demand in m3" for the simulation duration, "Length in m" for adjacents links of the node divided by 2.



Sector	Node	150 s	450 s	85950 s	86250 s	Mean Pres...	Demand m3	Length m
A	n100	49.202	49.218	49.169	49.156	49.173	32.578	73.211
A	n101	50.073	50.089	50.038	50.024	50.043	11.032	51.318
A	n102	50.51	50.527	50.475	50.46	50.479	7.453	83.497
A	n103	49.868	49.884	49.833	49.819	49.839	8.656	46.927
A	n104	53.203	53.22	53.165	53.149	53.168	0.42	80.941
A	n105	50.216	50.233	50.182	50.169	50.188	0.1	57.721
A	n106	50.693	50.71	50.657	50.642	50.661	9.975	50.076
A	n107	50.671	50.688	50.635	50.621	50.641	0.053	48.98
A	n108	54.339	54.357	54.301	54.285	54.304	0.205	68.088
A	n109	50.712	50.729	50.676	50.662	50.682	0.041	29.509
A	n110	50.51	50.524	50.481	50.469	50.486	2.115	59.241
A	n112	50.744	50.759	50.713	50.701	50.719	13.955	52.866
A	n113	50.642	50.658	50.608	50.595	50.616	12.367	70.596
A	n114	53.687	53.704	53.649	53.633	53.652	14.712	56.61
A	n115	50.721	50.731	50.701	50.694	50.709	0.464	105.481

The "Demands" Tab shows for each node of calculated sectors one line with all the demands values for all time steps of Zomayet simulation and at the end of the line the value of total demand of the node in m3 for the simulation.

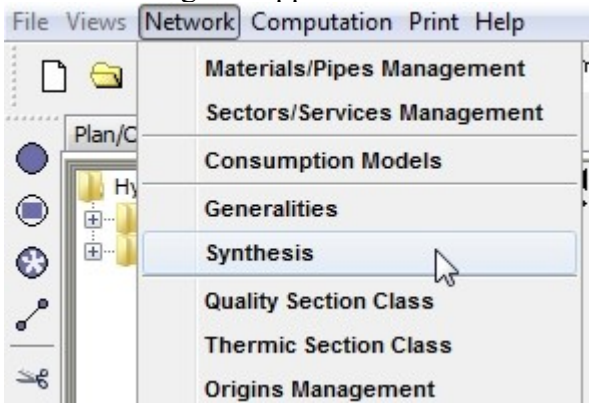


Sector	Node	150 s	450 s	750 s	1050 s	85950 s	86250 s	Demand m3
A	n100	0.09	0.087	0.084	0.083	0.094	0.093	32.578
A	n101	0.03	0.029	0.028	0.028	0.032	0.031	11.032
A	n102	0.02	0.019	0.019	0.018	0.021	0.021	7.453
A	n103	0.024	0.024	0.023	0.023	0.025	0.025	8.656
A	n104	0.001	0.001	0.001	0.001	0.001	0.001	0.42
A	n105	0	0	0	0	0	0	0.1
A	n106	0.028	0.027	0.026	0.025	0.029	0.028	9.975
A	n107	0	0	0	0	0	0	0.053
A	n108	0.001	0.001	0.001	0.001	0.001	0.001	0.205
A	n109	0	0	0	0	0	0	0.041
A	n110	0.006	0.005	0.005	0.005	0.006	0.006	2.115
A	n112	0.04	0.039	0.038	0.038	0.042	0.042	13.955
A	n113	0.035	0.034	0.033	0.032	0.036	0.036	12.367
A	n114	0.044	0.044	0.043	0.042	0.046	0.047	14.712
A	n115	0.001	0.001	0.001	0.001	0.001	0.001	0.464

Synthesis

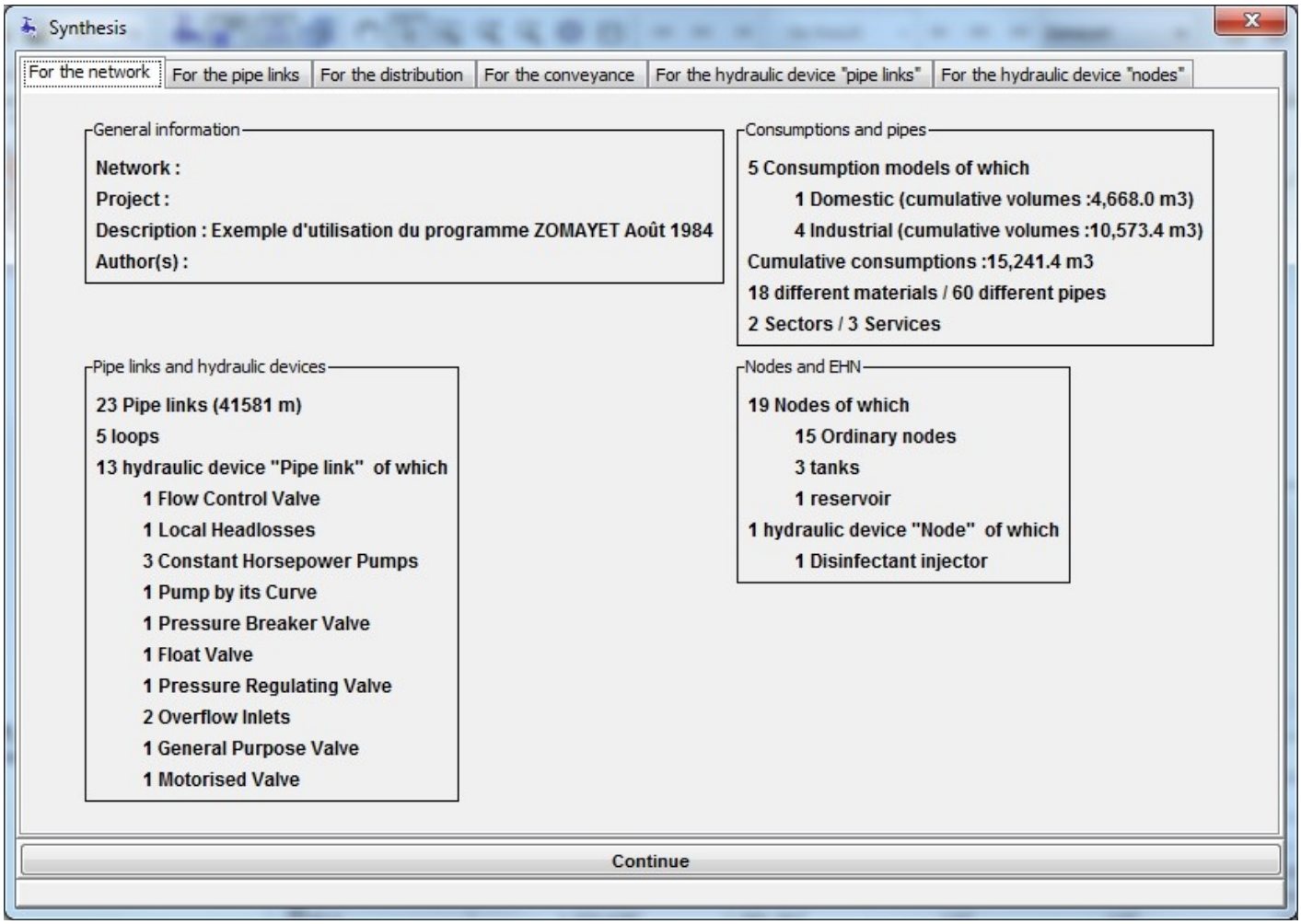
Synthesis of the collection of data comprising the project studied

Select “Synthesis” in the menu “Network” (Keyboard shortcut: <Alt>+R+S). The Synthesis of the network dialog box appears...



It contains six tabs for a synthesis of the different elements classed by families: general, pipe segments, distribution, supply, pipe hydraulic devices and node hydraulic devices.

Synthesis for the network

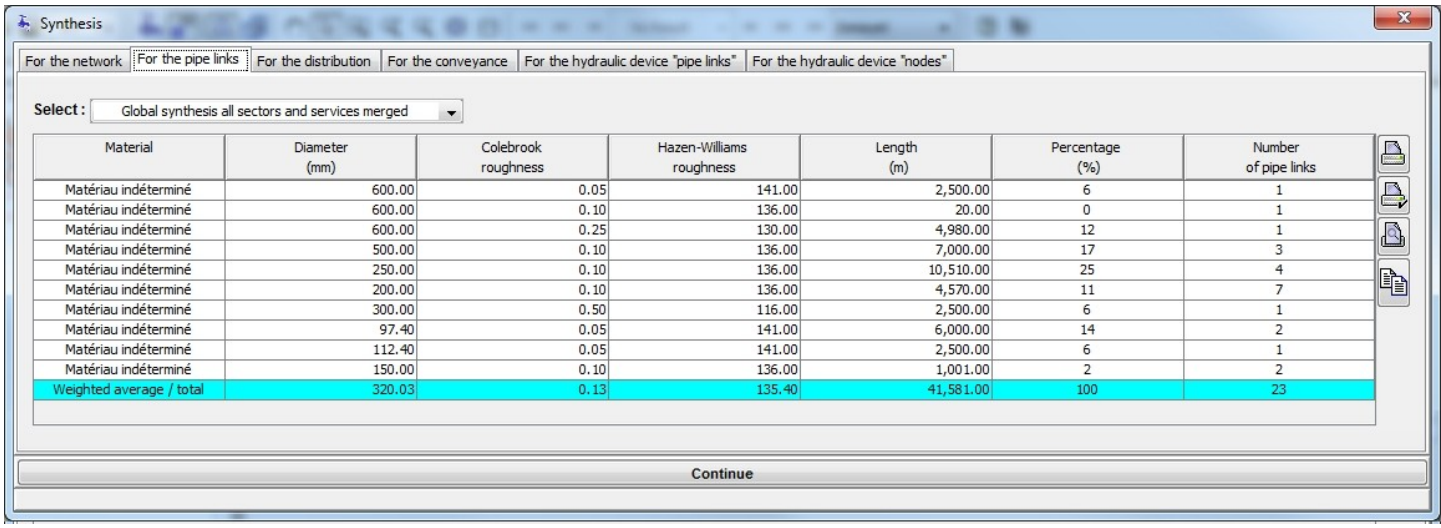


- “General information” shows general information linked to the current project including company, network and author(s).

- “Consumptions and pipes” presents the numbers of each of the different types of consumption model as well as the number of materials and pipe types described.
- “Pipe segments and their hydraulic devices” shows the total number of pipes, the loops as well as the number of hydraulic devices by type.
- “Nodes and their hydraulic devices” shows the total number of nodes and number per type (ordinary, tank, source) as well as the number of hydraulic devices by type.

The Continue button closes the dialog box (identical to all the tabs concerning the synthesis).

Synthesis for the pipe links



A table presents the statistics by material and by diameter, including notably the length per pipe type, the percentage of the total length and the number of pipe segments belonging to this pipe type.
The last line, highlighted in cyan shows information on the average type weighted by the length for the diameter and roughness columns, and by the total types for the length, percentage and number of pipes columns.
Remark 1: all the tables presented in this “Synthesis” chapter, including the column headers can be transferred to other programs by selecting using <Ctrl>+A and copying <Ctrl>+C then pasting.
Remark 2: on the right of all the synthesis tabs with the table are three icons which have the following roles:

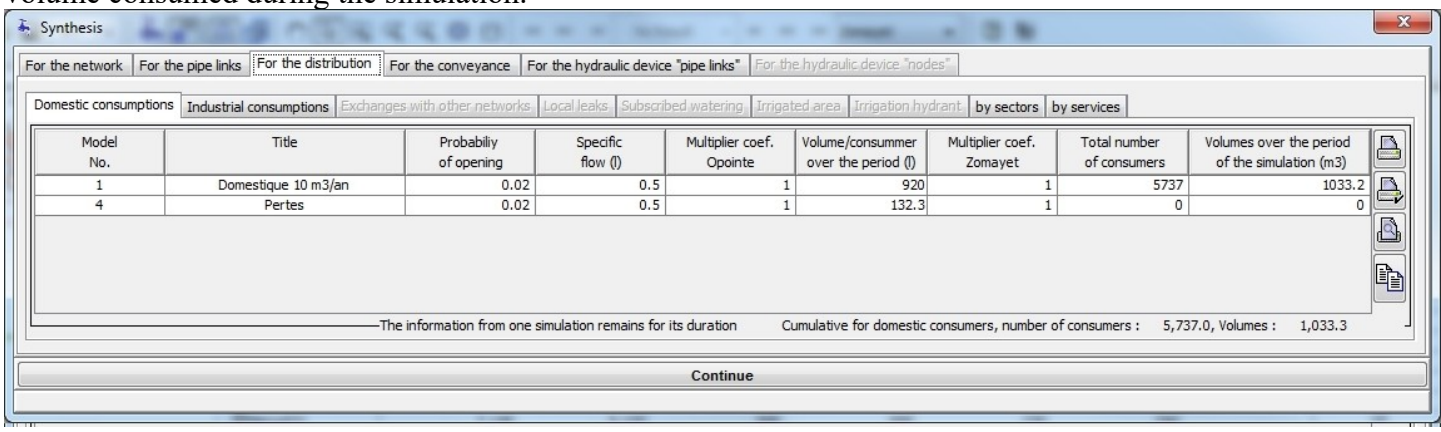
- prints the table with the default size of worksheet, margins and orientation to the default printer.
- opens the print properties dialog, which includes choice of printer (including generating pdf files if PdfCreator ® or such like is installed), the worksheet orientation (portrait or landscape), the paper format and margins. These values, once modified and not cancelled, apply to all subsequent print operations during the Porteau session or until the next modification.
- provides a print preview. A dialog box opens and shows the format of the page that will be printed. A button allows the printing to go ahead and a button closes the dialog box without printing. A scroll list allows the scale to be chosen, either 10, 25, 50 or 100%.

Synthesis for the distribution

A group of 7 tabs is proposed to detail the synthesis of consumptions.

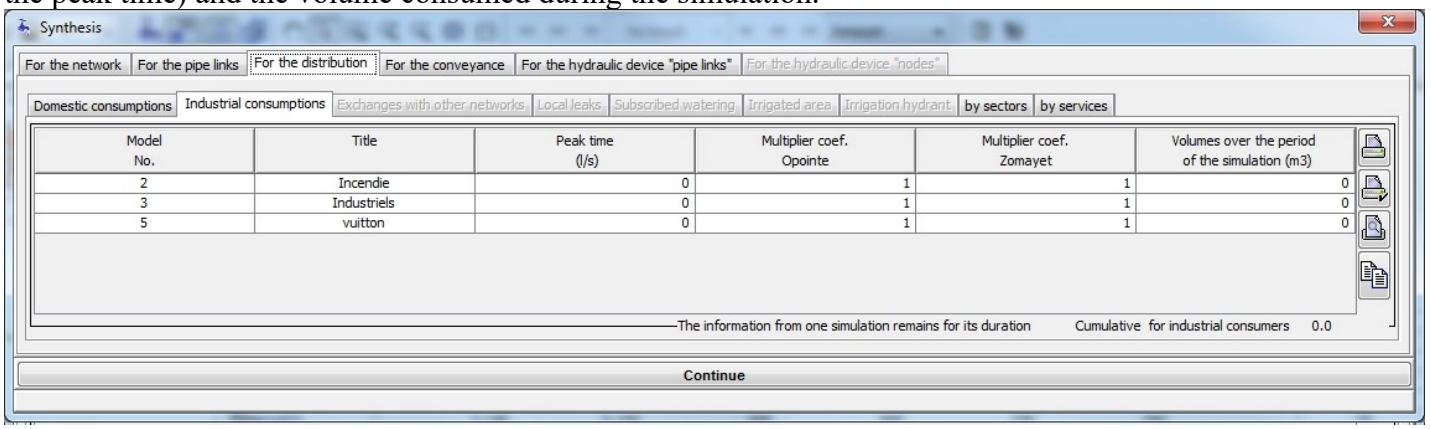
Domestic consumptions

This table shows the number of the consumption model, the title, the total number of users and the volume consumed during the simulation.



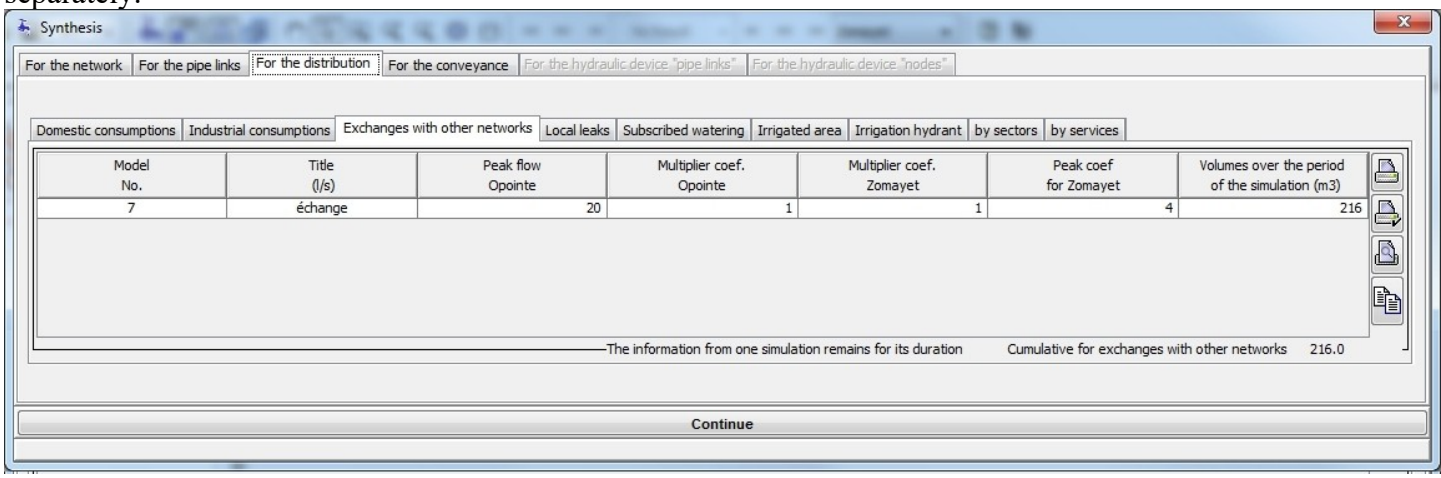
Industrial consumptions

This table shows the number of the consumption model, the title, the peak time (sum of consumptions at the peak time) and the volume consumed during the simulation.



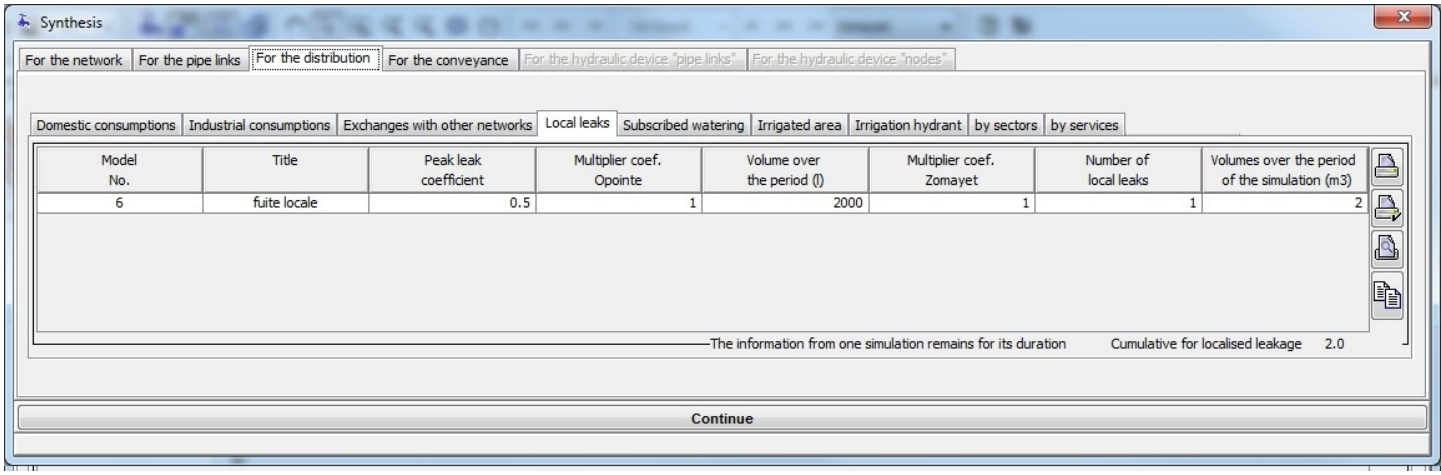
Exchanges with other networks

This table presents a report of the exchanges with other networks allowing these volumes to be identified separately.



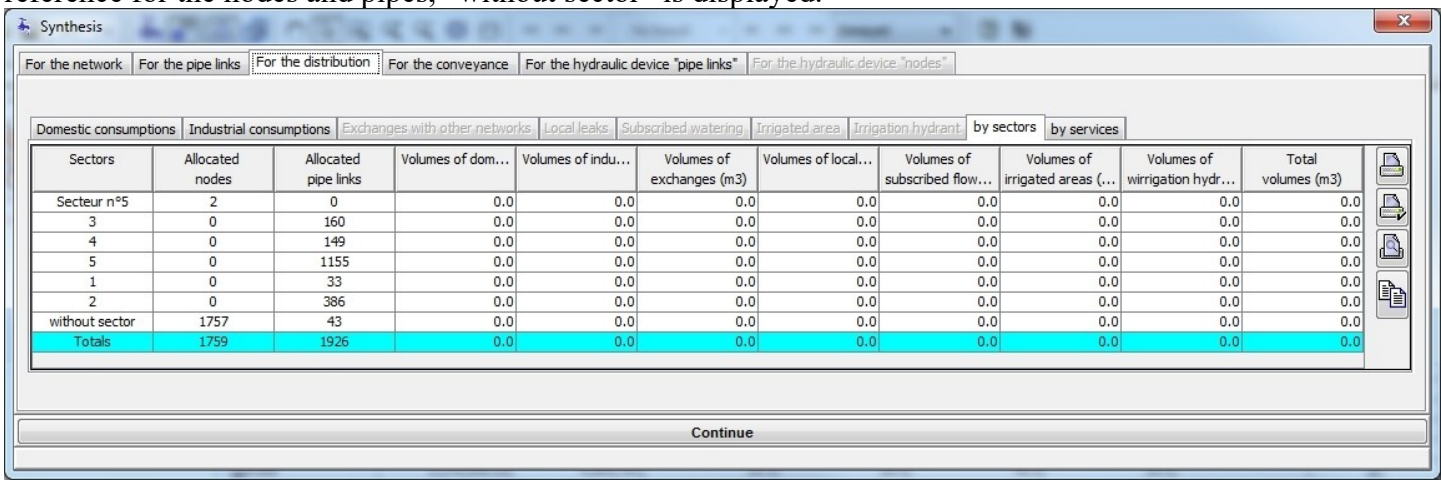
Localised leakages

This table presents the synthesis of the volumes, clearly identified in the consumption models as leakage.



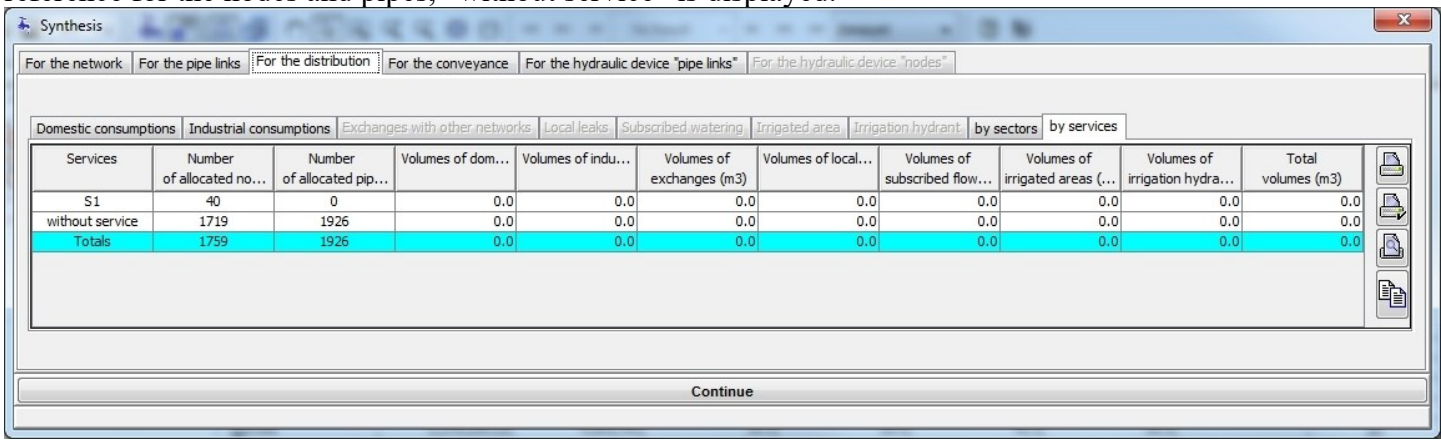
By sectors

This table gives information on the number of nodes and pipe segments allocated to each sector described, and for each family of consumption (domestic, industrial, exchange with other network, localised leakage) the volumes of water consumed during the period of simulation (24 hours by default). The last line of the table highlighted in cyan shows the total for each column. In the absence of a sector reference for the nodes and pipes, “without sector” is displayed.



By services

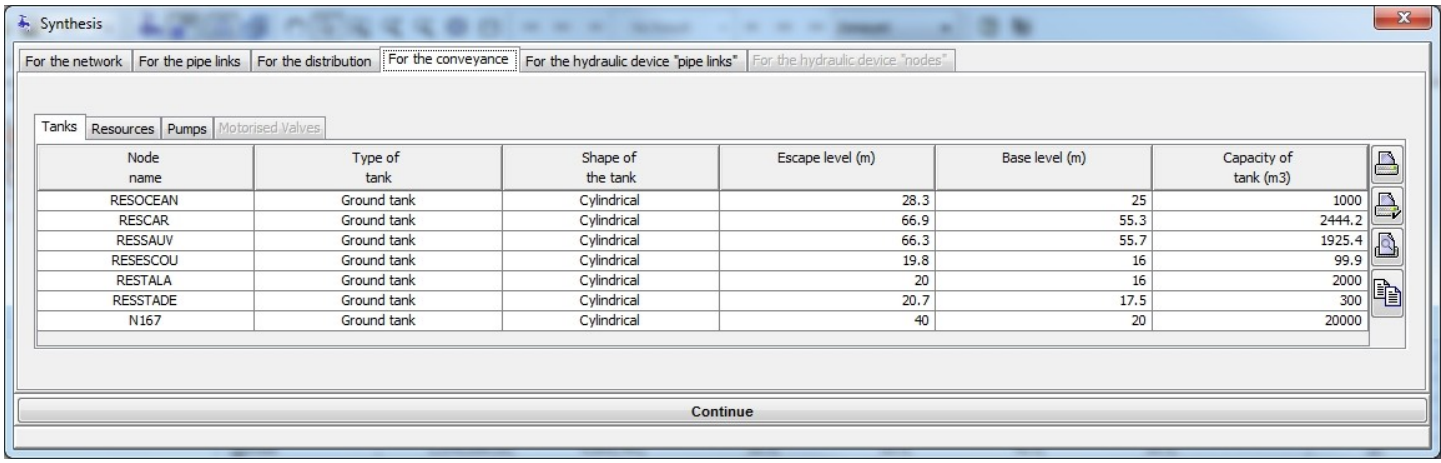
This table gives information on the number of nodes and pipe segments allocated to each service described, and for each family of consumption (domestic, industrial, exchange with other networks, localised leakage) the volumes of water consumed during the period of simulation (24 hours by default). The last line of the table highlighted in cyan shows the total for each column. In the absence of a service reference for the nodes and pipes, “without service” is displayed.



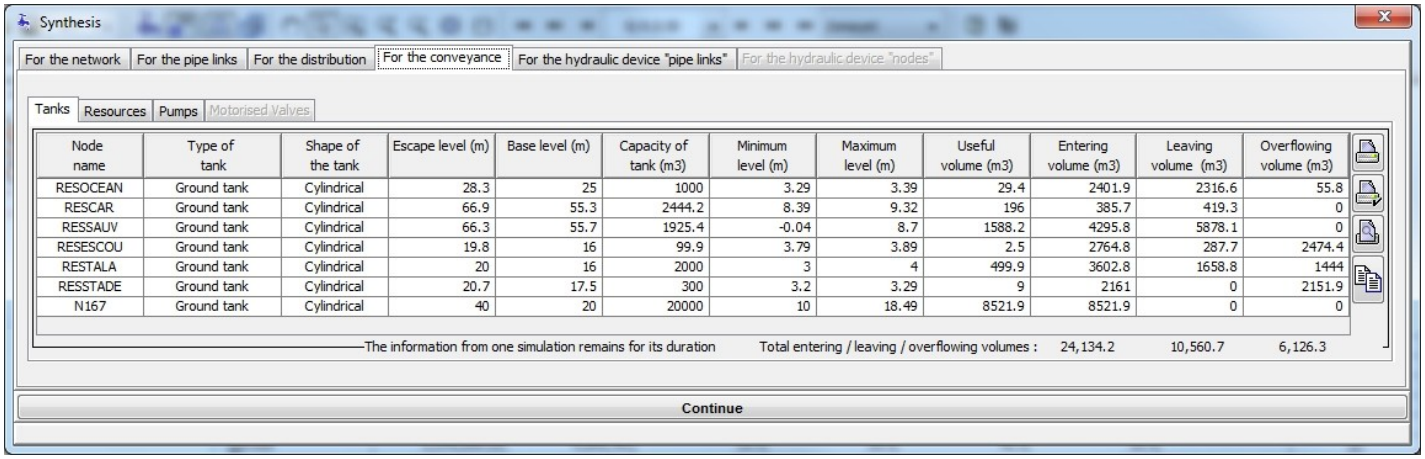
Synthesis on supply

This synthesis provides information across 4 tabs relating to tanks, reservoirs, pumps and motorised valves.

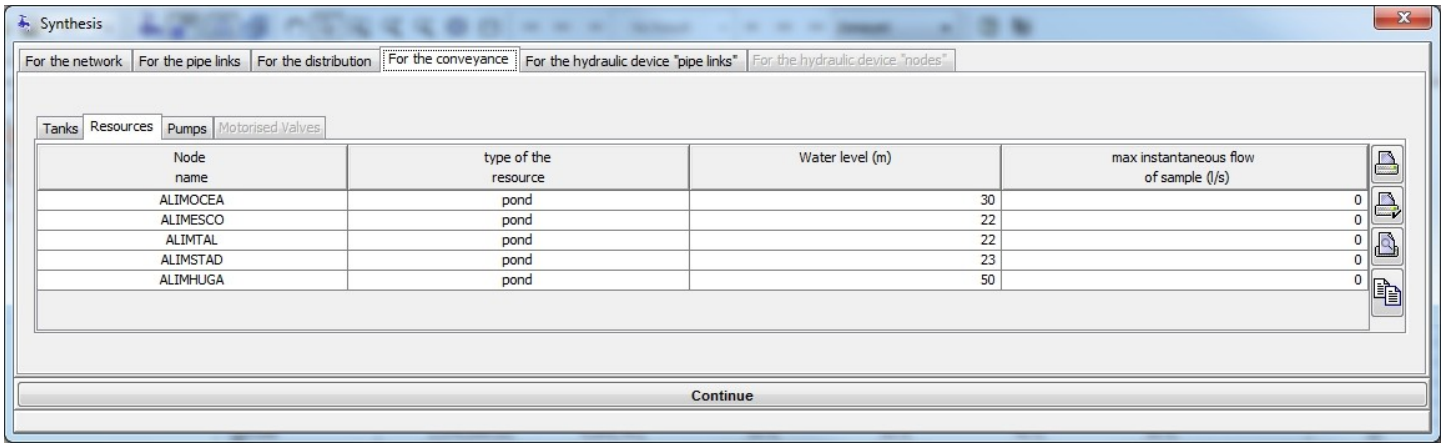
- Tanks



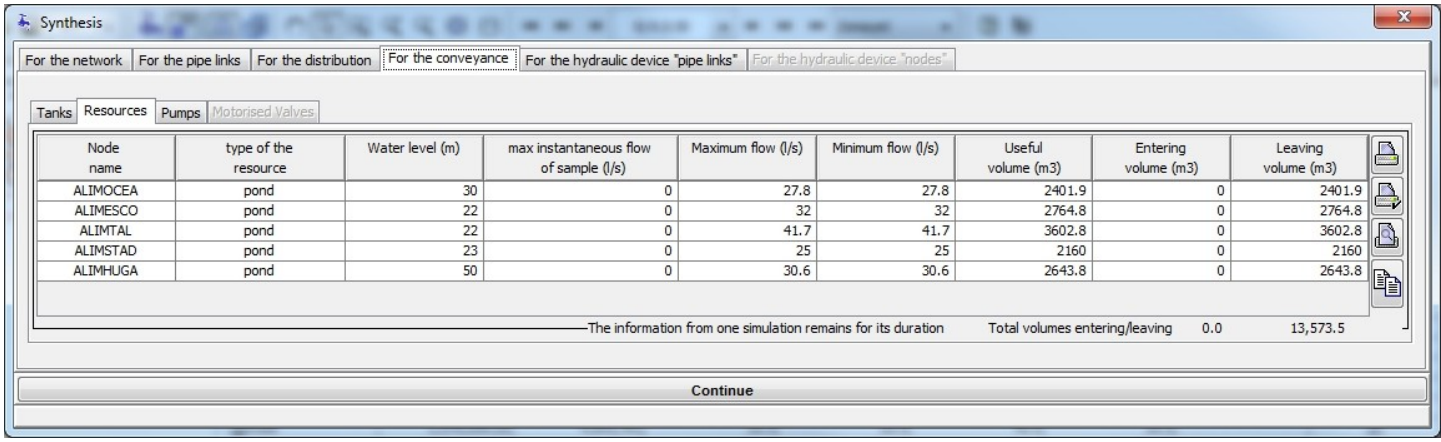
In the presence of Zomayet calculation results, several columns are added concerning these results with the totals at the bottom of the screen:



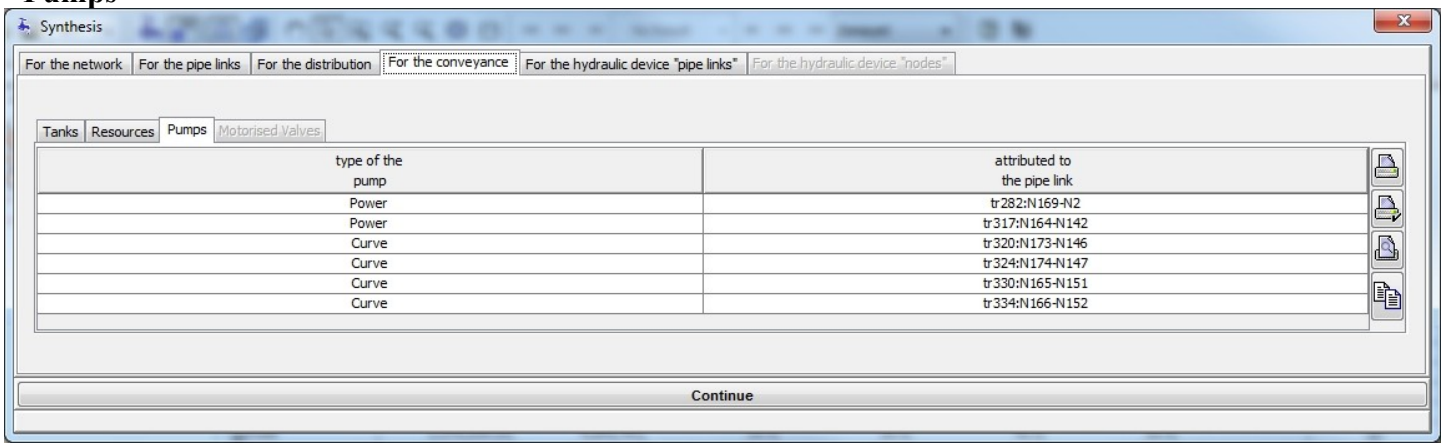
- Reservoirs



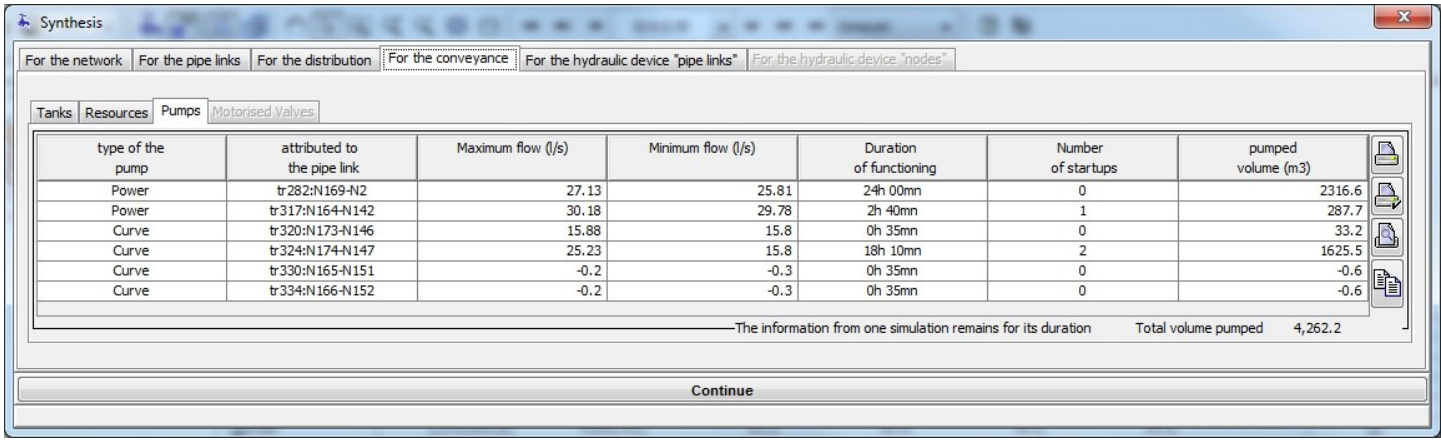
In the presence of Zomayet calculation results, several columns are added concerning these results with the totals at the bottom of the screen:



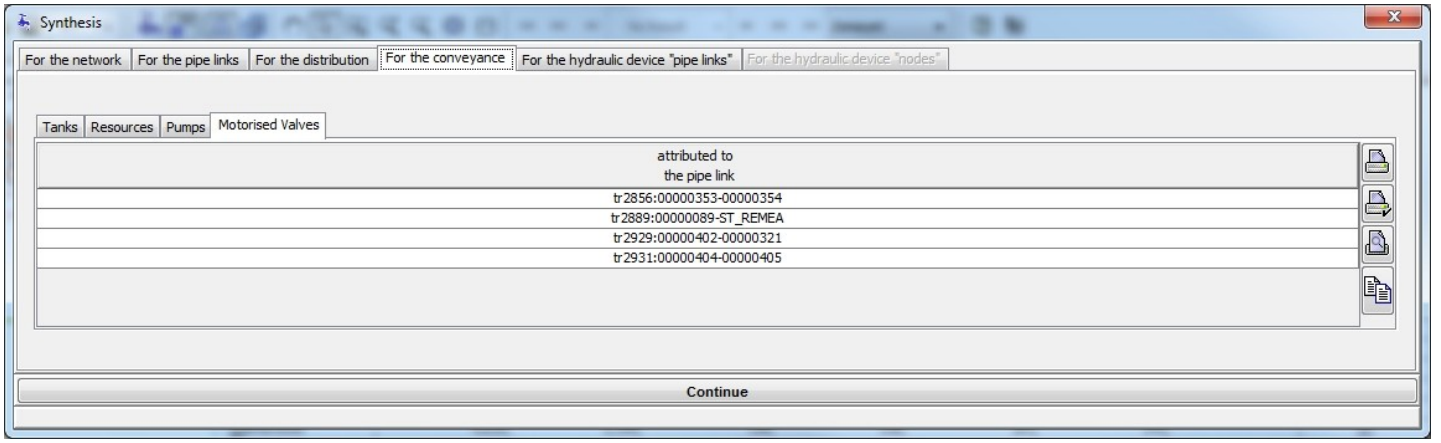
- Pumps



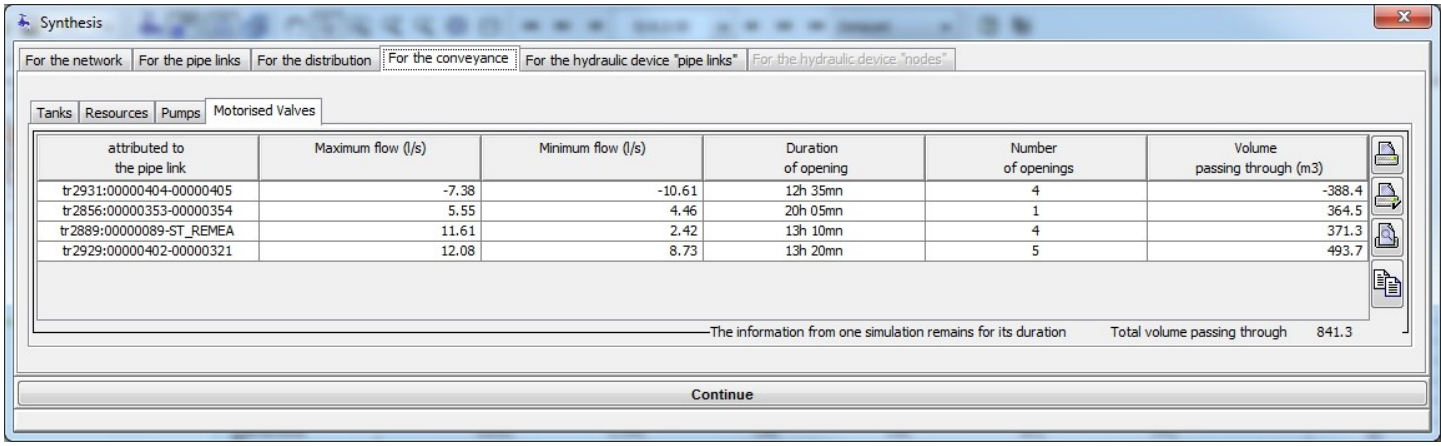
In the presence of Zomayet calculation results, several columns are added concerning these results with the totals at the bottom of the screen:



- Motorised valves



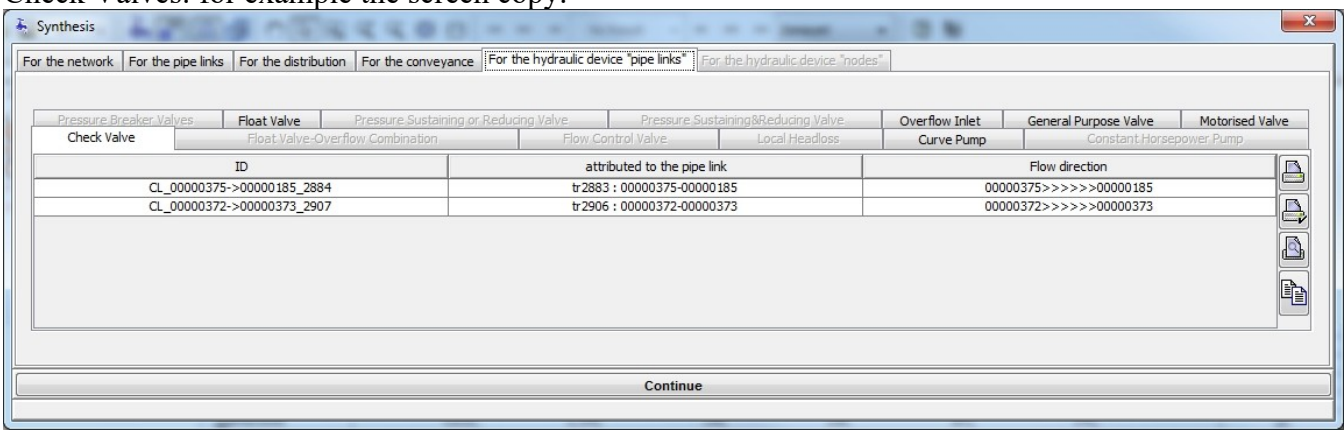
In the presence of Zomayet calculation results, several columns are added concerning these results with the totals at the bottom of the screen:



Synthesis for pipe link hydraulic devices

For each type of device, a tab makes a list of them.

- Check Valves: for example the screen copy.

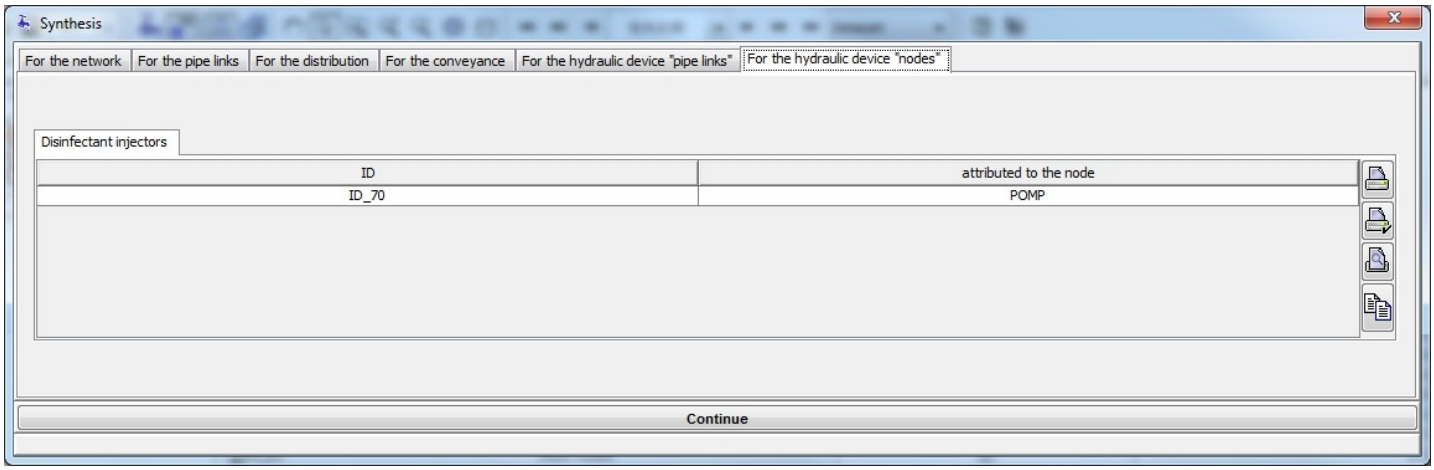


- Float Valve-Overflow Combination
- Flow Control Valve
- Local Headloss
- Curve Pumps
- Constant Horsepower Pumps
- Pressure Breaker Valves
- Float Valve
- Pressure Valves (Sustaining or Reducing)
- Overflow Inlets
- General Purpose Valves
- Motorised Valves

Synthesis for node devices

As the device of pipe, a tab makes a list for each device node type.

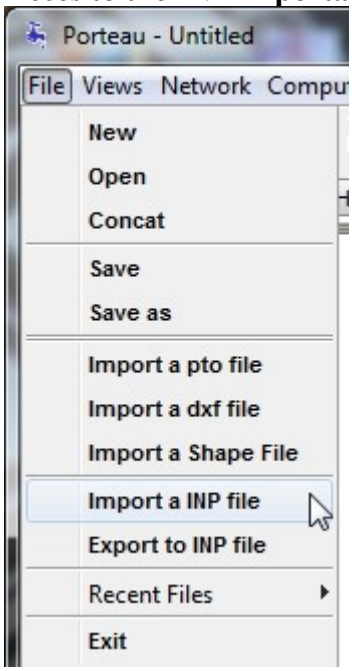
Disinfectant controllers



Data import

INP import

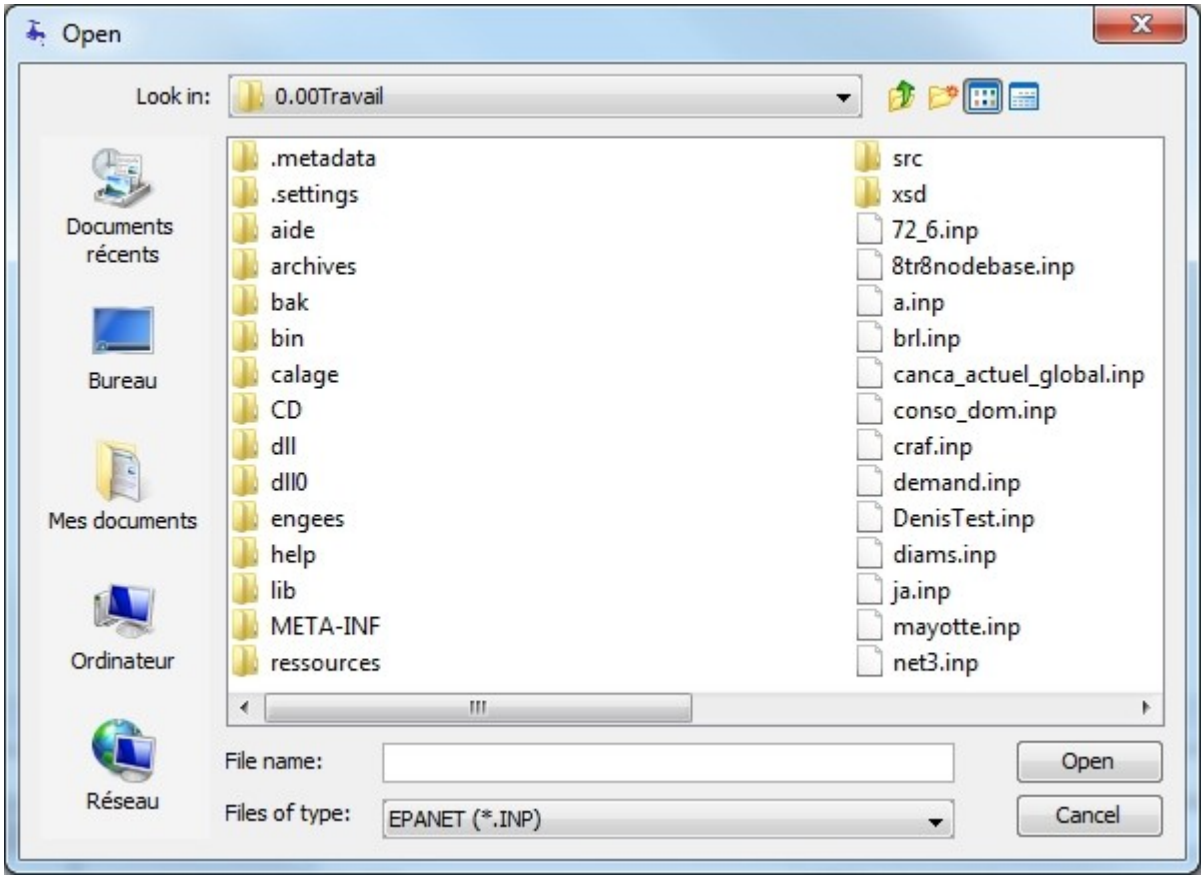
Acces to the INP importation menu



This dialog box is accessible by selecting the menu “File”, sub menu “Import an INP file”.

The dialog box appears for the choice of file to treat.

Choice of INP file



Choose the file in Epanet INP format to convert.

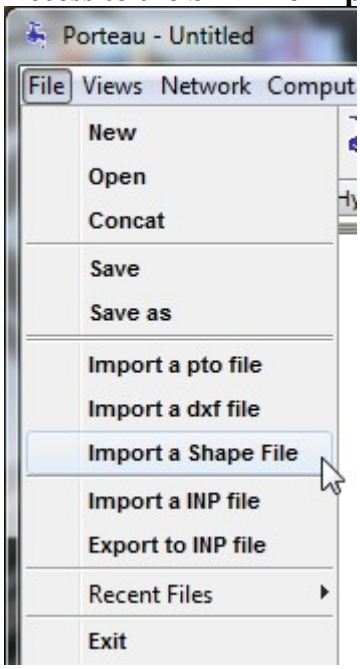
Conversion

After validation, the file is read and converted line by line.
Firstly the file is read to convert the OPTIONS, TIMES and BACKDROP sections.
Only the variables compatible with Porteau are transferred.
The background layer is not converted, it must be retreated by the **"select"** and **"resize"** background map dialog boxes.
The PATTERN and CURVE sections are read to store the names of "patterns" and "curves" referenced in other sections.
The file is then read again from the beginning.
The TITLE section is converted into three lines for the title, the network and the general description.
The JUNCTIONS section is converted as an ordinary node. If consumptions are stored on the nodes without reference to a pattern model, a new consumption model of industrial type is created with a constant graph through the duration of the patterns and named "Epanet Default Demand Model". The unit must be **LPS**.
The TANKS and RESERVOIRS sections are converted into Tanks or Resources according to the data.
The PIPES section is converted into pipe segments that are able to carry a closed valve - a check valve. Each new value of combinations of diameter and roughness creates a new pipe type in the project catalogue associated to the named material "INP Import". Following the law of head loss used, the roughness coefficient is converted from Hazen-Williams to Colebrook or vice versa so that all the fields are correct in the pipe types and pipe segments. The conversion is made by simple equivalence between the law of Hazen-Williams and that of Swamee-Jain for a velocity of 0.5 m/s.
The PUMP section is converted into a pipe with 1m of length and carrying a pump.
The VALVE section is converted into a pipe with 1m of length and carrying the device.
The PRV type is translated as a Pressure Reducing Valve, constant throughout the simulation.
The PSV type is translated as Pressure Sustaining Valve, constant throughout the simulation.
The FCV type is translated as a Flow Control Valve, constant throughout the simulation.
The TCV type is translated as a Local HeadLoss.
The GPV type is translated as a valve with head loss curve a function of flow.
The PATTERN section is converted into a function of time profile with a value for each time step.

A pattern that is not used is not imported.
The CURVE section is converted into profiles of different types following their use.
A curve that is not used is not imported.
The DEMANDS section is converted into consumption at ordinary nodes. Warning, if a first consumer is stored in the JUNCTIONS section on the node, it is replaced by the first value in this section.
The COORDS section is converted into schematic and map coordinates for all the nodes.
The VERTICES section is converted into intermediate points on the pipe sections.
The TAGS section is converted into sectors for nodes and into material for sections and pipes.
All other sections are not converted.

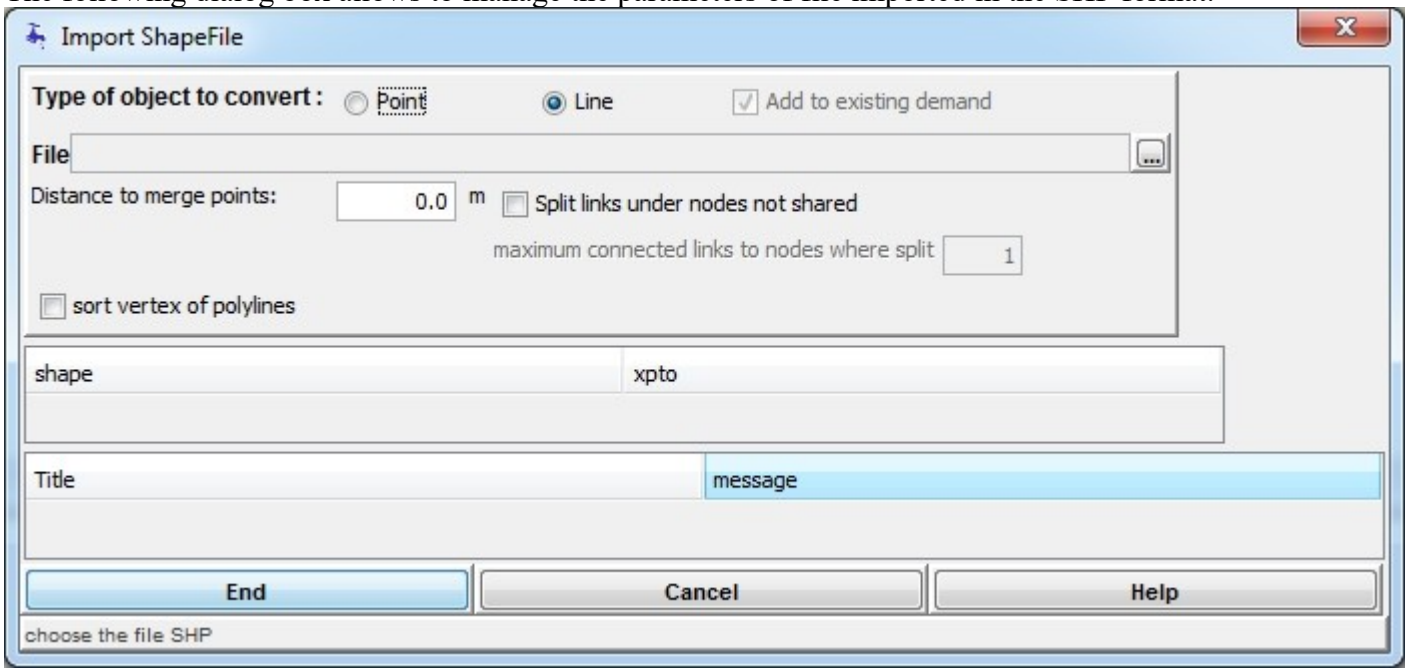
SHP import

Access to the SHP file importation menu



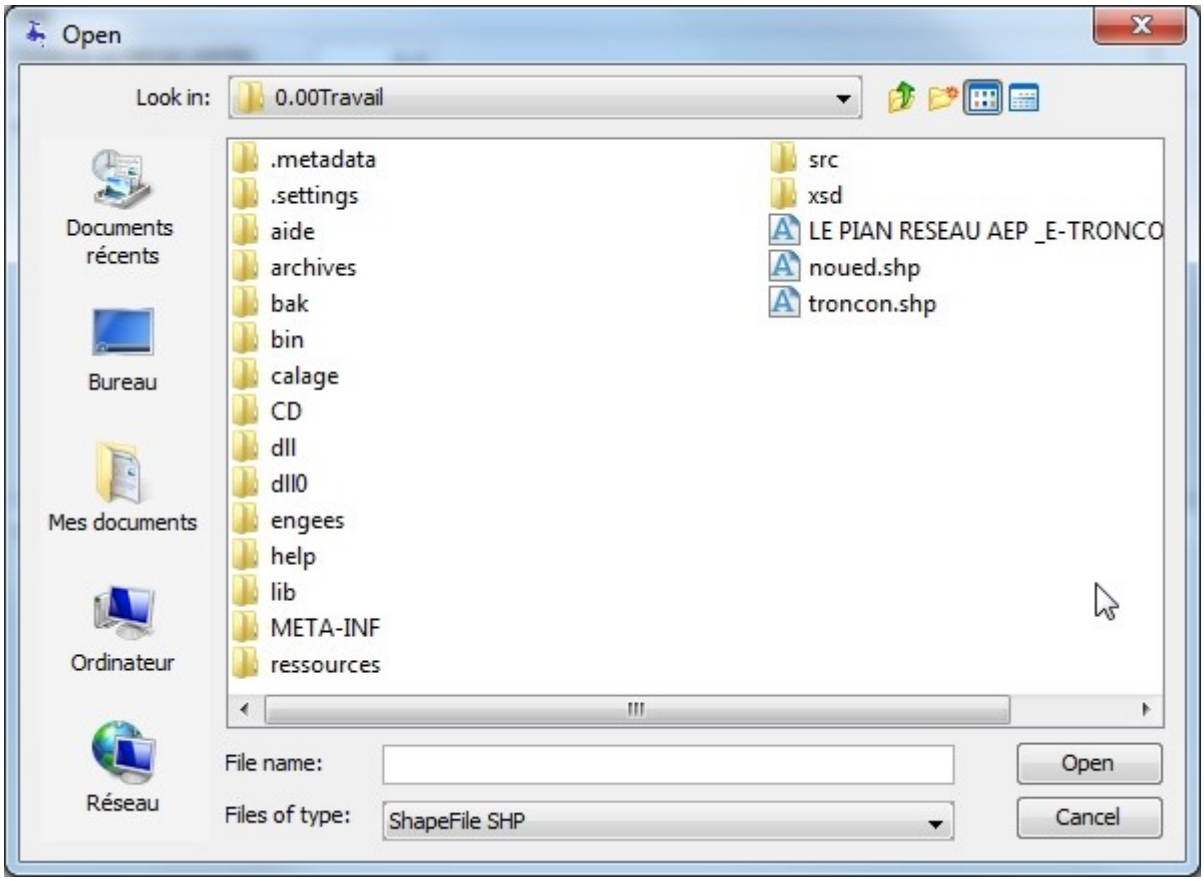
This dialog box is accessible by selecting the “File” menu, “Import a Shape File” sub menu.

The following dialog box allows to manage the parameters of file imported in the SHP format.



Clicking the button, access the choice of the file to import.

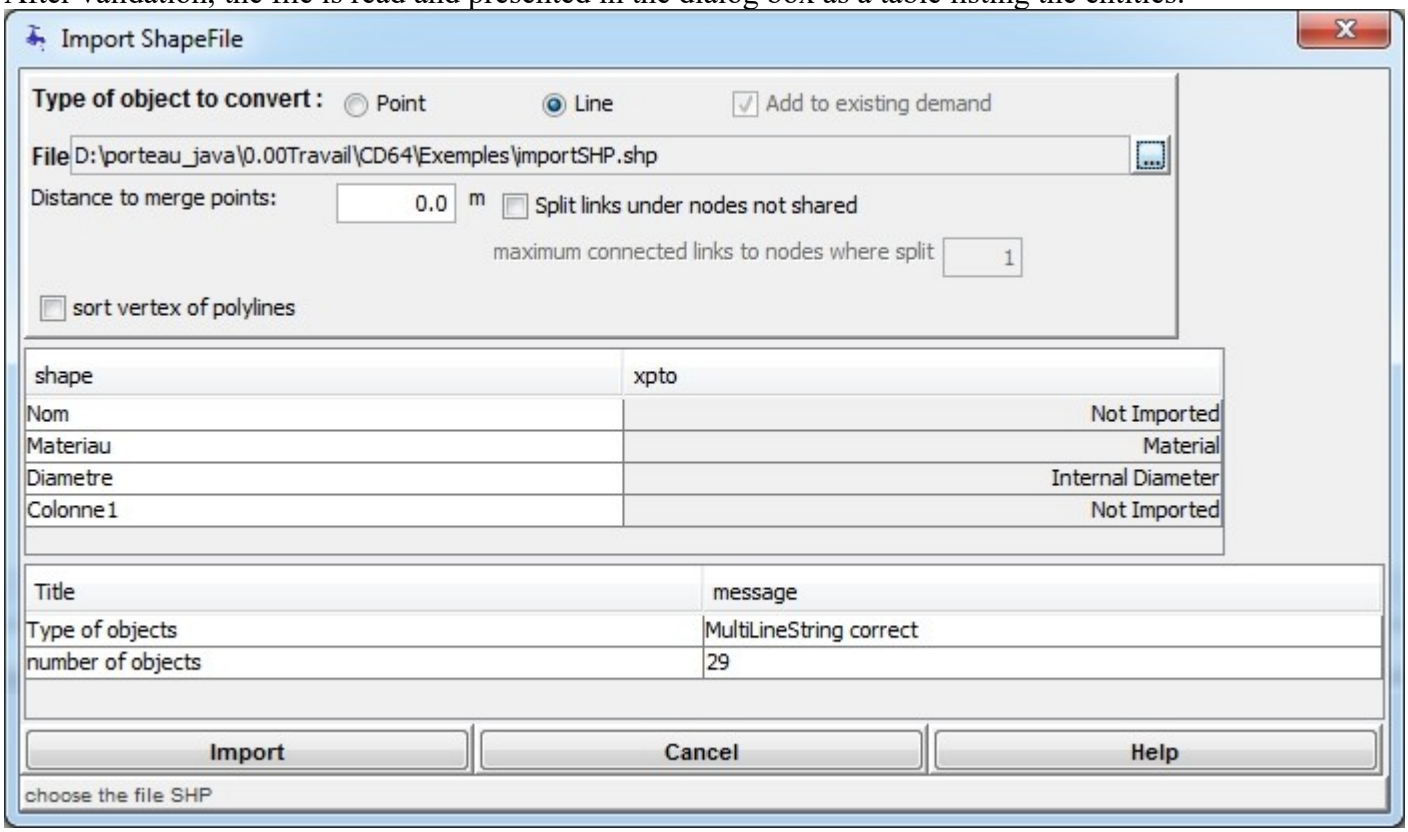
Choice of SHP file



Choose the file in the compatible SHP format of ARCGIS to convert.

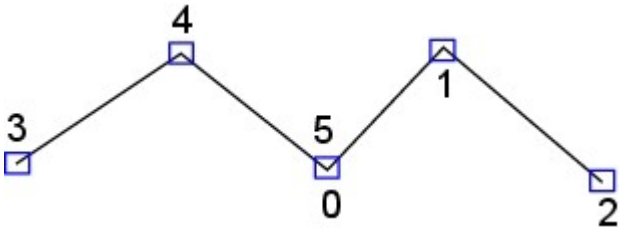
Conversion options

After validation, the file is read and presented in the dialog box as a table listing the entities.

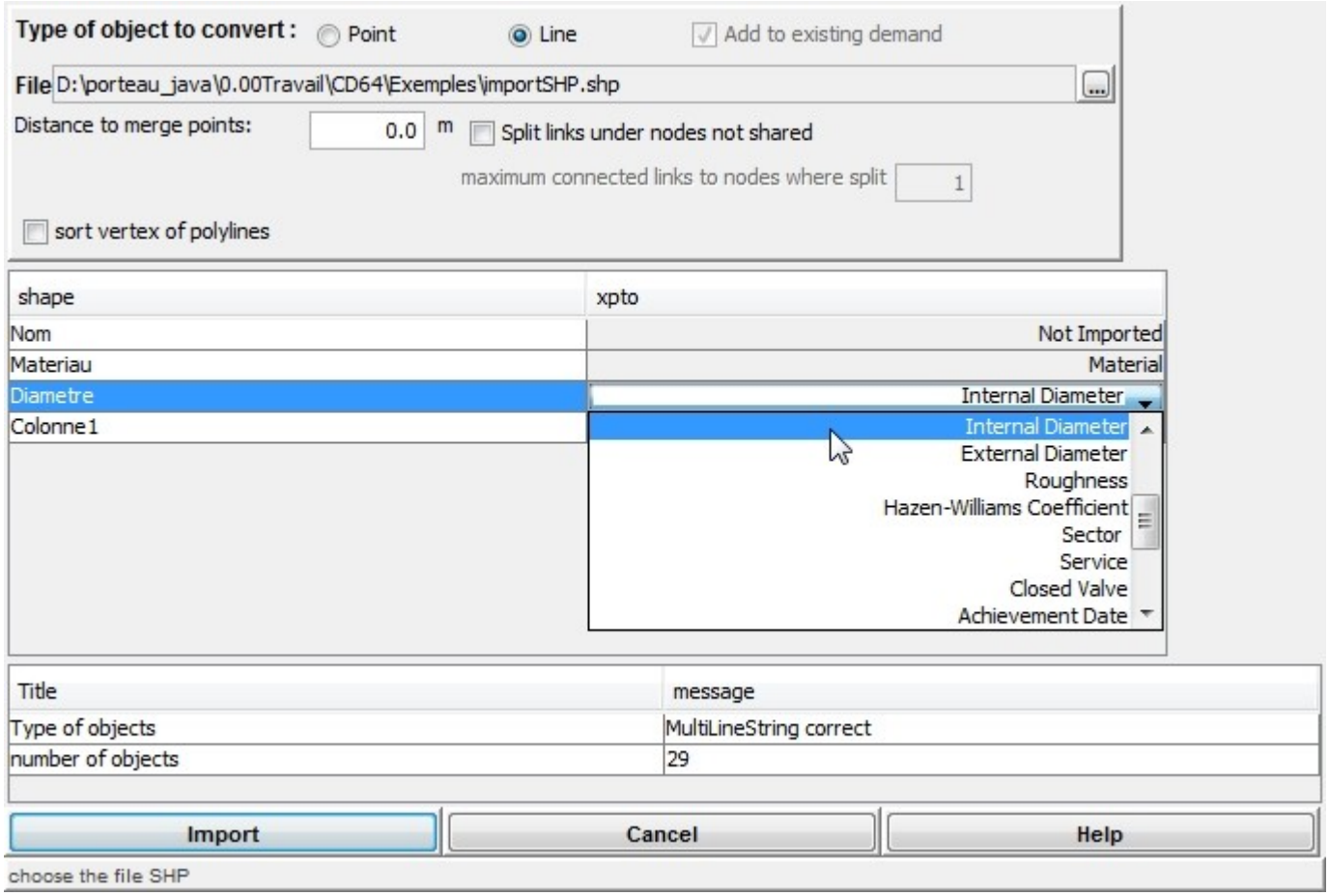


The choice between both radio buttons **Point** or **Line** allows to select the type of object to be imported of

the file. This choice depends on the processing of the information towards pipe sections for “Line” or towards nodes for “Point”. In the choice “Line”, if pipe sections were designed without hooking on properly ends to each other, sometimes by letting small lenghts between points as a rule mixed up, it is possible to merge their nodes. It reaches on the **distance of fusion**: if distance = 0.1 m then all the nodes close to 0.1 m at the most will be confused in a single node.
In case of "fusion" of two nodes, a line is added in the table of following messages (log), this table can be completely selected (by click on one line and ctrl+A), copy to the clipboard, and paste in a spreadsheet. This is usefull to create a shape of points where Porteau changes data to merge the nodes, by export coordinates in a csv file and create a shape in a GIS, to modify the original imported shape of the network. In the choice “Point”, if a node is already present in coordinates (X, Y) at the **distance of fusion** near the imported point, the previous node is taken to allocate it the new attributes. In the choice “Point”, the checkbox **Add to the existing consumption** allows to choose and treat the attributes either a new consumption (if the model is already present on the node, we clear the quantity), or an addition to the existing consuption (if the model is already present on the node, we add the new value to the existing quantity).
In the choice “Line”, the checkbox **Cut the pipe sections under nodes not shared** allows to share pipe sections for which end we observe another section orthogonal distance less than the distance of fusion a segment of the section, this appears especially when an extremity is entered without connection with a start node of a pipe section but with an intermediate point of the polyline.
The checkbox "sort vertex of polylines" allows to import polylines, sometimes results of merging lines in GIS not correctly sorted to be used for hydraulics, the next picture illustrates this problem with the rank of vertex.



It is then necessary to choose in the table the **attributes** to be imported by selecting in the xpto column the attribute to be allocated: a scroll list allows the choice of assignment. The same attribute can match only to a single field of the file SHP (DBF).

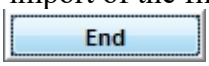


The available fields are:

- in LINE mode:
  - Not imported: the field is not transferred
  - ROOTNAME: root of name of nodes (prefix), string which is added at the beginning of the name of the node, thhis can be usefull to give a prefix name to a class of node (sector, city).
  - NOMDEBUT: name of the start top node
  - NOMFIN: name of the end top node
  - NOM: name of the pipe section
  - COMMENTAIRE : comment of the pipe section
  - MATERIAU: material of the pipe section, corresponds to the material of the water pipe allocated to the pipe section, if it does not exist in the project, it is added; Be careful, to be processed, a diameter must be also converted.
  - DINT: internal diameter in mm, allocated to the water pipe and in local diameter of the pipe section
  - DEXT: outside diameter in mm, allocated to the water pipe
  - RUGOSITE: roughness in mm, allocated to the water pipe and in local roughness of the pipe section
  - CHW: coefficient of Hazen-Williams, allocated to the water pipe and in local chw of the pipe section
  - SECTEUR: name of the sector, if the sector is not present in the project, it is added, ending nodes automaticaly inherit this attribute
  - SERVICE: name of the service, if the service is not present in the project, it is added, ending nodes automaticaly inherit this attribute
  - VF : if value is "closed", the pipe section will carry a closed valve
  - ANNEPOSE: achievement date of the pipe section

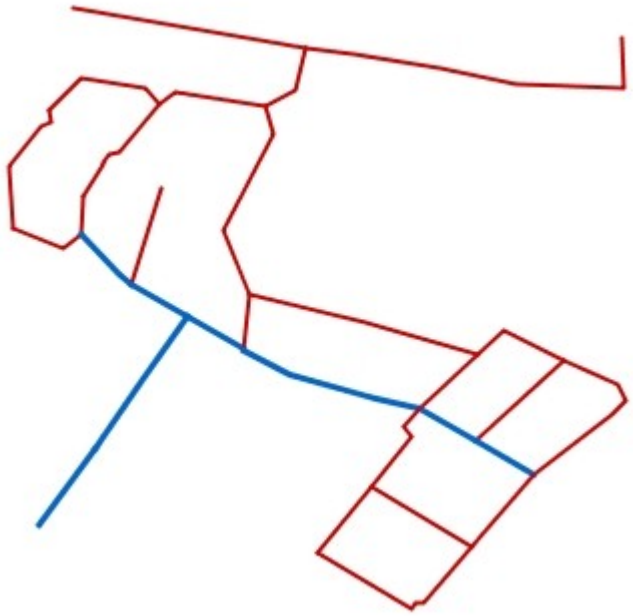
    - MODELE1: model of consumption for allocating a new consumer, a quantity must be also converted. If the model is already present in the network, it is taken for model of the quantity to be allocated. Idem for 2 and 3
    - VALEUR1: quantity allocated to the consumer of the model 1. Idem for 2 and 3.
- in POINT mode:
    - Not imported: the field is not transferred
    - TYPE : type of node, if not present, the node is odinary, otherwise the node can take the following types : JUNCTION, RESSOURCE, RESERVE
    - NOM : name of the node, if it is not present, the name is automatically generated
    - Z : ground level
    - BAS : low user level
    - HAUT : high user level
    - DESIREE : required piezometric level
    - SECTEUR : name of the sector, if the sector is not present in the project, it is added
    - SERVICE : name of the service, if the service is not present in the project, it is added
    - COMMENTAIRE : comment
    - MODELE1 : model of consumption for allocating a new consumer, a quantity must also be converted. If the model is already present in the network, it is taken for model to the quantity to allocated. Idem for 2 and 3
    - VALEUR1 : quantity allocated to the consumer of the model 1. Idem for 2 and 3

Validate the options and import the file by clicking on 

or cancel the import.
After the import of the file, the button changes aspect.
Click on 

once all the files SHP imported.

Example of SHP import
The “importSHP.SHP” file



A network containing polylines with the material as attribute of color:

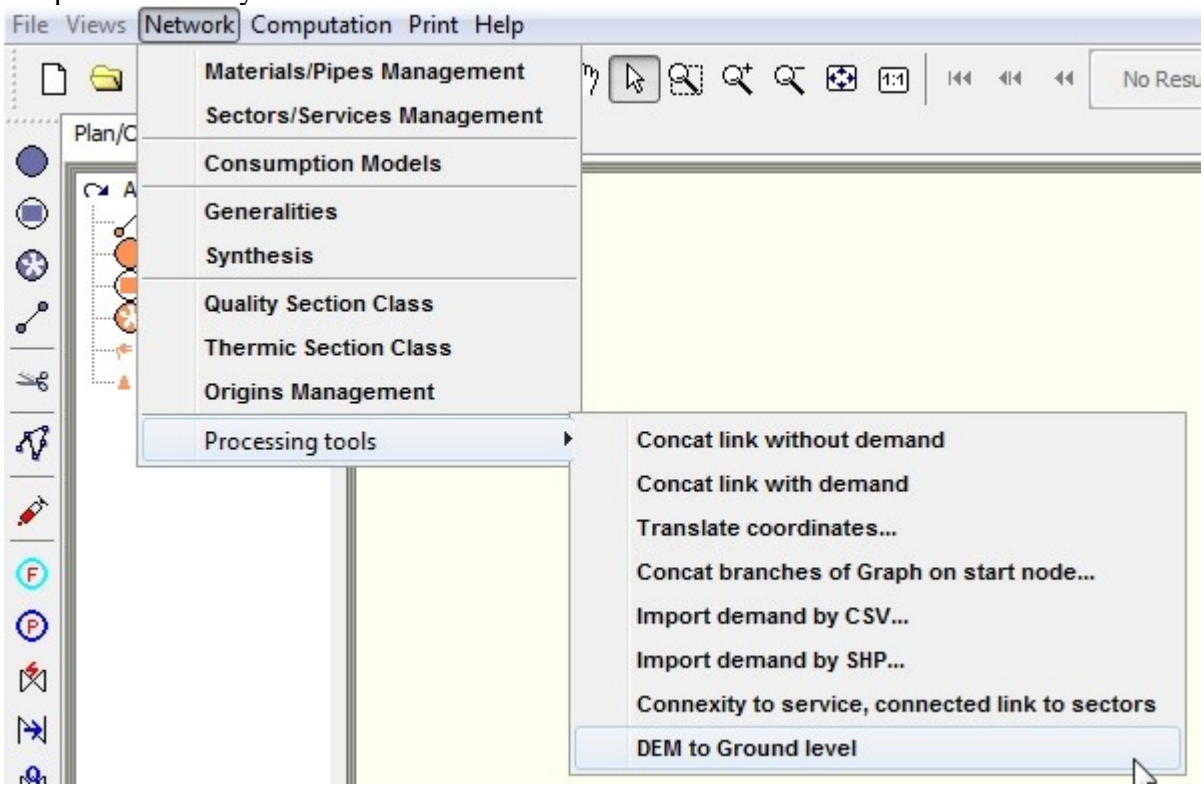
- in red the PVC.
- in blue the Cast Iron.

Pipe sections also carry an attribute of diameter in meters (to show that it is imported such as) and a sector or a service in the column1 attribute. At the end of SHP file processing, the table of message is

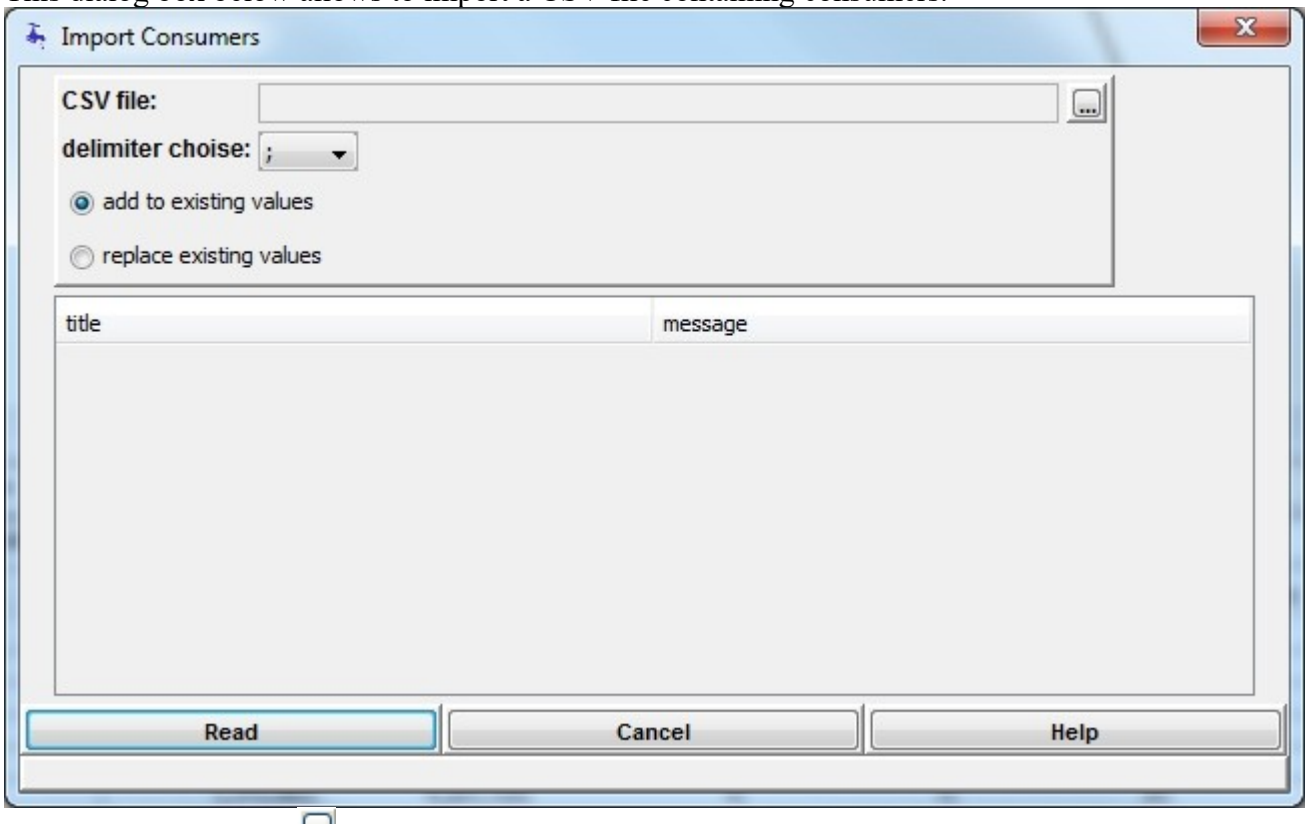
filled, it can be exported towards clipboard by Ctrl+A and Ctrl+C. According to the quality of SHP file data, several messages can specify anomalies not allowing the import, in this case, the not corresponding recordings are not treated, as a polyligne having the same start node and end node, an already existing polyline between two nodes, etc. The final number of handled object and the linear are displayed.

Import of consumptions in CSV

This dialog box is accessible by selecting the menu “Network”, sub menu “Processing tools”, sub menu “Import demand by CSV...”.



This dialog box below allows to import a CSV file containing consumers.



By clicking the button  , we reach the choice of the file to be imported.

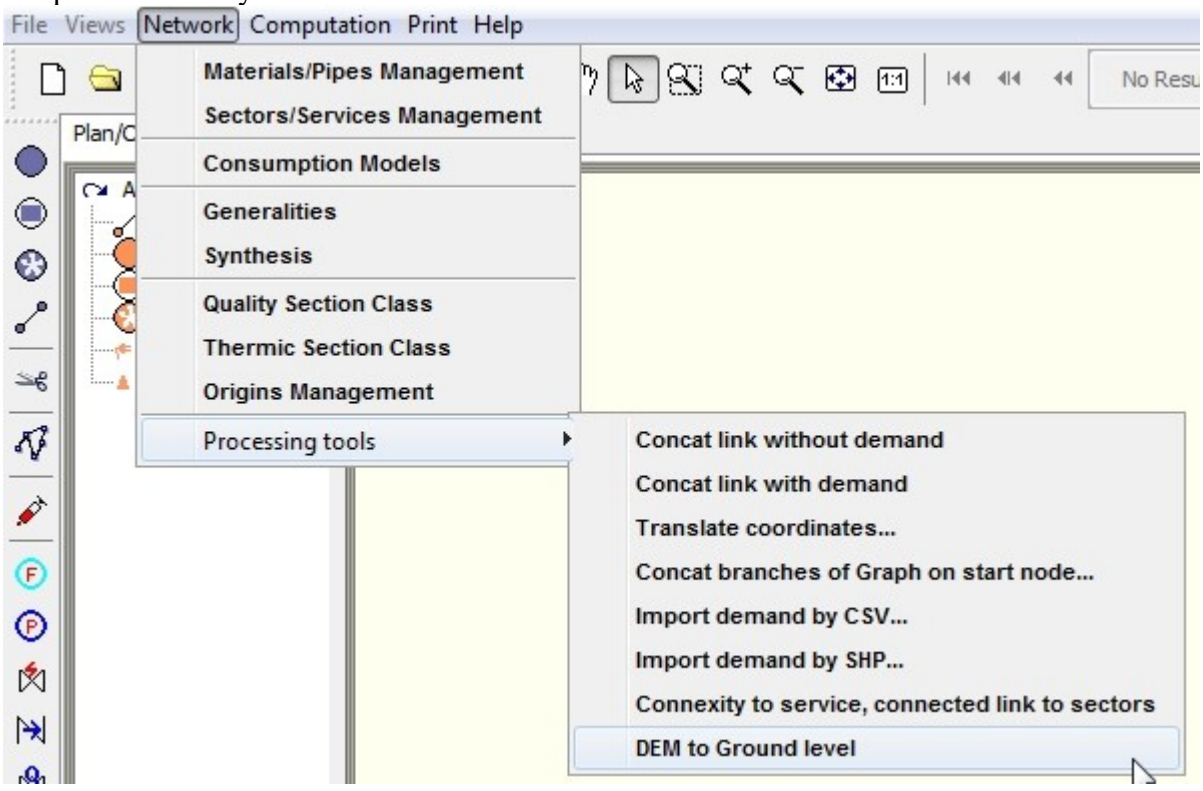
The file NECESSARILY has to contain four columns without headers:
1 – the name of the ordinary node or both extremities of the pipe section
2 – the type NODE for node or LINK for pipe section
3 – the name of the model already existing in the project
4 – the quantity to be imported

Example of rows:
VILA; NODE; Domestic; 151
VILA-VILB; LINK; Domestic; 15

Choice between the buttons:
replace existing values : Allows to replace the consumers of the model met in the CSV file so already present on the node or the pipe section,
add to existing values : Allows to add new consumptions to the already present consumption; be careful, if two lines contain the same node (or pipe section), the same model, both quantities are added in the import.

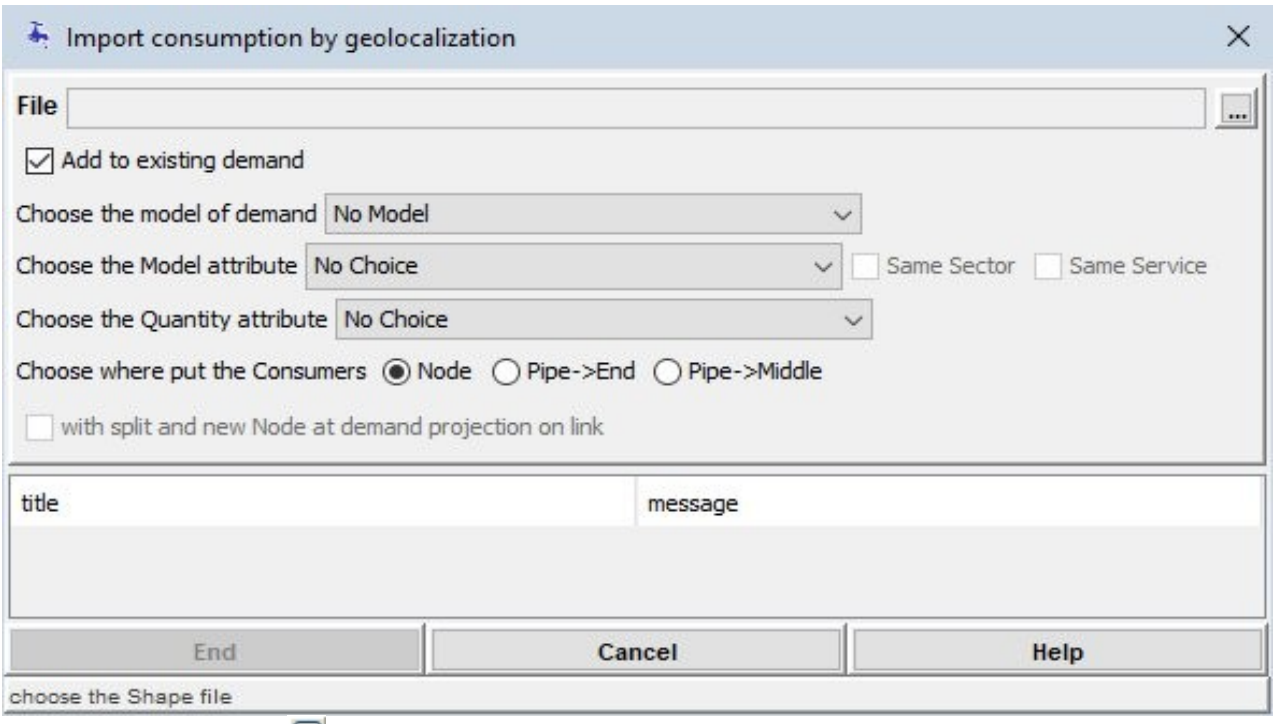
Import of consumptions in SHP

This dialog box is accessible by selecting the menu “Network”, sub menu “Processing tools”, sub menu “Import demand by SHP...”.



This dialog box below allows to import a SHP file containing points representing connections of the network or the location of the consumptions.
Three methods of projection can be chosen:

- The point is projected on the nearest ordinary node, the carried quantity is allocated to the node with the chosen model;
- the point is orthogonally projected on the segment of the nearest pipe section, the quantity is half allocated to extremity nodes if they can carry a consumption, to the only node otherwise;
- the point is orthogonally projected on the segment of the nearest pipe section, the quantity is allocated to the distributed demand all along the pipe section.



By clicking the button , we reach the choice of the file to be imported.
The checkbox **Add to existing demand** allows either to erase an already present quantity on the chosen model, or to add a new quantity by setting up a consumer with the model chosen below. By clicking the scrolling list No Model , we choose the model of consumption to which we allocate the quantities given by the addresses. A model must necessarily be present in the network to be able to continue the processing following this method.

By clicking the Choice of Model attribut scrolling list No Choice , we choose the attribute of the SHP file containing the model of consumption to be used if it is present in the network, or to create otherwise. This choice is required if the previous scrolling list is on “no Model”.

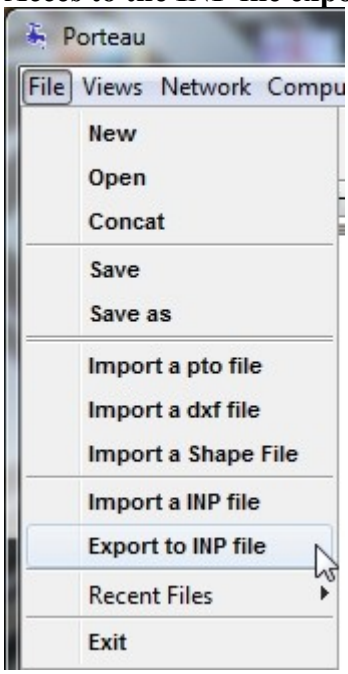
By clicking the Choice of Quantity attribut scrolling list No Choice , we choose the attribute of the SHP file containing the quantity to be processed and to be allocated to the model chosen by one of the two lists above.

By clicking one of the radio buttons Node Pipe->End Pipe->Middle , we choose one of three methods of allocation of the consumption.
In case of choice of projection of the distributed demand, the checkbox with split and new Node at demand projection on link allows to cut the pipe section in two parts and to create a new node to assign the consumption, this is particularly interesting for big consumers for whom we want to follow the pressure of service or any other parameter.
After clicking "Import", Porteau offers saving consumers import and their projection as a CSV file. The file content is: "#SHP_DEMAND_LINKS";LinkName;BeginName;EndName;IdDemand
SHP_DEMAND_LINKS;144;od932;od929;PDL_Secto.1
SHP_DEMAND_LINKS;144;od932;od929;PDL_Secto.2
The first line gives the column headers. The first column contains the type of projection, here as the distributed demand all along the pipe sections. SHP_DEMAND_NODE describes a projection on the nodes, SHP_DEMAND_LINKENDS a projection on the pipe sections with distribution to end-nodes to half. If several imports are made with the same logfile, the first import reappears for each new import in this file, defining then the phases of the import. The fields "LinkName", "BeginName" and "EndName" give the name of objects on which the projection is made. "IdDemand" gives the same ID of object of the SHP file. "DemandFactor" gives the part to the mode for the projection by the pipe section by end-node (each node receives 0.5).

Data Export

INP Export

Acces to the INP file exportation menu



This dialog box is accessible by selecting the menu “File”, sub menu “Export an INP file”.

The dialog box appears for the choice of file to write.

Conversion
All temporal profiles are converted by recreating, if needed, a value for each time step from the Zomayet calculation, this by interpolation conforms to the type of profile (LINE, STEP, .).
The TITLE section is made up of the fields containing into "Generalities" tab the following information: project, network, description and authors.
The JUNCTIONS section contains the ordinary nodes and eventually their consumption if it is unique.
The RESERVOIRS section contains the resource nodes.
The TANKS section contains the tank nodes. If the shape isn't cylindrical, a volume curve is created as a function of height. Warning, difference can remain taking into account the different model used for this type of tank.
The PIPES section contains the pipe segments. Those carrying a pump or closed valve are translated as a closed pipe with a line in the PUMP section repeating some of those informations.
The PATTERNS section contains the profiles for the consumption models and those of the nodes if needed.
The CURVES section contains the profiles of tank nodes and pumps.
The COORDINATES section contains the cartographic coordinates of the nodes.
The VERTICES section contains the cartographic coordinates of the intermediate points of the pipe segments.
The BACKDROP section contains the references to the background map and the coordinates for the positioning of the image.
The TIMES section contains the duration of the simulation, the time step and the time of the start of the simulation.
The duration and the time step are used for the creation of all patterns exported.
The OPTIONS section contains the units (LPS) and the law of head loss.

Export en SHP

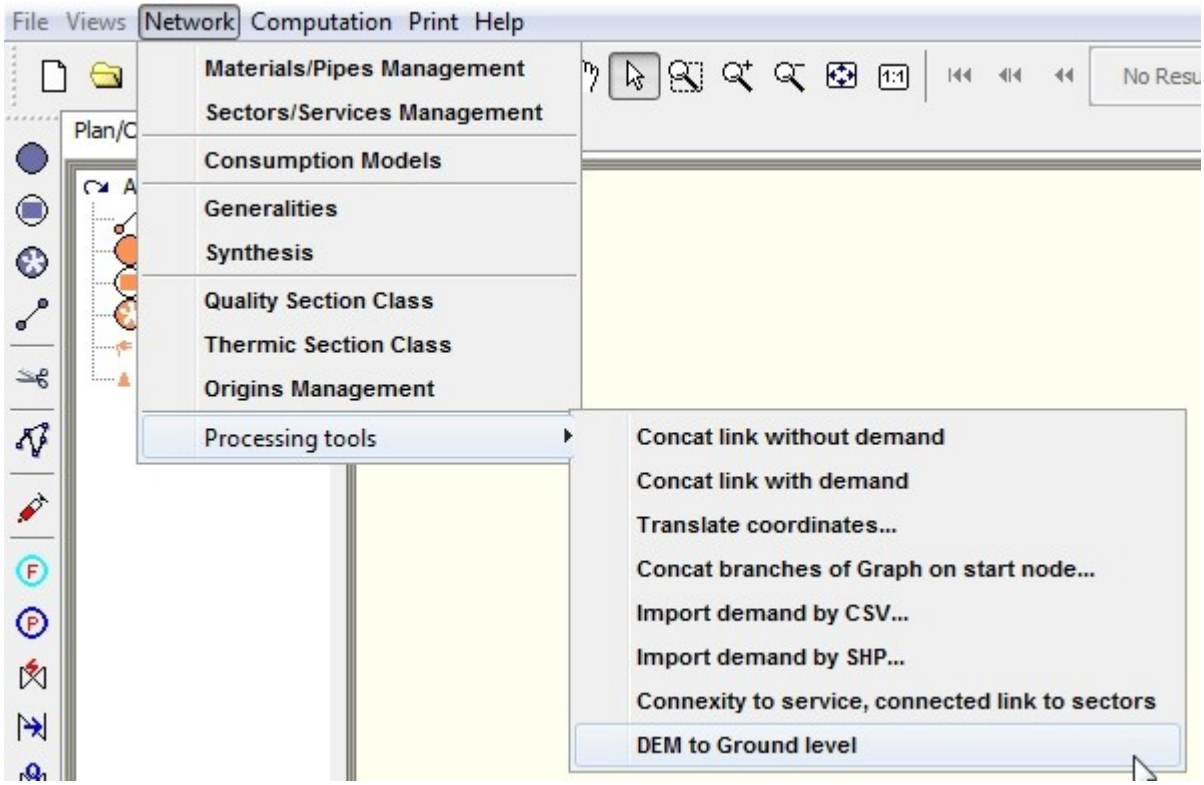
This function is accessible by the “File” menu, sub menu “Export to SHP file”. The dialogue box for choosing the file appears to enter the name of the file to be written. **Attention:** several files with the same name are written, only the extensions are different (Shp, shx, prj, dbf, qix, fix). If results of simulation are present, they are written in the CSV format in a file of extension csv and the name of which is completed by Results A unique identifier (ID) allows to connect the objects of the geographical layer to a line of the table csv. The exported attributes are:

- for nodes: ID, Name, Type (ordinary, reserve, resource), Ground (ground level), Service, Sector, Com (comment).
- for results of nodes according their presence in memory: ID, Name,
    - OPressure : the Opointe pressure
    - ZP_jjhhmmss : the Zomayet pressure à chaque pas de temps avec jj le jour de simulation, hh l'heure, mm les minutes et ss les secondes,
    - C_jjhhmmss : la concentration pour Qualité à chaque pas sauvegardé, même formatage du temps que Zomayet
    - A_jjhhmmss : l'âge moyen pour Qualité à chaque pas sauvegardé, même formatage du temps que Zomayet
- pour les tronçons : ID, Begin (nom du noeud amont), End (nom du noeud aval), DLocal (diamètre local), KLocal (rugosité locale), HWLocal (hazen williams local), MatPipe (nom du matériau), DPipe (diamètre du tuyau), Service, Sector (nom du secteur), Com (commentaire).
- pour les résultats de tronçons en fonction de leur présence en mémoire : ID, Begin, End,
    - OFlow : le débit Opointe
    - ZF_jjhhmmss : le débit pour Zomayet à chaque pas de temps avec jj le jour de simulation, hh l'heure, mm les minutes et ss les secondes,

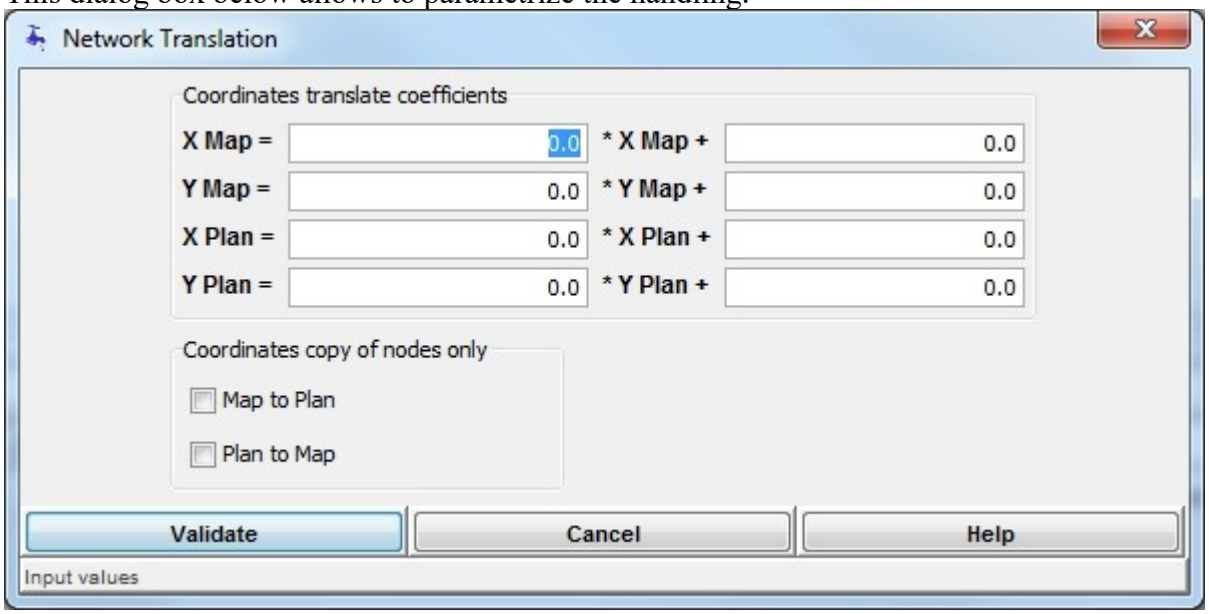
Processing of graph

Translating of coordinates

This dialog box is accessible by selecting the menu “Network”, sub menu “Processing tools”, sub menu “Translate coordinates...”.



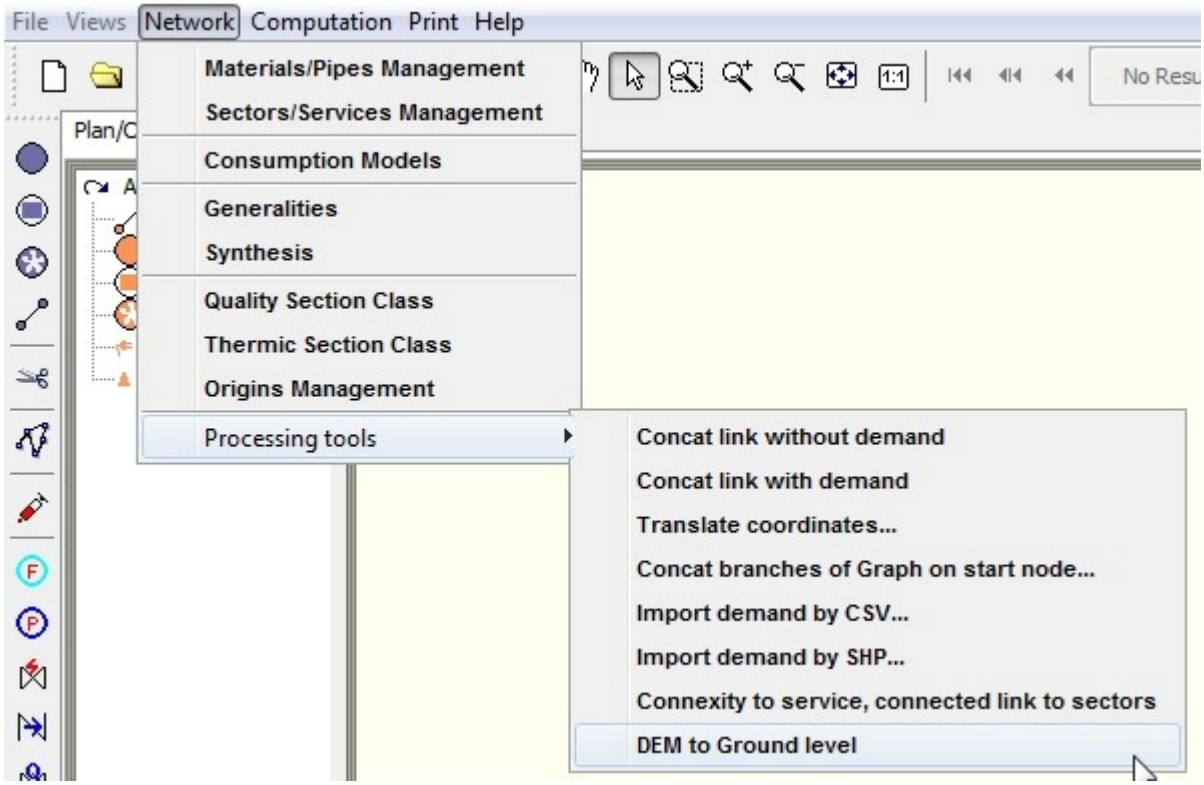
This dialog box below allows to parametrize the handling.



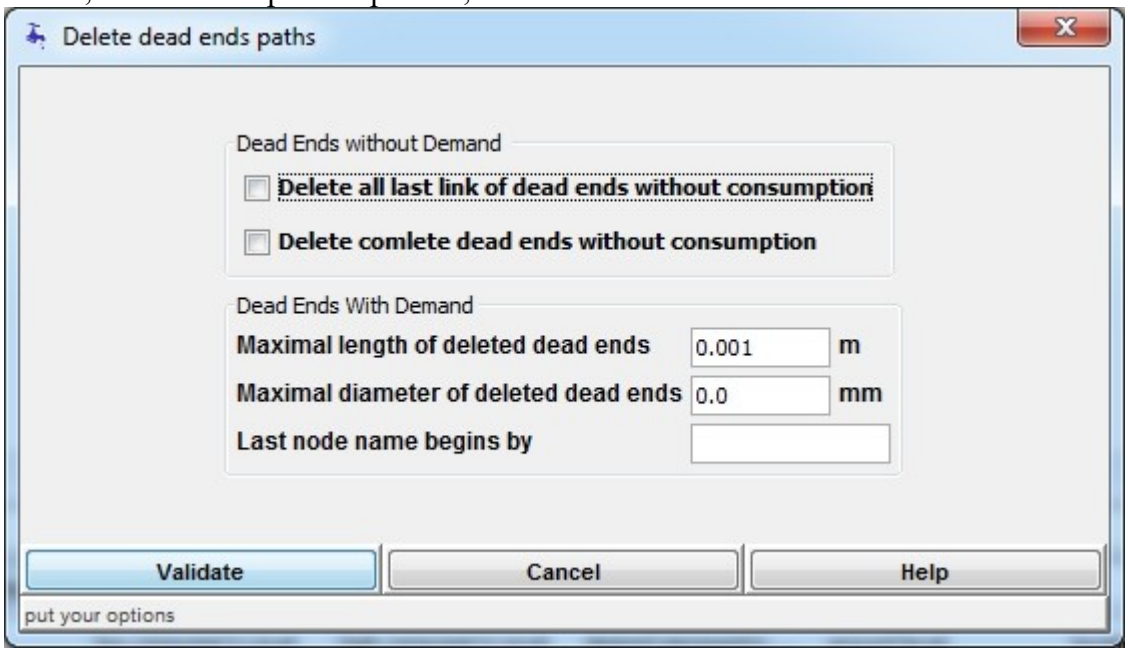
It is possible to work independently on both systems of coordinates.
Capture in the corresponding fields the values to transform the coordinates of nodes. The processing follows the linear formula for all the values: Xnew = a * Xprevious + b. In case of modification of map coordinates, the intermediate points of pipe sections are also changed. Checkboxes allow to inject a system of coordinates in the other one only for nodes; example for a file imported by Porteau 2 not containing map coordinates, copy of plan coordinates towards map coordinates to allow a display in map view, otherwise all the nodes will be in (0,0).

Concat dead ends

This dialog box is accessible by selecting the menu “Network”, sub menu “Processing tools”, sub menu “Concat branches of graph on start node...”.



This dialog box below allows to choose the parameters of the handling.
This one consists in locating all the dead ends, then to give the total consumption of the dead end to its "root", if no consumption is present, all is deleted.

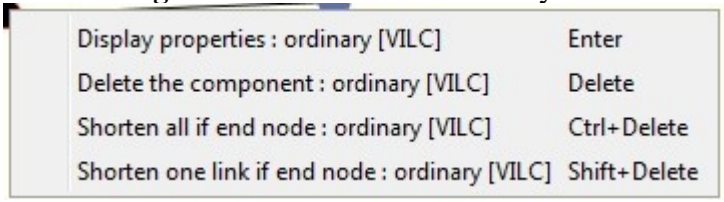


The removal of dead ends without consumption takes two options, which can be used independently of the options of removing with consumption.
The checkbox **Delete the last pipe section of the dead ends without consumption** allows not to include the length, if no node or no pipe section of the dead end carries consumer, and delete the last pipe section and the last node of the dead end.
The checkbox **Remove completely the dead ends without consumption** allows not to include the length, if no node or no pipe section of the dead end carries consumer. All the dead end is removed.
The removal of dead ends with consumption can be made according to three criteria that can be mixed:
The field **maximal length of deleted dead ends** allows to give the length from which a dead end is kept. The length is the total of the lengths of pipe sections composing the dead end. It can not be null.
The field **maximum diameter of deleted dead ends** allows to give the diameter of the largest pipe sections from which a dead end is kept.

The field **Name of the last node starts with** allows to remove only the dead ends for which the last node has a name beginning with the entered text, the name is then carried over to the start node of the dead end. Caution if a node is the start node for several dead ends, the met first name will be applied. This function can be used to bring the connections of fire hydrants on main pipes without taking the identifier of the hydrant (example : PI0000).
The validation of removal settings traces these actions in a CSV format, this choice of the name of this file is unique for a Porteau session, if cancel is chosen, no tracing will be done until the next call of the removal function.
In the console of Porteau, a balance sheet of deleted nodes and deleted pipe sections appears.
The logfile contains:

- to delete without consumption:
  "#DEADEND";LinkName;BeginName;EndName;ErasedNodeName;RootNodeName
  DEADEND;od6->od7;od6;od7;od6;od7
  DEADEND means a delete end of the dead end, followed by the name of the deleted pipe section, the name of its ends, the name of the deleted node and the name of the kept node.
- to delete on request of length, diametee, name of node:
  "#REQUESTED_DEADEND";LinkName_NodeName;BeginName;EndName;RootNodeName
  REQUESTED_DEADEND_LINK;od313->od314;od313;od314;Nd257
  REQUESTED_DEADEND_LINK;od316->od313;Nd257;od313;Nd257
  REQUESTED_DEADEND_NODE;od314;Nd257
  REQUESTED_DEADEND_NODE;od313;Nd257
  REQUESTED_DEADEND means there is a request by criteria, followed by either the name of the pipe section or the name of the deleted node, the corresponding names and the root node where consumption is returned. Here a dead end has two pipe sections that are erased and the two nodes and the root Nd257 receives the accumulated consumption to that possibly already on that node.

It is possible to reduce a simple dead end.
A double **right** click on the node extremity of the dead end shows the context menu.



Following the chosen line, either "shorten all if end node" or "shorten one link if end node", in every case, the consumptions are relocated towards the root of the dead end or towards the node becoming the bottom of dead end.

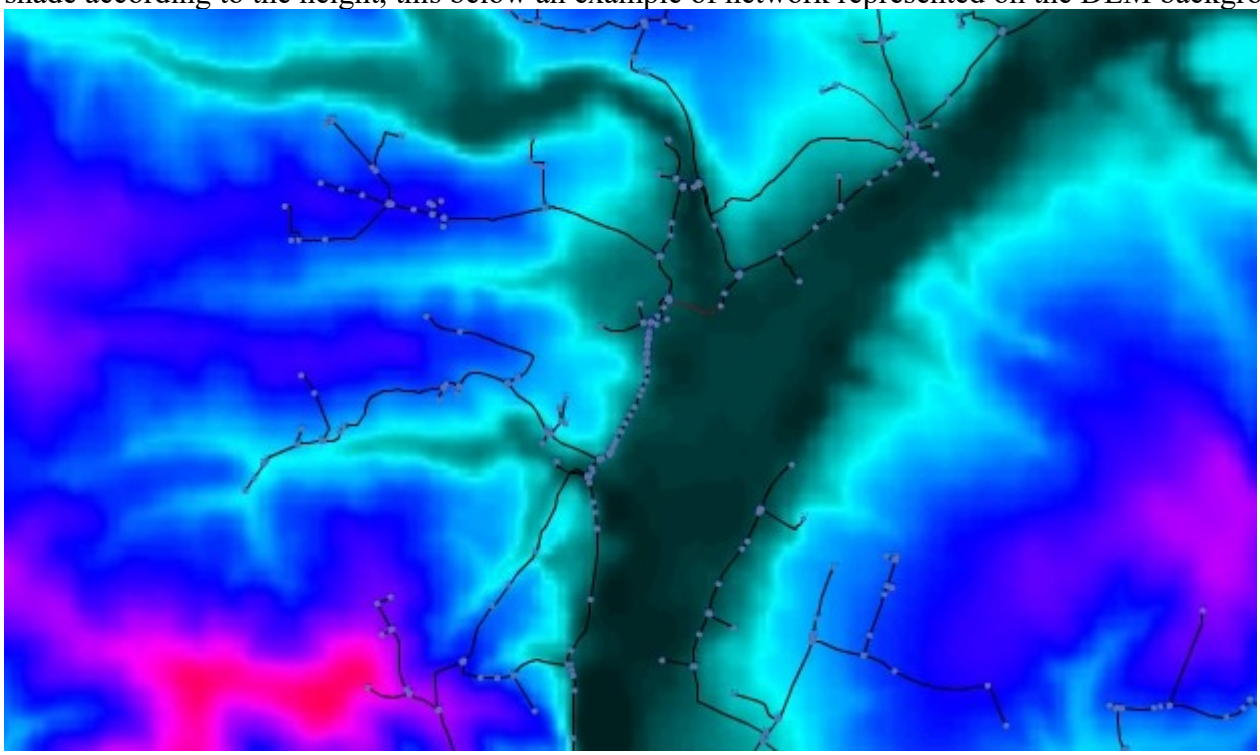
DEM to ground level

This function is accessible by selecting the menu “Network”, sub menu “Processing tools”, sub menu “DEM to ground level”.
The dialog box of the choice of the file containing the DEM allows to read a file packaging the meshing of same coordinates as the level of nodes to calculate. The formats being able to be read are:

- ASC: Ascii Grid ARC/INFO or Arcgrid, containing in text the data
- TIF or TIFF: GeoTiff
- IMG: Erdas Imagine

If a selection of node is active, only the levels of the selected nodes are calculated, otherwise all the ground levels are calculated and erased.
The file is read by the Geotools library and the GDAL plug-in,the nodes are projected on the meshing,

then the value is interpolated by a bicubique spline method. After processing of a DEM in image with a shade according to the height, this below an example of network represented on the DEM background .



Reproject the network

This function is accessible by selecting the menu “Network”, sub menu “Processing tools”, sub menu “Change the network projection”.
The dialog box of the choice of the EPSG Codes of projection system allows to change the projection (GIS) of the network, nodes and links, for using the network in a new system. The formats being able to be read are:

- Origin EPSG Code: contains the code in EPSG format of the actual projection of the network (example 3857)
- destination EPSG Code: contains the code in EPSG format of the desired projection of the network (example 2154)

In case of error in the value of code or if the two codes are the equals, the state bar displays a message.