GRIF 2022

Petro

User Manual

Version May 6th, 2022

Abstract

This document is the user manual of *Petro* module of *GRIF 2022*

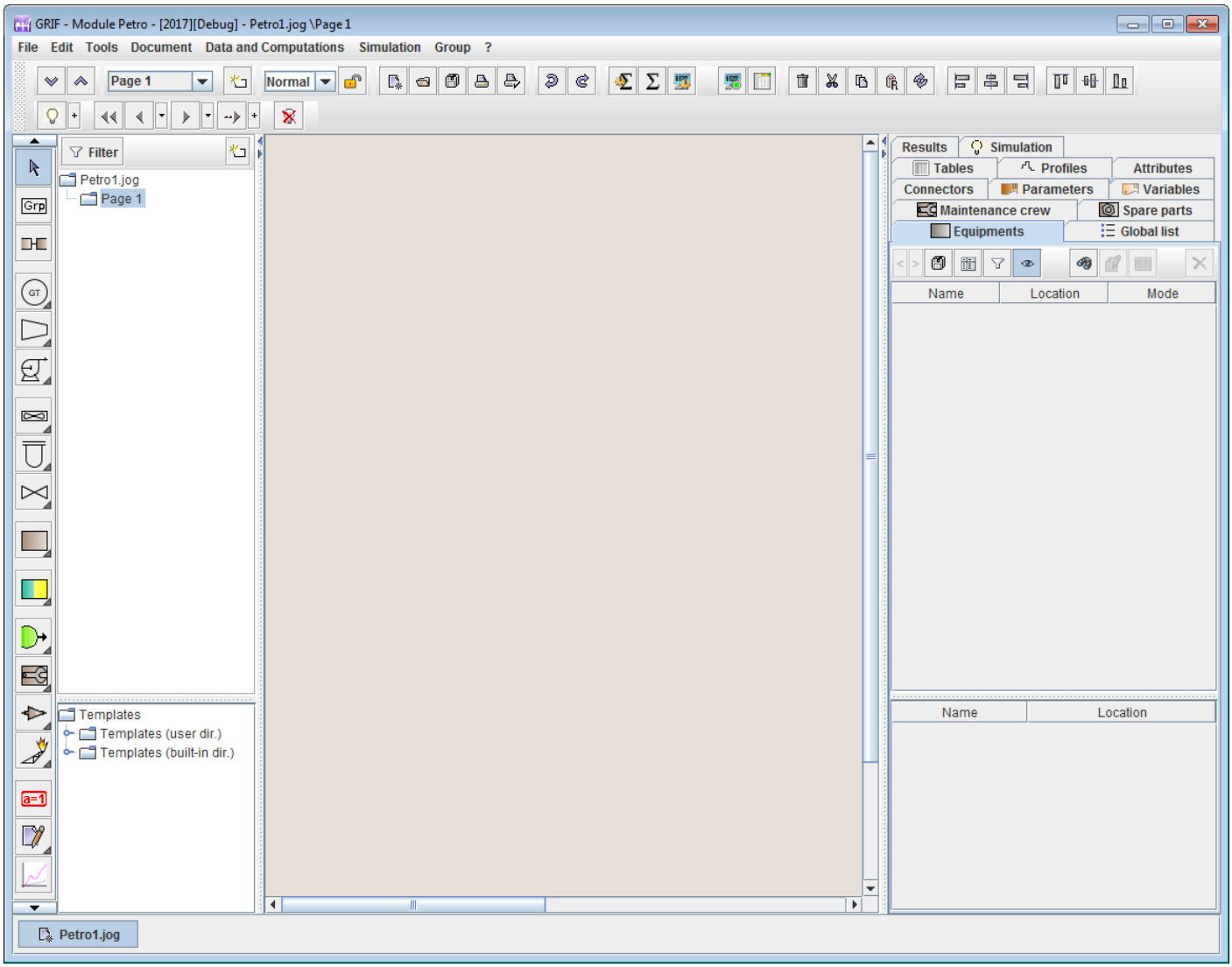
Table of Contents

1. Description of the interface

1.1. Main window of the Petro module

The main window is divided into several parts:

- **Title bar**: The title bar shows the names of the module and file being edited.
- **Menu bar**: The menu bar gives access to all the application's functions.
- **Icon bar (shortcuts)**: The shortcut bar is an icon bar (horizontal) which gives faster access to the most common functions.
- **Tool bar**: The tool bar (vertical) enables you to select the elements for modeling.
- **Input zone**: A maximum amount of space has been left for the graphical input zone for creating the model.
- **Tree**: A tree is between input zone and tool bar. It enables to walk through pages and groups of the document.
- **Template**: Templates are hidden behind the tree. They are grouped in two files following the save directory (User or built-in directory).
- **Set of tables**: Tables are gathered in "hidden" tabs on the right.



1.2. Vertical toolbar

Each model used in operating dependability has its own icons. All the graphical symbols for stochastic flow diagrams are shown on the vertical icon bar on the left of the data input screen.

The vertical toolbar contains the following items:

	Selection tool Select the graphical objects in the input zone.
	**Group** Add a group (sub-page) to the model.
	Link tools: link two components/connectors.
	Rotating machine blocks: all of the components representing rotating machines.
	Static machine blocks: all of the components representing static machines.
	Tank blocks: used to manage a production buffer.
	Generic block: used to define a block which has its own Petri net.
	Generic resources: used to define its own resources.
	Generic rotating machine block: defines a rotating machine whose Petri net is based on the initial model, but to which the user wishes to add functions.
	Generic static machine block: defines a static machine whose Petri net is based on the initial model, but to which the user wishes to add functions.
	Source: defines a flow source for the model.
	Target: defines an output for the model.
	Profile block: defines a new profile for the flows.
	Ramp-up block: defines progressive ramp-up of all or part of an installation.
	Utility connection tools: defines a flow output, to become the source of a utility network.
	Maintenance crew block: defines a maintenance team. This tool cannot be connected to the network through graphical links.
	Spare part block: defines a stock of spare parts. This tool cannot be connected to the network through graphical links.
	Common cause failure block: defines a common cause failure. This tool cannot be connected to the network through graphical links.
	Statistical block: defines random shutdowns. This tool cannot be connected to the network through graphical links.
	Timer: enables to activate a chronometer under condition.
	Connector: builds a convergent or divergent connector.  The connectors are automatically created when links are created, and this tool simply serves to explain the creation of a connector where necessary.

	Separator connector: distributes the flows among the branches in specific proportions.
	Input/output connectors: divides up the network so it can be spread over several pages. The output connector and its shortcut are created automatically.
	Flare output: defines a flare, a component that is able to burn any excess effluent it receives (e.g. gas).
	Flare connector: redirects the flows to a secondary route (the flare) if the main route is unavailable.
	**Dynamic field tool** creates dynamic comments displaying the data in the model.
	**Comment tool** creates static comments.
	**Line tool** creates lines or arrows of different styles.
	**Rectangle tool** creates rectangular zones of different colours. These zones can be anchored to the page.
	**Polygon tool** creates a zones with an polygon shape. These zones can be anchored to the page.
	**Circle tool** creates circular zones of different colours. These zones can be anchored to the page.
	**Ellipse tool** creates a zones with an ellipse shape. These zones can be anchored to the page.
	**Curve tool** draws curves by selecting the computation results in the results base.
	Simulation tools: enters the interactive simulation mode.

1.3. Data Editing Tables

1. Description of the Tables

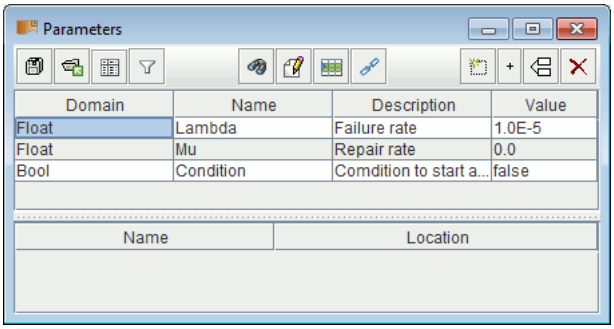
To create or modify data (parameters, variables, etc.), tables are available in the **Data and Computations menu** and in tabs at the right of the view. All the GRIF 2022 data tables operate in the same manner.

It is possible to edit all tables in another screen using **Data and Computations - Editing tables (new windows)** menu.

The data editing table/panel is divided into 3 parts:

- The upper part consists of a toolbar;
- The main part containing the data table.
- The bottom part indicating what the selected data is used for. This table is available only if the given data can be used by another data. The first column of this table indicates the name of these elements, the second indicates their location in the document (page, group). A click on a line from this lower table will open the page where the item is located and select it.

Here is an example illustrating the parameter table



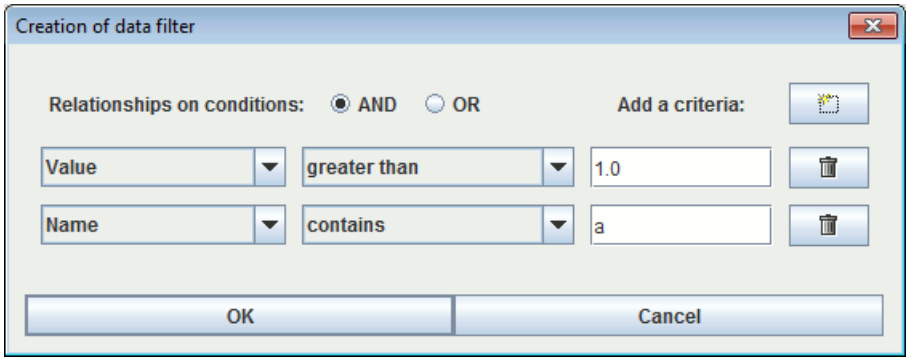
Different actions are available depending on the type of data displayed. Below is a non-exhaustive list of actions that can be found on the data tables.

	Saves the table in a text file.
	Import data from another Petro model or from CSV file.
	Opens the column manager (cf. Section 1.2, “Column manager”).
	Displays a panel for searching or filtering data (cf. Section 1.1, “Filter and sorting data”).
	When the **display selection** button is pressed, a click in the table leads to the selection in the input area.
	Find and/or replace expression in the table .
	Edit the selection.
	Multiple modifications made to all the selected data.
	Permit to merge data in a unique data.
	Creates new data.
+	Create the number of data indicated by user.
	Duplicate the selected data (ask a new name)
	Deletes the selected data (one or many).

1.1. Filter and sorting data

The filter panel enables you to display only what is necessary in the data table.

It consists of a search part: the text entered is searched in all the cells of the table, only the lines whose text is present are preserved; and an advanced filtering part allowing to consider finer criteria according to the different fields of the data. It is possible to combine several filtering criteria, as below:



Select **AND** or **OR** to choose the type of association between each line (filter criterion). A line is a Boolean expression divided into 3 parts:

1. the first is the column on which the filter is used;
2. the second is the comparator;
3. the third is the value to which the data will be compared.

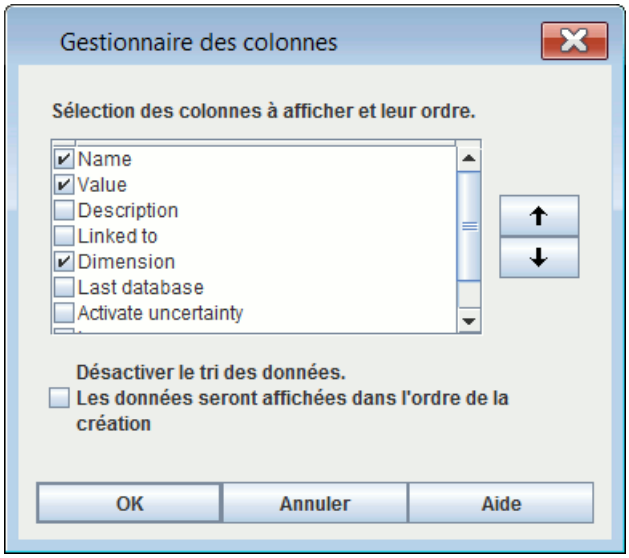
If the Boolean expression is true, the data will be kept (displayed); otherwise the data will be masked. When the filter is enabled its value is displayed between < and >.

The data in a column can be sorted by double clicking the header of this column. The first double click will sort the data in ascending order (small triangle pointing upwards). The second double click on the same header will sort the column in descending order (small triangle pointing downwards).

The choices that are made are kept on the current document. They will be reapplied when reopening your document and do not affect other documents in the application.

1.2. Column manager

A table can contain many columns and to improve its readability it is possible to choose the columns that will be displayed as well as their order. To do this, click on the **Columns Manager** button, the following window opens:

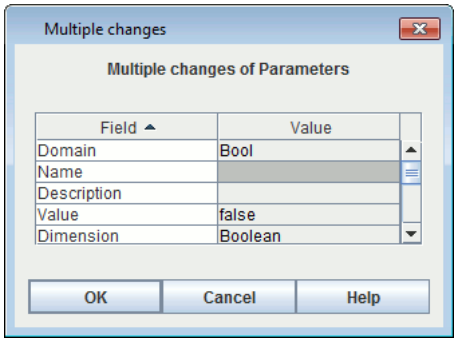


You can choose the columns to be displayed by selecting (or deselecting) the corresponding check boxes. The arrows on the right are used to move the columns up or down in the list to choose the order of the columns. The **Disable data sorting** check box disables the data sorting. This improves the application's performance with very complex models.

The choices that are made are kept on the current document. They will be reapplied when reopening your document and do not affect other documents in the application.

1.3. Multiple edition

To modify data, simply double-click on the cell to modify. When several lines are selected (using the CTRL or SHIFT keys) changes can be made to all the selected data by using **Multiple changes**. A window then opens to allow you to make these changes.

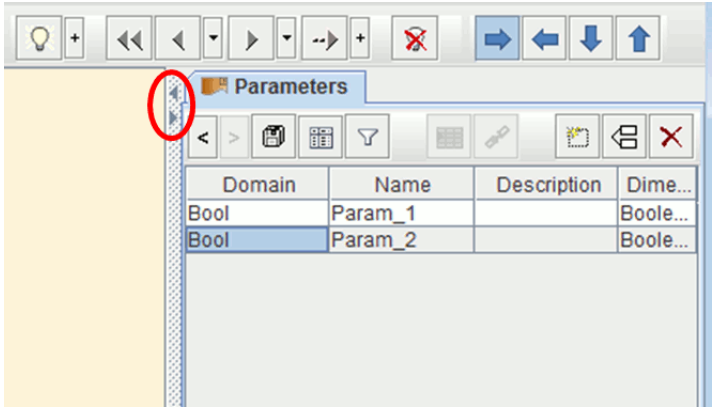


Items which cannot be modified are greyed. The white lines indicate that the selected data does not have the same value for the field in question. A new value can be entered which will be taken into account for all the selected data. The lines with no background colour indicate that all the selected data has the same value for this field (in this example the selected data is all "Float"); they can be changed to give a new value to all the selected data.

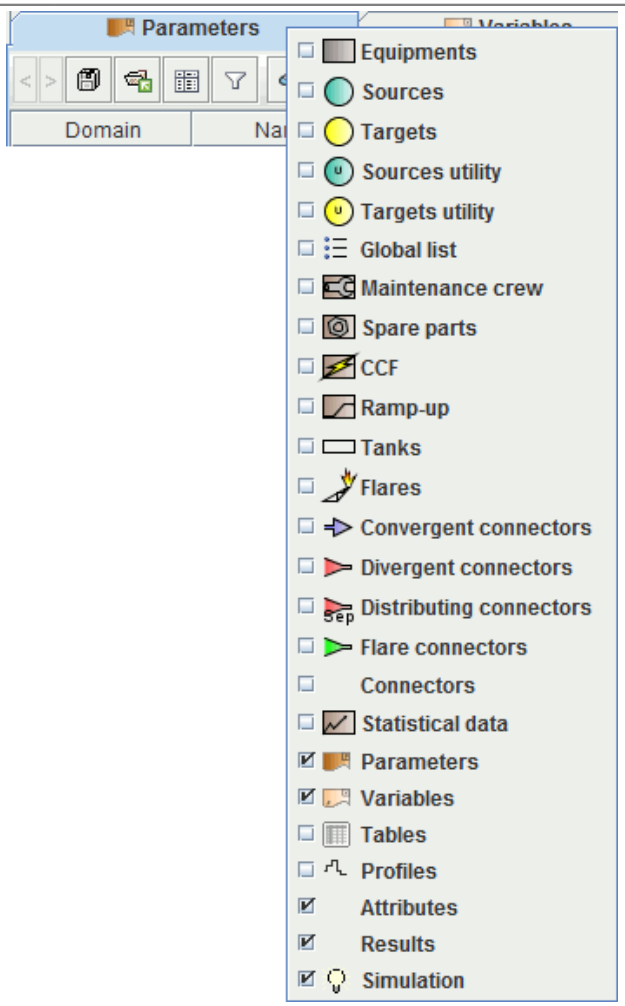
2. Table accessibility

As mentioned above, the tables can be accessed via the **Data and Computations** menu; in this case, each table is displayed in a separate window.

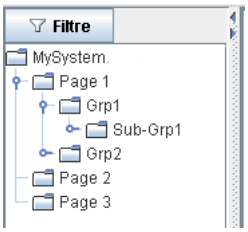
To avoid having too many windows open, all the tables are grouped together in tabs on the right-hand side of the application. This area can be hidden/displayed using the small arrows above the input zone.



It is possible to choose the tables in this zone by right clicking on the tabs. A contextual menu appears, in which the user can select the tables s/he wishes to display.



1.4. Tree view



To help users to walk through the document (pages, groups and sub-groups), a tree is available on the left of the application. By default, every element is displayed, you can use **Filter** button in order to select elements you want to display or not.

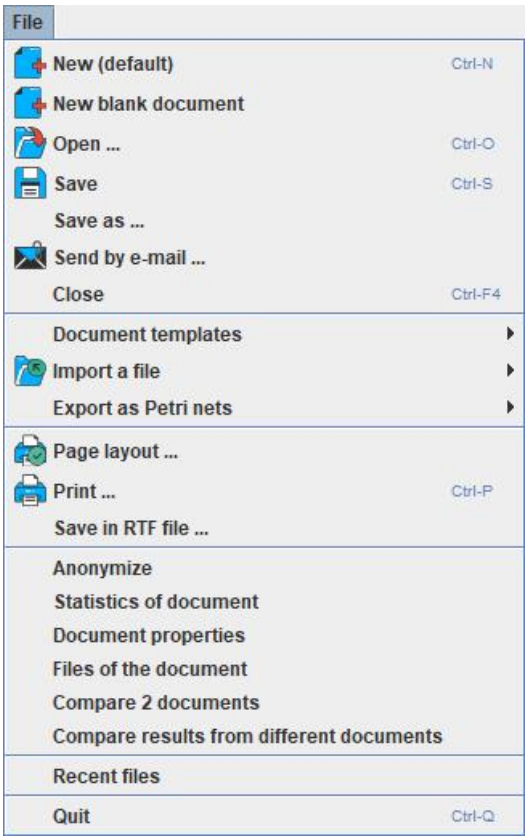
You can expand or collapse a node in a recursive way with a right click on the node.

As explained for tables on the right, you can "hide" the tree.

2. Menus presentation

2.1. File

The menu **File** contains the basics commands : open, close, save, print, etc.



The functionality **New (default)** opens a new document, which will be initialized from the default module's model. You can change the default's model, see Section 17.1, “Document template”

The functionality **New blank document** creates a new blank document.

The functionality **Open** opens an existing document.

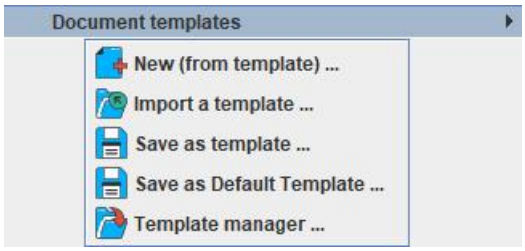
The functionality **Save** saves the current document into a file. The default proposed location for the backup is {répertoire home de l'utilisateur}/GRIF/2022/Petro

The functionality **Save as ...** lets you save a copy of the file you are working on, with a different name or a different location.

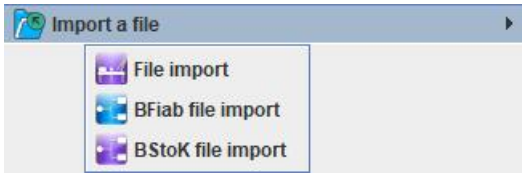
The functionality **Send by e-mail** allows you to attach the current document to an e-mail and then to send it. The configuration of the messaging tool is to be done in the application options Section 19.2, “Executables”

The functionality **Close** lets you close the current document. A window offers to save the file if changes have been made.

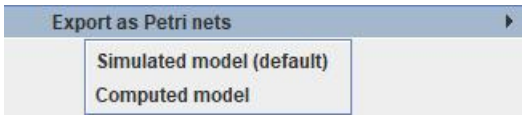
The menu **Document templates** includes features related to document reuse and pre-configuration, see Section 17.1, “Document template”.



The menu **Import a document** contains all the import formats supported by the module.



Function **Export as Petri nets** generates a .jpp file that contains:
The **Computed model**: This is the model used when launching calculations.
Or the **Simulated model (default)** : This is the model used by the interactive simulation. It contains in addition to the calculated model, display information for better readability.



The print functions **Page layout**, **Print** and **Save in RTF file** are described in the section Section 18, “Printing”

The functionality **Anonymize** deletes all the comments and names filled in by the user. The document does not contain any information helping to understand it.

The functionality **Statistics of document** allows to have some information about the document (number of pages, number of groups, etc.).

The functionality **Document properties** allows you to edit the properties of the current document. The fields include: name, creation date, creator, description, version, ... This function is described more specifically in the section Section 6.14, “Document properties / Track change / Images management”

The functionality **Files of the document** includes files within the current document. These files can then be exported in your reports. This feature is described more specifically in the section Section 6.16, “Files of the documents”.

The functionality **Compare 2 documents** highlights the changes made between 2 versions of the same document. This feature is described more specifically in the section Section 6.15, “Compare 2 documents”.

The functionality **Compare results from different documents** opens a window on which you can select results on different document in order to compare them.

The menu section **Recent files** list recently opened files to access them faster.

The functionality **Quit** exits the application. Open documents will be closed.

2.2. Edit

The menu **Edition** contains all the commands needed to edit the current model.

The functionalities **Undo** and **Redo** allow you to cancel or redo the last actions performed. The size of the history of undoable actions are configurable in the application options.

The functionalities **Copy**, **Cut**, **Paste** and **Paste and renumber** are described more specifically in this section Section 6.1, “Copy / Paste / Renumber (without shortcut)”

The functionality **Remove** deletes selected graphic elements.

The functionalities **Overall change ...** and **Selection change ...** search and then replace names and identifiers of the current document or selection. This feature is described more specifically in the section Section 6.3, “Overall change” and Section 6.4, “Selection change”.

Actions **Glue** and **UnGlue** link or unlink graphical objects between them. This allows to fix the position of objects. This feature is described more specifically in the section Section 6.10, “Gluing/Associating graphics”

The functionality **Select all** selects all the graphical elements of the page.

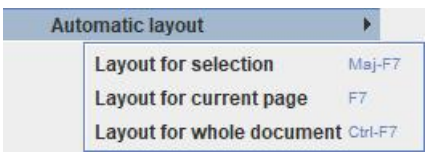
The functionality **Clear selection** deselects items in the current selection. The selection is then blank.

The functionality **Select connected part** selects all graphical elements connected to each other by a link. This feature is described more specifically in the section Section 6.7, “Selecting a connected section”

The functionality **Select sub-tree** from a gate, selects the entire descending parts of the tree.

The functionality **Select from node to root** from a gate or event, selects the entire ascending parts of the tree.

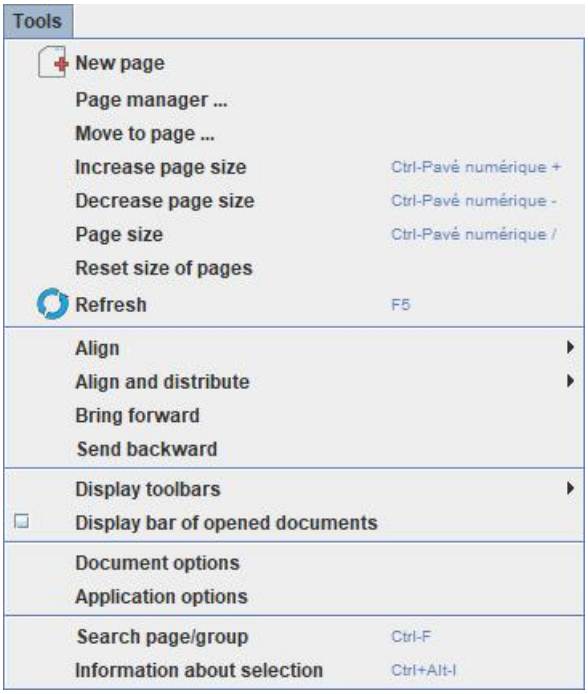
The menu **Automatic layout** gives access to different modes of graphical layouts. This layout can be configured in the document options. See Section 19.3, “Graphics”



The functionality **Properties** edits the logical properties of the current selection.

2.3. Tools

The menu **Tools** contains all the commands needed to manage the current model (management of pages, alignments, options ...).



The functionality **New page** : Create a new graphical page on the current document.

The functionality **Page manager ...** : Open a page manager on which you can rearrange the pages of the document.

The functionality **Move to page ...** : Move the current selection to another page or group in the document.

The functionality **Increase page size** : Increase the graphical input area of the current page.

The functionality **Decrease page size** : Decrease the graphical input area of the current page.

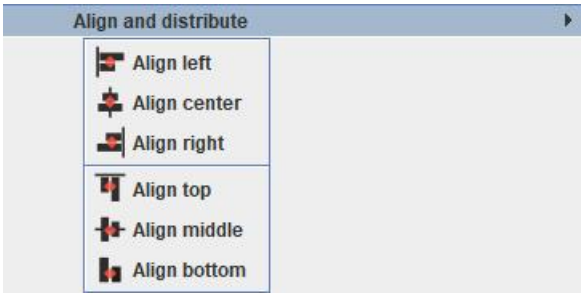
The functionality **Page size** : Open a window to manually configure the size and zoom of the current page. This feature is described more specifically in the section Section 6.8, “Zoom and page size”

The functionality **Reset size of pages** : All pages will be reset to factory format.

The functionality **Refresh** : Refresh the graphical objects in the current page.

The menu **Align** provides alignment functions for graphical objects. These features are described more specifically in the section Section 6.5, “Alignment”.

The menu **Align and distribute** contains the same functions as in the menu **Align**. In addition, it is possible to distribute, in the enclosing space, the elements of the selection.These features are described more specifically in the section Section 6.5, “Alignment”.



The functionality **Bring forward** : Move the selected elements one layer forward.

The functionality **Send backward** : Move the selected elements one layer backward

The **Display toolbars** menu lets you show or hide certain shortcut groups from the toolbar.

The functionality **Display bar of opened documents** : Displays in the lower part of the application, a shortcut bar to access documents already opened in GRIF.

The functionality **Document options** : Opens a window to configure the document options. You have the possibility to configure a very large number of GRIF-Workshop's features (cf. Section 19, “Options of GRIF - Petro”). Some options only apply to the application and are accessible via the menu **Application options**, and others are relative to the document being edited and are defined in the menu **Document options**. However, to avoid having to redefine your options between each document, document options are also available in the application options.
These options will then be applied to all newly created documents.
You can also save the current document settings as the default settings for the application. To do this, open the window **Application options**, then the tabulation **Options** and finally check **Save the options of the current document as default options in the application**.
You will find in this same panel the possibility to override the document options by the application options. To do this, check **The application manages the default options of the documents. Apply the default options to the current document**.

The functionality **Application options** : Opens a window to configure the application options. This window is described more specifically in the section Section 19, “Options of GRIF - Petro”

The functionality **Search page/group** : Find and locate a group or document page.

The functionality **Information about selection** : Display a window based on the selected graphical elements. This window gives additional information about the current selection.

2.4. Document

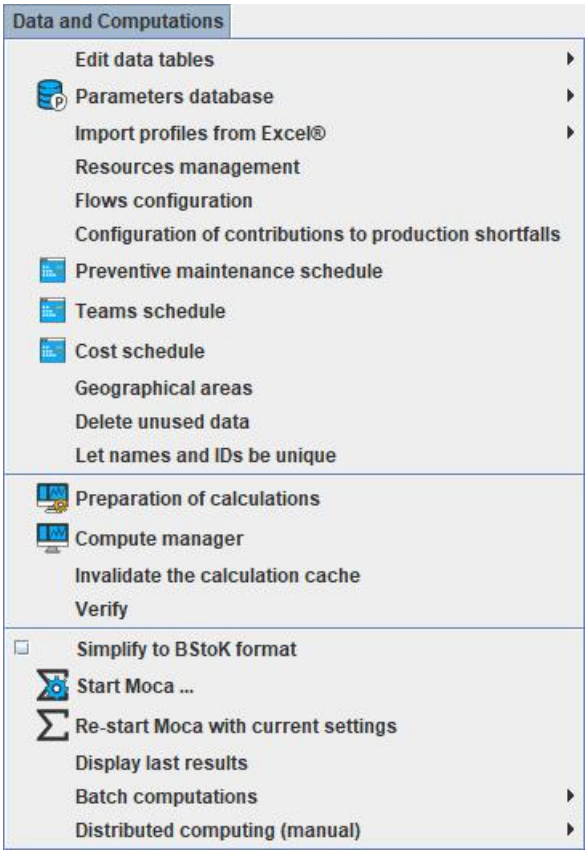
The menu **Document** gives access to all documents being modified or produced.

The functionality **Next** : Selects the next document

The functionality **Previous** : Selects the previous document

2.5. Data and Computations

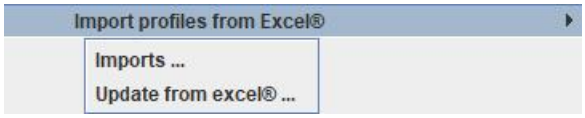
The menu **Data and computations** is divided into two parts : data management (creation and management of the different parameters) and the parameterization / launch of the calculations (calculation duration, sought calculations ...).



The menu **Edit data tables** provides access to a non-blocking window set that presents the data as tables.

The menu **Parameters database** groups all the functionalities concerning the connection of the application to a specific parameters' database. For more details on the parameters databases, refer to Section 15, “Database of parameters”.

Menu **Import profiles from Excel®** is a utility for importing or updating flow profiles from excel files.



Function **Resources management** opens a window showing the resources (maintenance teams and spare parts) used by components and failure modes. This window can also be used to globally modify the settings of component failure modes. For more information on this window, see Section 3.10, “Resources management table”

Function **Flows configuration** allows you to edit or add new flows to the document. Flow configuration is described in more detail here Section 3.2, “Flows configuration”

Function **Configuration of contributions to production shortfalls** allows to configure the calculations of each component to the system losses. For more information on the calculation of contributions to production shortfalls, refer to Section 13.1.1, “Configuring the computations”

Function **Preventive maintenance schedule** allows to have a global and chronological view of preventive maintenance defined on the components. This function is described here: Section 3.11.1, “Presentation”

The functionality **Teams schedule** :

Function **Cost schedule** allows you to plan costs that will be taken into account during the calculations. For more information on cost settings, refer to Section 3.12, “Cost management”

The functionality **Geographical areas** :

The functionality **Delete unused data** : Cleans the document by deleting unused data. A window opens and proposes to manually select the elements to delete.

The functionality **Let names and IDs be unique** : Identifies and modifies duplicate data in the model. In normal use of the software, it is not necessary to use this function.

Function **Preparation of calculations** is a utility that lets you run multiple calculations in parallel on one or more documents at a time. This function is generally used to perform sensitivity analyzes. For more information on using this tool, refer to Section 13.5, “Preparation of calculations”.

The functionality **Compute manager** : Opens a non-blocking window to manage the calculations launched by the application. For more details on the compute manager, refer to Section 13.2, “Compute manager”.

The functionality **Invalidate the calculation cache** : To optimize calculations, some calculations data are cached. Invalidate calculation cache allows you to completely empty these data and ensure authentic results. In normal use of the software, it is not necessary to use this function.

The functionality **Verify** : Checks model data and displays errors.

The check box**Simplify to BStok format** is an option to modify the way the petri net is generated before the calculations. With this mode enabled, different optimizations will be implemented: single-flow export, no back propagation, no demand profiles and no automatic stand-by.

The functionality **Start Moca ...** : Opens the configuration window for Moca computations. For more details on Moca computations, refer to Section 13.1.1, “Configuring the computations”.

The functionality **Re-start Moca with current settings** : Restarts the computation with the last parameterization of Moca computations.

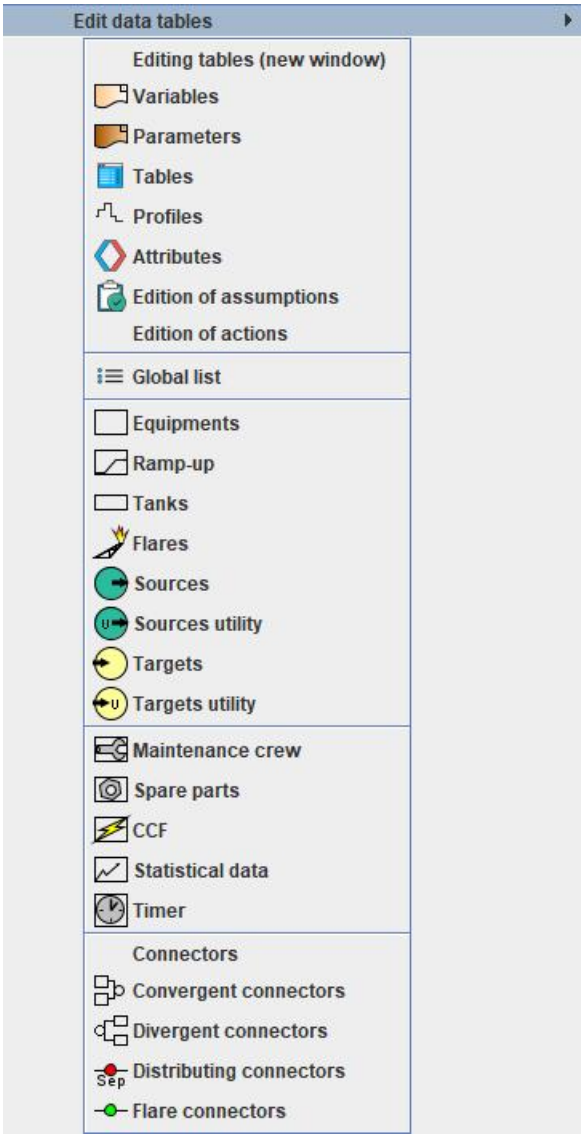
Function **Display last results** : Displays the result window of the last calculation performed. For more information on the result window, refer to Section 13.1.2, “Reading the results (New GUI)”.

The menu **Batch computations** includes the functionalities of multiple computations by variation of parameters. For more details on batch computation, refer to Section 13.4, “Batch computation”.

Function **Distributed computing (manual)** allows you to split the calculations, so that it can be run in parallel on several separate machines.

2.5.1. Edit data tables

The menu **Edit data tables** provides access to a non-blocking window set that presents the data as tables.



Editing tables (new window) : Opens a new non-blocking window containing all the editing tables of the data.

Variables : Opens a non-blocking window containing the variable editing table The creation and settings of variables are detailed here Section 8, “Variables”.

Parameters : Opens a non-blocking window containing the parameter editing table (constants).

Tables : Opens a non-blocking window containing the tables editing panel.

Profiles : Opens a non-blocking window containing the tables profiles editing panel.

Attributes : Opens a non-blocking window containing the attributes editing table. Attributes settings are detailed here Attributes.

Edition of assumptions : Opens a new non-blocking window containing the editing table of assumptions. The assumptions' settings are detailed here Section 6.17, “Hypothesis”.

Edition of actions : Opens a new non-blocking window containing the editing table of actions. The actions' settings are detailed here.

Global list : Opens a non-blocking window containing a table of all the components (equipments, inputs, output, resources, ...) of the model.

Equipments : Opens a non-blocking window containing the equipment editing table. The concept of equipment includes both rotating machines (Section 4.2, “Rotating machines”), static machines (Section 4.3, “Static machines”) and generic components (Section 4.7, “Generic blocks”).

Ramp-up : Opens a non-blocking window containing the ramp-up editing table. Ramp-up settings are detailed here Section 4.5.3, “Ramp-up block”.

Tanks : Opens a non-blocking window containing the tank editing table. Tanks settings are detailed here Section 4.5.4, “The Tank”.

Flares : Opens a non-blocking window containing the flare editing table. Flares settings are detailed here Section 4.5.1, “The flare”.

Sources : Opens a non-blocking window containing the sources editing table. Sources settings are detailed here Section 4.1.1, “Source”.

Sources utility : Opens a non-blocking window containing the sources editing table of utility pages. Sources utility settings are detailed here Section 5.1.1, “Utility source”.

Targets : Opens a non-blocking window containing the targets editing table. Targets settings are detailed here Section 4.1.2, “Target”.

Targets utility : Opens a non-blocking window containing the targets editing table of utility pages. Targets utility settings are detailed here Section 5.1.2, “Utility target”.

Maintenances crew : Opens a non-blocking window containing the maintenance crew editing table. Maintenances crew settings are detailed here Section 4.6.1, “Maintenance crew”.

Spare parts : Opens a non-blocking window containing the spare parts editing table. Spare parts settings are detailed here Section 4.6.2, “Spare parts”.

CCF : Opens a non-blocking window containing the Common Cause Failure editing table. CCF settings are detailed here Section 4.6.3, “Common cause failures”.

Statistical data: Opens a non-blocking window containing the block Statistical editing table. Statistical block settings are detailed here Section 4.6.4, “The Statistics block”.

Timer : Opens a non-blocking window containing the block timer editing table. Timer block settings are detailed here Section 4.6.5, “Timer”.

Connectors : Opens a non-blocking window containing the connector edit table. The concept of connectors groups together converging connectors, diverging connectors, flare connectors and separator connectors.

Convergent connectors : Opens a non-blocking window containing the convergent connectors edit table. Convergent connectors settings are detailed here Section 4.4.1, “Convergent connectors”.

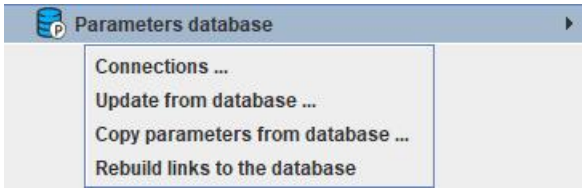
Divergent connectors : Opens a non-blocking window containing the divergent connectors edit table. Divergent connectors settings are detailed here Section 4.4.2, “Divergent connectors”.

Distributing connectors : Opens a non-blocking window containing the distributing connectors edit table. Distributing connectors settings are detailed here Section 4.4.3, “The connector-separator”.

Flare connectors : Opens a non-blocking window containing the flare connectors edit table. Flare connectors settings are detailed here Section 4.4.4, “The flare connector or discharge connector”.

2.5.2. Parameters database

The menu **Parameters database** groups all the functionalities concerning the connection of the application to a specific parameters' database. For more details on the parameters databases, refer to Section 15, “Database of parameters”.



The functionality **Connections ...** : Opens the parameter connection's manager.

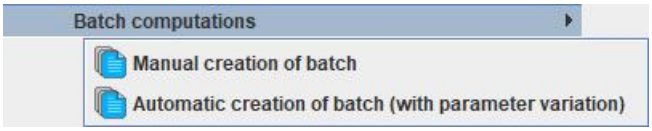
The functionality **Update from database ...** : Updates the settings of the current document parameters that are connected to a database by updating their values. Opens a window to select the data to update.

The functionality **Copy parameters from database ...** : Imports from a parameter database a set of data in the current document. Displays a database parameter table, the user can select the data to import into his document.

The functionality **Rebuild links to the database** : Attempts to reconnect parameter's settings of a document to data from the database Opens a window that highlights parameters that can be reconnected.

2.5.3. Batch computations

The menu **Batch computations** includes the functionalities of multiple computations by variation of parameters. For more details on batch computation, refer to Section 13.4, “Batch computation”.

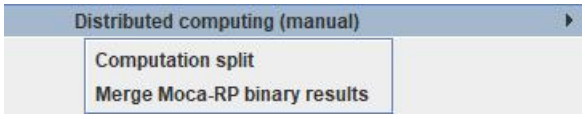


The functionality **Manual creation batch** : Opens a window to create calculation batches. Each batch must be configured by the user.

The functionality **Automatic creation of batch (with parameter variation)** : The user fills in the parameters to be varied, the application generates the calculation batches by making all the combinations of the parameters.

2.5.4. Distributed computing (manual)

Function **Distributed computing (manual)** allows you to split the calculations, so that it can be run in parallel on several separate machines.



Function **Computation split** is the action to choose the number of computers on which the calculations will be made. Target computers must have an installation of GRIF 2022 - Petro and a valid license. Drop the batches (.ini and .mk13) on each of the target computers, and double-click on the mk13 file. The window of the Moca-Launcher opens then allows to start the calculations on 1 or more CPU of the computer. Once the calculations are complete and the results are retrieved, use the **Merge Moca-RP binary results** function to get the final result.

Function **Merge Moca-RP binary results** allows to obtain the final result of a calculation that has been split. Place the intermediate results (.bin) in a directory of the computer where the split took place, open your document at the origin of the computation within Petri12 and then start the step of merging the results.

2.6. Simulation

Menu **Simulation** includes all the features available for interactive simulation. For more information on interactive simulation, see Section 12, “Interactive simulation”.

Function **Start simulation** : start the interactive simulation.

Function **Go to initial state** : restart the simulation

Function **Go back (one step)** : allows you to step back one step in the simulation history

Function **Go forward (one step)** : replay the last step in the simulation history

Function **Go next step** : allows to play the next transition present in the scheduler of the transitions of the simulation.

Function **Stop simulation** : allows to stop the current simulation and to return to an editable model.

Function **Display simulator** : opens a non-blocking window displaying interactive simulation tools.

Function **Legend** : displays a legend about the color codes used by the simulator

2.7. Group

The menu **Group** concerns the capture and management of sub-models grouped into independent subsets.

The functionality **Group** : Puts the selected elements into a new group. A new group graphic object is created. Selected elements are moved within the group.

The functionality **New group** : Create a new empty group on the current page.

Function **New empty sub-system** allows to create a new subsystem without connection or component.

The functionality **Sub-system from selection** allows you to create a new subsystem from the selected elements. For more details on subsystems, refer to Section 3.8, “Sub-systems creation”.

The functionality **Name change** : Edits the name of the selected group.

The functionality **Picture change** : Assigns or modifies the graphical rendering of the group, by adding an image. The name of the group will be displayed below the image.

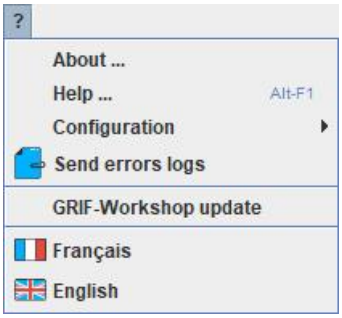
The functionality **Ungroup** : Removes the group and creates all the elements that the group used to contain.

The functionality **Edit group** : Open the group page.

The functionality **Quit group edition groupe** : Go back to the parent page of the group.

2.8. ?

The menu **?** combines several GRIF global configuration functions and provides access to the module's online help.



The functionality **About ...** : Opens an information window about the software version used.

The functionality **Help ...** : Provides access to the module's online help.

The menu **Configuration** groups together several configuration elements of GRIF.

The functionality **Send errors logs** : Sends an email to your reseller with the module's log files.

The functionality **GRIF-Workshop update** : Updates GRIF. This function detects the existence of a more recent version of GRIF. If such a version exists, you will be offered to install it.

The functionality **Français** : Change the application language to French.

The functionality **English** : Change the application language to English.

2.8.1. Configuration

The menu **Configuration** groups together several configuration elements of GRIF.

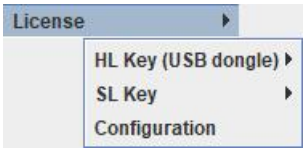
The menu **Licence** groups the configuration functions of the license server. For more information on the use of licenses, please refer to the GRIF installation manual.

The menu **Associate GRIF files** forces your operating system to associate the GRIF files and the different modules that open them.

The menu **Network configuration** : Configures network access to update the system.

2.8.1.1. License

The menu **Licence** groups the configuration functions of the license server. For more information on the use of licenses, please refer to the GRIF installation manual.



The menu **Hardware Licence (HL)** configures USB license dongles.

The menu **Software Licence (SL)** configures license servers that do not require a USB dongle.

The functionality **Configuration**: Configures the access to the license server.

2.8.1.1.1. HL Key (USB dongle)

The menu **Hardware Licence (HL)** configures USB license dongles.

The functionality **Generate c2v...**: Generates a c2v (Client To Vendor) file. This file will be requested by your reseller to create an update of your license.

The functionality **Apply v2c...**: Applies a v2c (Vendor To Client) file. This file will be returned by your reseller to apply the update of your license.

2.8.1.1.2. SL Key

The menu **Software Licence (SL)** configures license servers that do not require a USB dongle.

The functionality **Generate fingerprint...**: Generates a c2v (Client To Vendor) file. This file will be requested by your reseller to create your license.

The functionality **Generate h2h...**: Generates a h2h file (Host To Host)This file is needed to transfer your license to a new server. This feature must be used on the source server. For more information on the license transfer procedure, please refer to the GRIF installation manual.

The functionality **Generate c2v...**: Generates a c2v (Client To Vendor) file. This file will be requested by your reseller to create an update of your license.

The functionality **Apply v2c...**: Applies a v2c (Vendor To Client) file. This file will be returned by your reseller to apply the update of your license.

2.8.1.2. Associate GRIF files

The menu **Associate GRIF files** forces your operating system to associate the GRIF files and the different modules that open them.

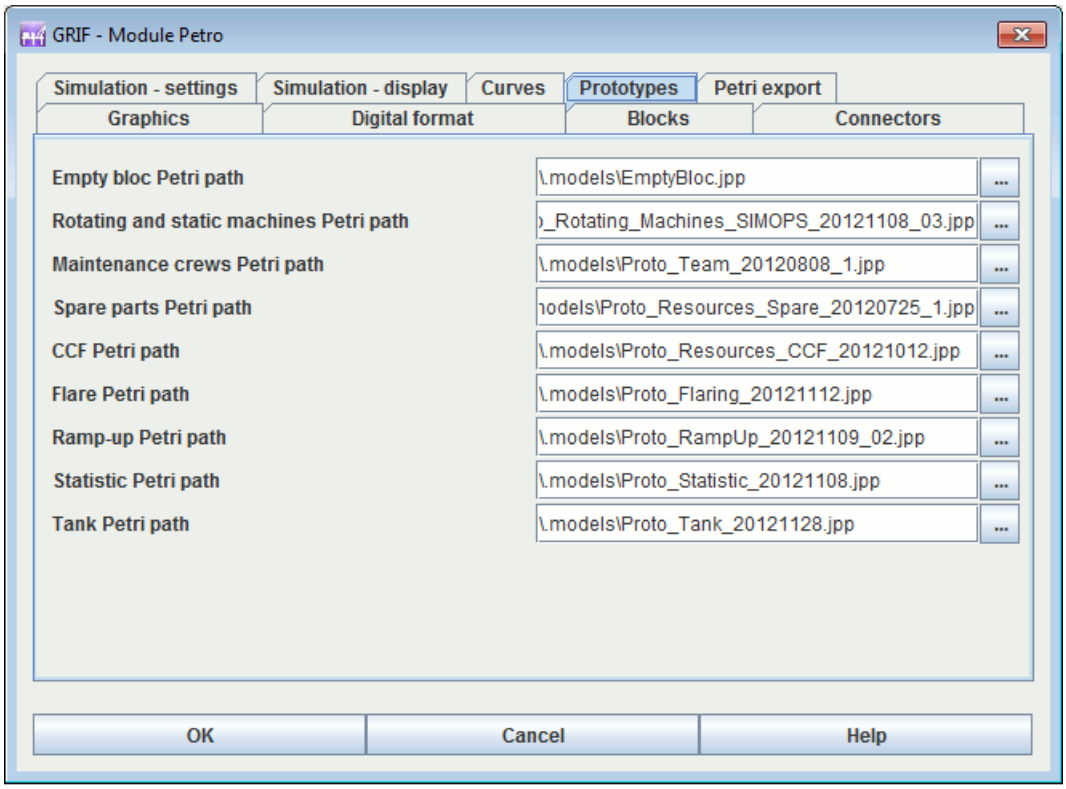
The functionality **For current user** : Associates GRIF files to the current user

The functionality **For all users** : Associates GRIF files to every users. This operation requires administrator rights.

3. General points

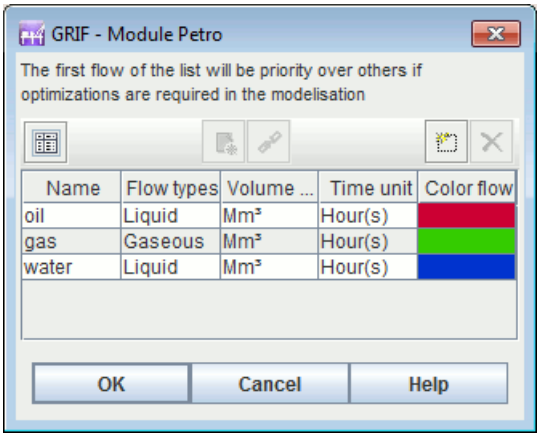
3.1. Prototypes

The Petro module makes it easier to build a Petri net based on production flows. Each exploitable component in Petro serves to configure the Petri net, which will be used by the computation engine. Some components are directly linked to a Petri net. All of the models or prototypes used can be accessed via the user directory, **userdir/Petro/.model/**. You can modify these prototypes in the GRIF-Workshop Petri module. It is however advisable to modify copies of these prototypes and the prototype paths in the document options:



3.2. Flows configuration

The **Flow configuration** function defines flows circulating in the system. The order of the flows is important for calculating priorities, so the first flow will be considered as having priority over the others if optimisations are required on converging connectors.

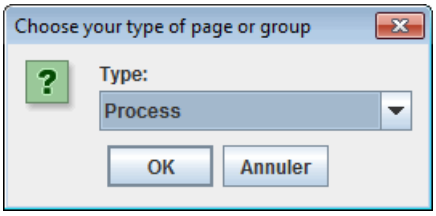


3.3. Type of page

When a new page or a group is created, there are two possible work modes:

1. **Process network**: for working with all the flows. This type of page is used when modelling your production network.

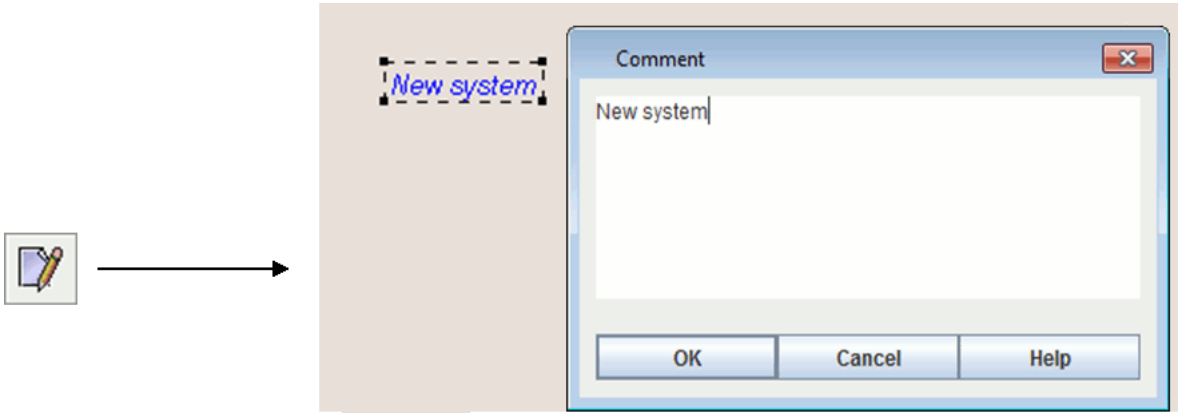
2. **Utility network**: for working with a single-flow network to model an auxiliary circuit. The outputs (targets) of utilities are not considered as production outlets. The utility networks serve to condition the start-up/shutdown of a component for example.



3.4. Entering Comments

To add a comment anywhere on the model, simply click on the pencil icon and position the cursor in a graphical entry zone. The **Comment** dialog box opens and you can then enter your comment.

The "%" sign is a special character and you have to type it twice for a single "%" to be displayed.



3.5. Inputting links and connectors

LINKS

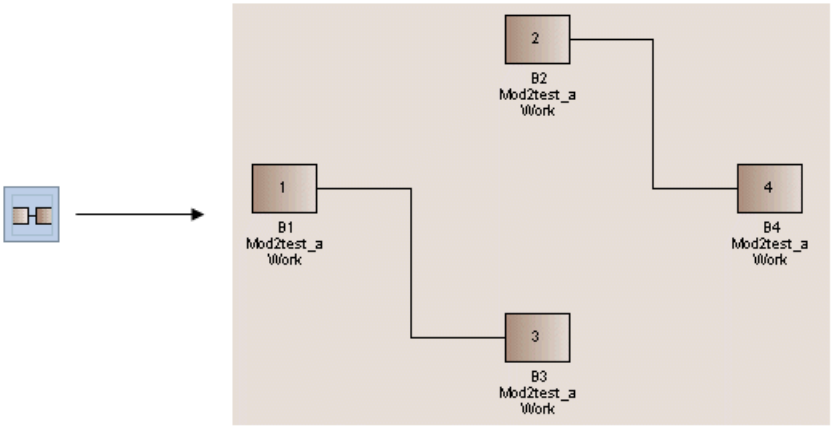
- Function: directly links two elements (block or connector).

- Graphical representation: a non-directional arc.

- Creating a link:
  1. select the corresponding icon on the vertical tool bar;
  2. select a start element (block or connector) by a click left on it;
  3. drag the mouse (without releasing the button) to the arrival element and release the button.

  The order in which the links are drawn can in some cases have an importance (cf. below).

- Example:



In the above example, two serial links have been drawn: one between blocks **B1** and **B3** then the other between **B2** and **B4**.

CONNECTORS

- Function: this element can be the "source" and/or "target" of several links.
  - If it is the "Source" of several links, it is called a **divider** connector.
  - If it is the "Target" of several links, it is called a **K out of N** connector.

- Graphical representation:
  - the "conventional" connectors are brown triangles;
  - the **divider** connectors are red triangles;
  - the **K out of N** connectors are blue triangles.

  Depending on the links which are linked to the connector, it is automatically converted into a **divider** connector or into a **K out of N** connector.

- identification: each connector has
  - a number: It is the "true" identifier (that which will be used by the computation engine). The numbers are automatically incremented. Two connectors cannot have an identical number.
  - a name: enables you to give the connector a name to locate yourself better in the model.

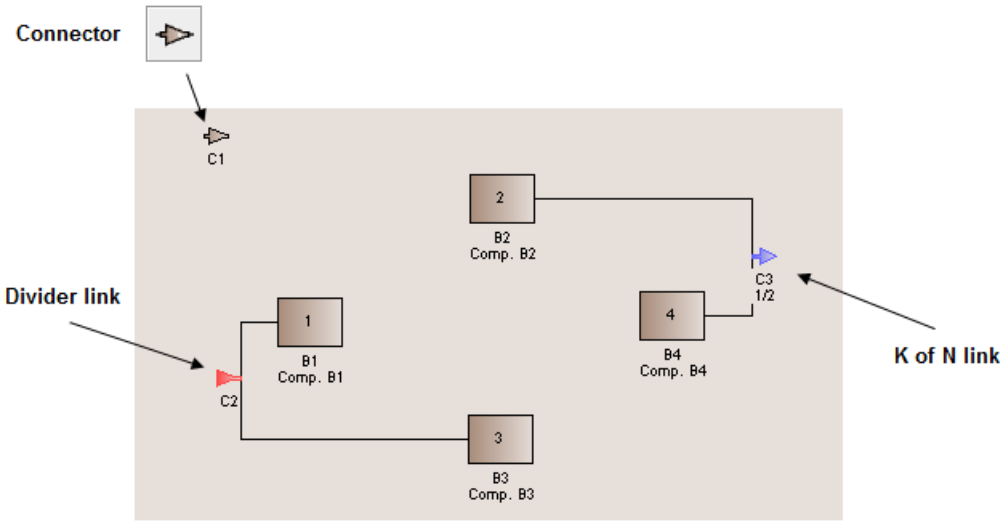
- The properties of **convergent** and **divergent** connectors are set out in sections Section 4.4.1, “Convergent connectors” and Section 4.4.2, “Divergent connectors”.

- Operation:

  To create a "conventional" connector,
  1. select the corresponding icon on the vertical tool bar;
  2. click left at the desired location in the input zone.

  Divider or **K out of N** connectors can be created directly. To do this
  1. select the corresponding icon on the vertical tool bar;
  2. select a start element (block or connector) by a click left on it;
  3. drag the mouse (without releasing the button) to the arrival element and release the button.

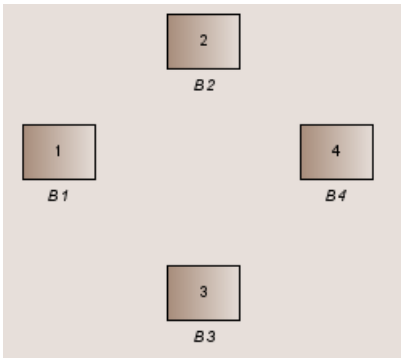
- Example:



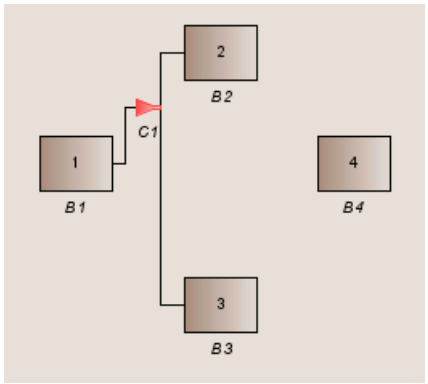
In the example above, a **divider** connector has been drawn between blocks **B1** and **B3** and a **K out of N** connector has been drawn between **B2** and **B4**.

The connectors can be created automatically by creating links between the different elements. Here are some examples to help you understand how this works:

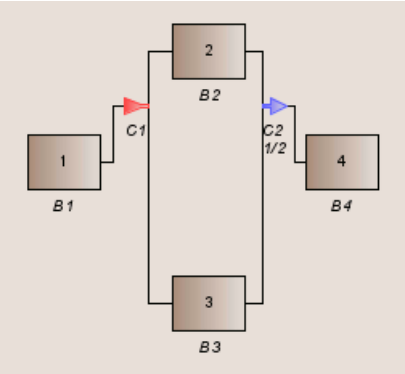
- Let **B1**, **B2**, **B3** and **B4** be four blocks.



- If a link is drawn between **B1** and **B2** followed by another between **B1** and **B3**, then a **divider** connector is automatically created.



- If a link is drawn between **B2** and **B4** followed by another between **B3** and **B4**, then a **K out of N** connector is automatically created (with **K** equal to 1).



- To check that a connector with several inputs and several outputs becomes a conventional connector again, draw a link between **B4** and connector **C1**.

To link new components to B4 (for example), the "target" of the link can be either block B4 or connector C2.

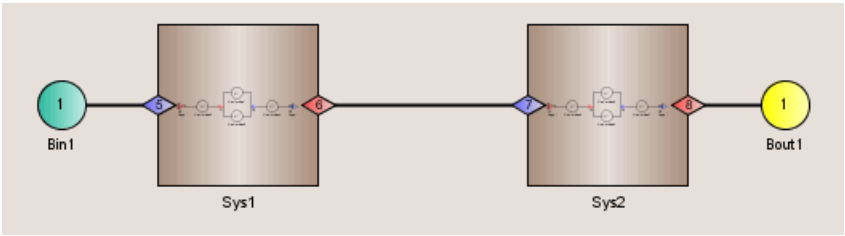
Care must be taken when creating automatic connector since the direction of the links created is very important. Indeed, the direction will determinate the type of connectors (cf. example above).

3.6. Shortcuts to the connectors

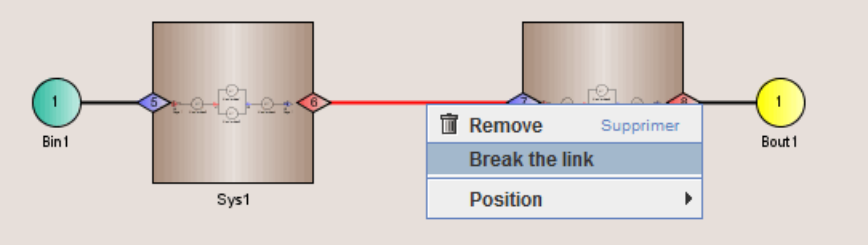
The concept of a **shortcut** (or repeated element) was introduced in the Petro module for four main reasons:

- To link together portions of the model;
- To avoid graphically complex model, and keep readability;
- To simplify the use of the **Group** function (cf. below);
- To highlight what is essential and what is not.

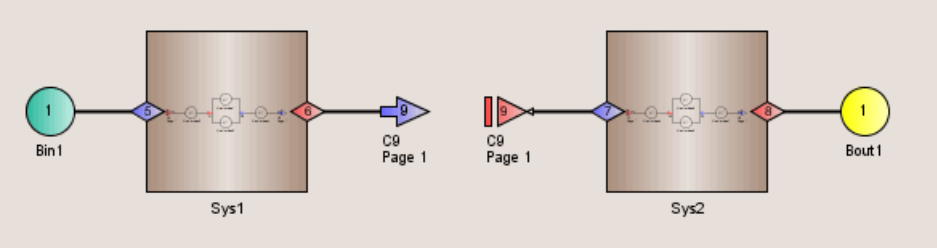
Let two blocks **Sys1** and **Sys2** be connected by single link:



A shortcut link is created simply by right clicking on it. A contextual menu then invites you to **Break the link**. This action will automatically create the connector and its shortcut and make the connections between the blocks.



The **Output** connector (blue) has the same name and number as the **Input** connector (red). Although they are linked from a "computational logic" point of view, the two blocks are now graphically completely independent. They can now be placed on different pages or in different groups (see below).



By double clicking on the **Input** or **Output** connector, it is possible to navigate from one part of the system to the other. Right click on the **Output** connector (blue) to modify its name and identifier.

3.7. Page and group management

The use of shortcuts allowed us to obtain two Petro which have no graphical link between them. They communicate only by **shortcuts**. This can be used, for example, to place each subpart on a different page:

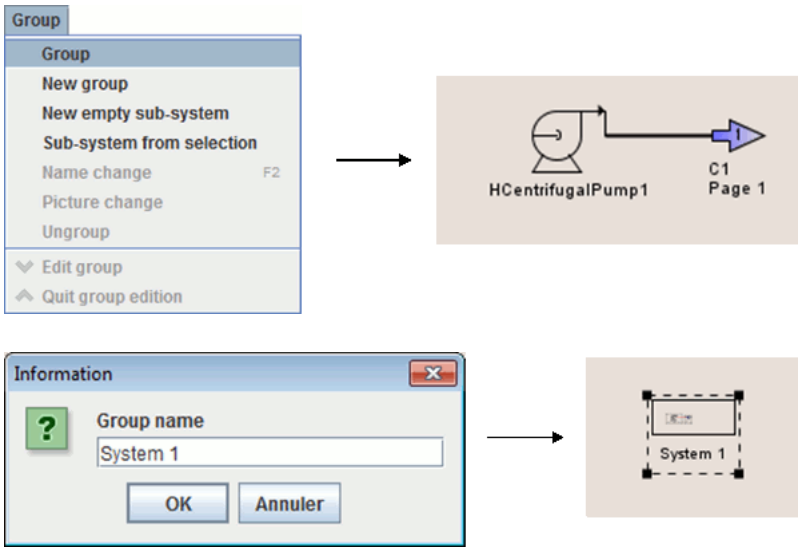
1. Create a new page by clicking the corresponding icon in the icon bar (or use menu **Tools - New Page**). A page number 2 is thus created.
2. Return to page 1 by selecting the page using the page selector in the ideographic command bar (or use menu **Tools - Page manager**).
3. Select the part to be moved.
4. Open menu **Tools - Change page**.
5. Select page 2 and click **OK**. The part selected is transferred to page 2 but it continues to communicate with page 1 via the **shortcuts**.

For large models the division method described above is very useful.

Another possibility for entering large Petro is to use the **Group** concept. This is made possible by the **shortcuts** and the fact that the data is global for a document. This enables quite separate subparts to be created:

1. Select a subpart.
2. Use menu **Group - Group**. A dialogue box then opens asking for the name to be given to the group being created.
3. Enter the desired name and click **OK** (e.g.: "System 1"). The group is created: the subnet is replaced by a rectangle assigned with the chosen name.

You can also create an empty group with **Group - New Group** menu or group tool in the left toolbar.

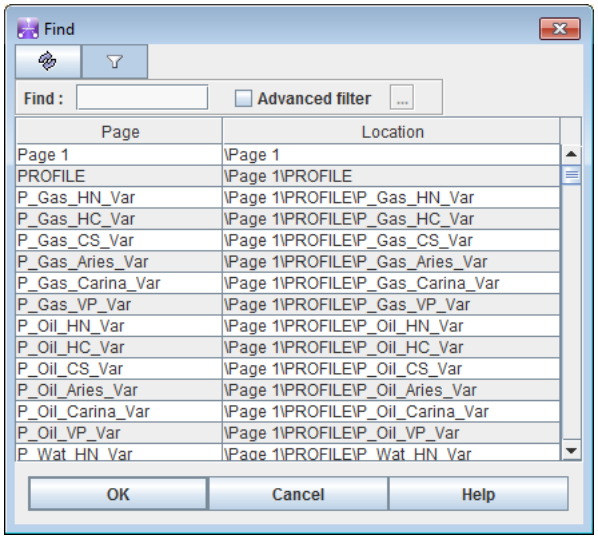


With a right click on the group, it is possible to view inside the group if **Overview of the contents** is checked.

Each group can then be **edited**, **renamed** or **ungrouped** using the commands in the **Group** menu. The group can also be edited with a click right or using the "cursor down arrow" on the left of the page manager. In Edit mode, the submodel can then be modified as you wish. When the modification is terminated you return to the previous figure by exiting group editing by menu **Group - Quit Group Edition**, or using the "cursor up arrow" on the left of the page manager. It's also possible to choose a picture for a group by using **Group - Change Picture** menu.

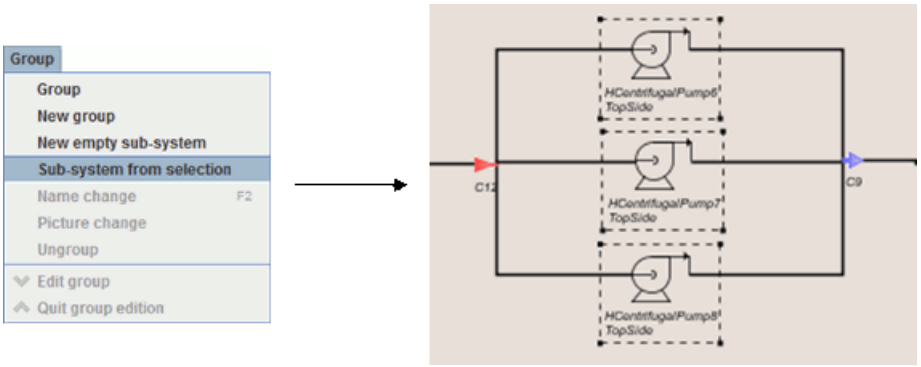
Groups can be grouped recursively.

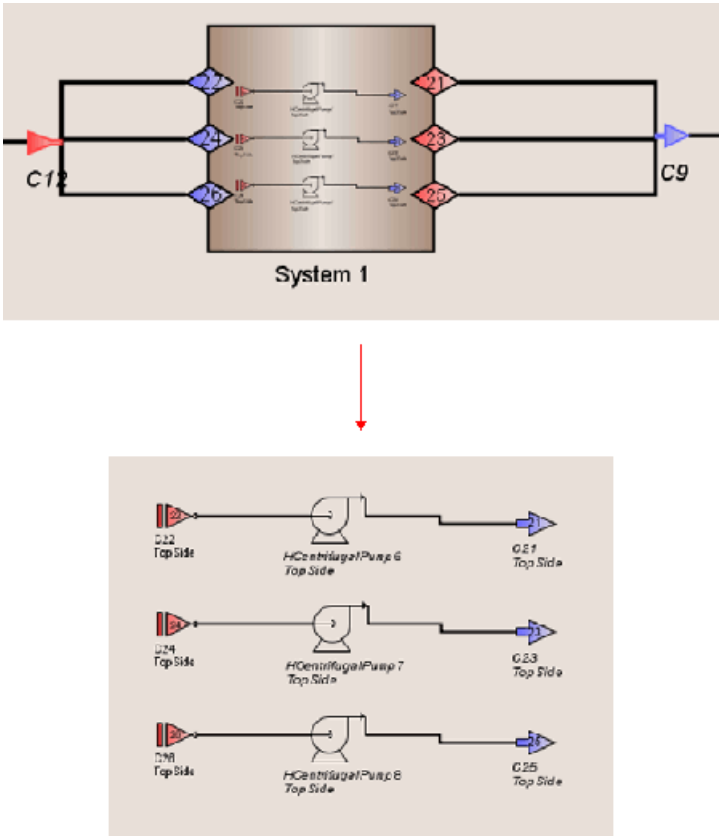
La combinaison **CTRL + F** permet d'effectuer une recherche dans les groupes. Une fois la liste des différents groupes affichée, il est possible de les filtrer ou d'effectuer une recherche dessus.



3.8. Sub-systems creation

In some cases it may be useful to group part of the diagram which involves breaking the links. To do this, use the **Break and Group** function. This is what the result could be for a simple example:

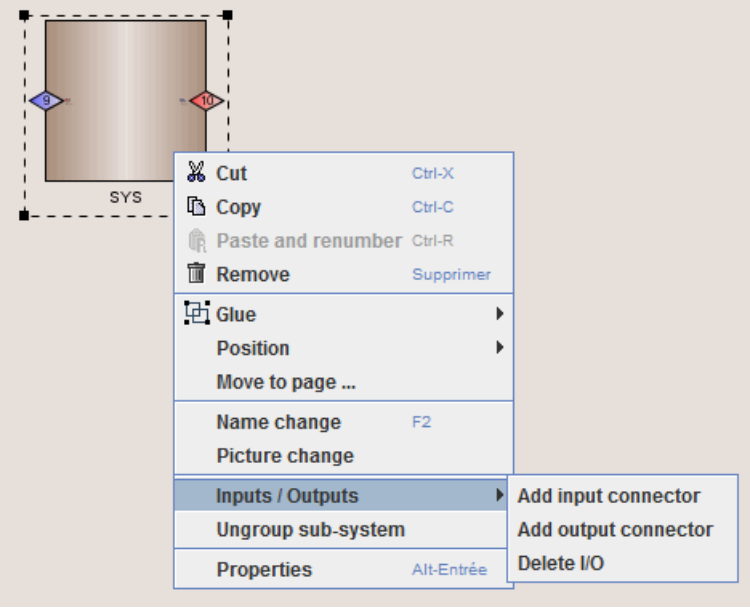




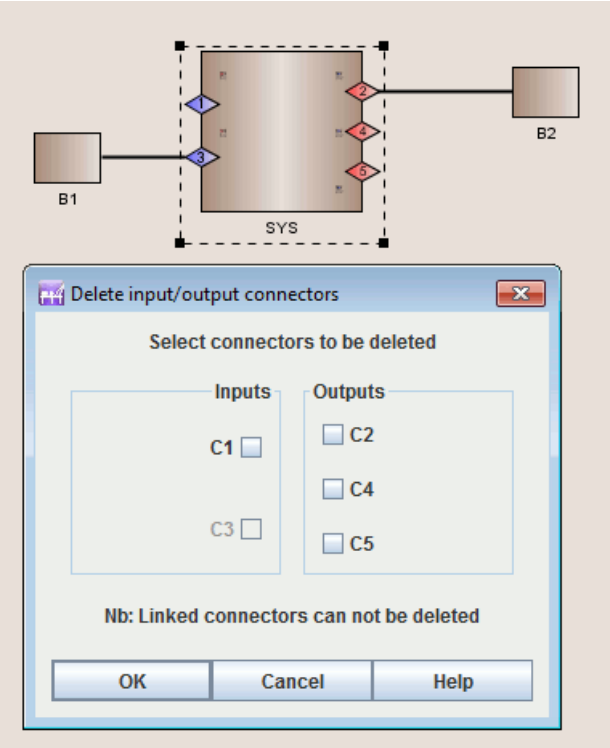
The above figures show how the different shortcuts have been arranged. For each "broken" link, an **Output** connector and its corresponding **Input** have been automatically created. This mechanism has allowed the selected part to become a sub-system.

You can create sub-systems with the **Group** menu, or with a right-clic on the page. By default, a sub-system consists of an input connector, an output connector, and a "group" in which one the behavior of the sub-system is described.

Inputs or outputs can be added or deleted doing a right-clic on the sub-system.



Connector deletion is made with thanks to this window:



Before being deleted, a connector must not be linked to another node (neither in the sub-system, nor outside).

3.9. Sub-system capacity

In some cases, the capacity of a sub-system can be defined, when a component of the sub-system uses a relative capacity of this sub-system. (see section Section 4.2.1.1, “Nominal capacity”).

The figures below show how the capacity of a sub-system can be configured.

Creating the sub-system:

Editing the capacities of the components to enter a relative capacity of sub-system "Sys_B"

Editing the properties of the sub-system:

Configuring the capacity of the sub-system:

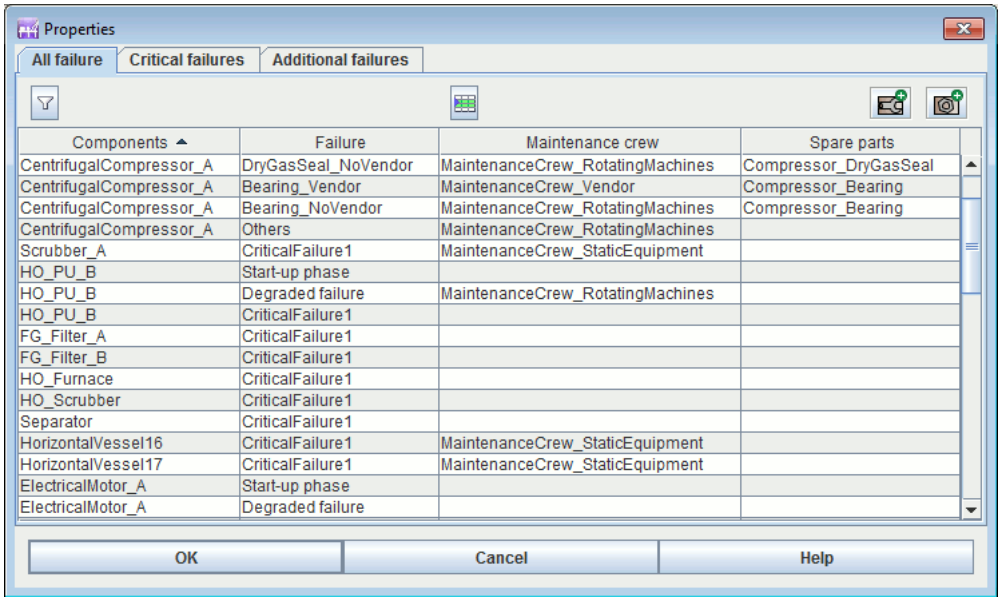
In the same way as for rotating machines, you can choose to provide a **Real capacity** or a **Relative capacity** of the global system or parent sub-system.

The list of components affected gives an overview of the components that use the sub-system when defining their relative capacity.

3.10. Resources management table

This table provides a general view on the use of maintenance teams and spare parts. Each failure mode of each component has its own line in this table. It is composed of 3 tabs offering a breakdown according to the types of failure:

- All failures;
- Critical failures;
- Additional failures;



The following functions appear in the tool bar:

Filter the types of failures displayed. Can also be used to display/hide the components that are not subject to resources management.

Multiple changes. To Add/remove resources in several components in the same time.

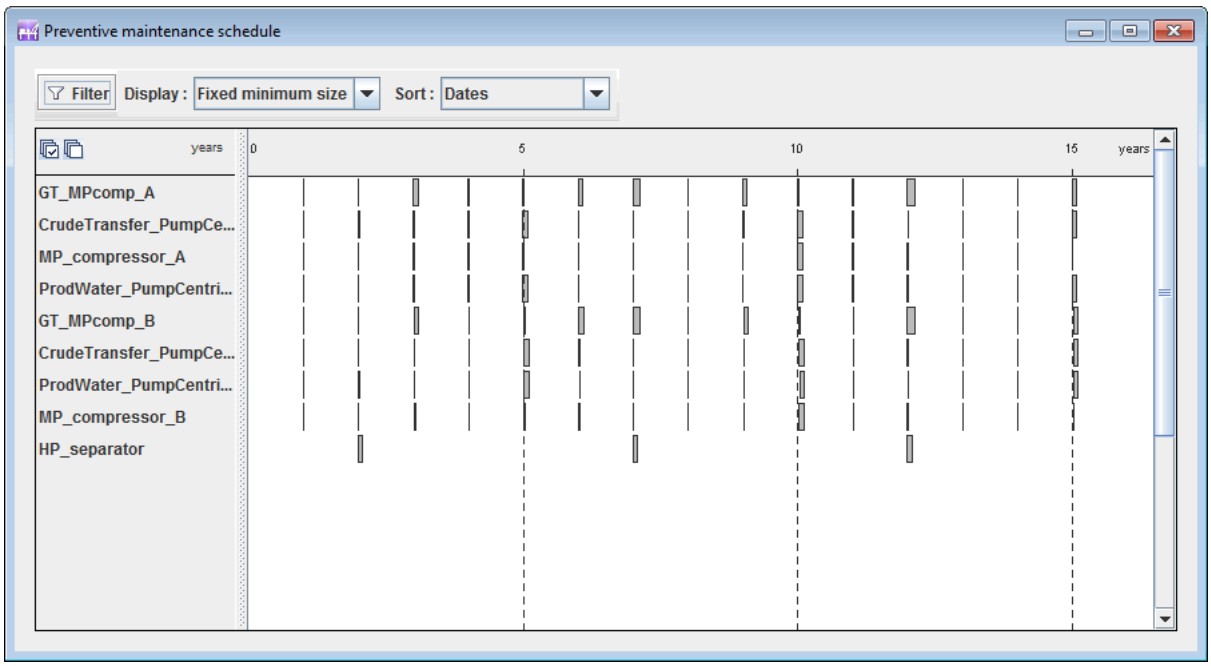
Add maintenance crew. Create directly a new maintenance crew.

Add spare parts. Create directly a new type of spare parts.

3.11. Preventive maintenance schedule

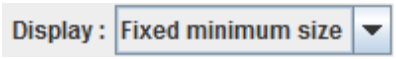
3.11.1. Presentation

This function gives a global and chronological view of the preventive maintenance operations planned for each component.



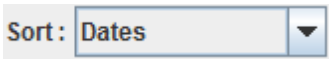
The following functions appear in the tool bar:

Filter the types of maintenance displayed. Can also be used to display/hide the components that are not subject to preventive maintenance.



Choice of the period display policy.

- Linear: size of the periods proportional to their length.
- Logarithmic: logarithmic period size.
- Fixed minimum size: linear with stages for minimum sizes.

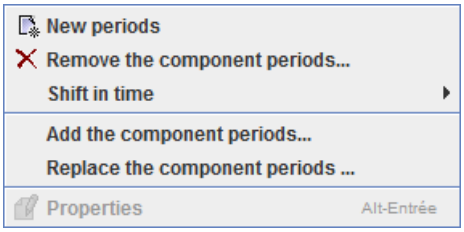


Vertical sort on the display of the schedule.

Select all components.

Deselect all components

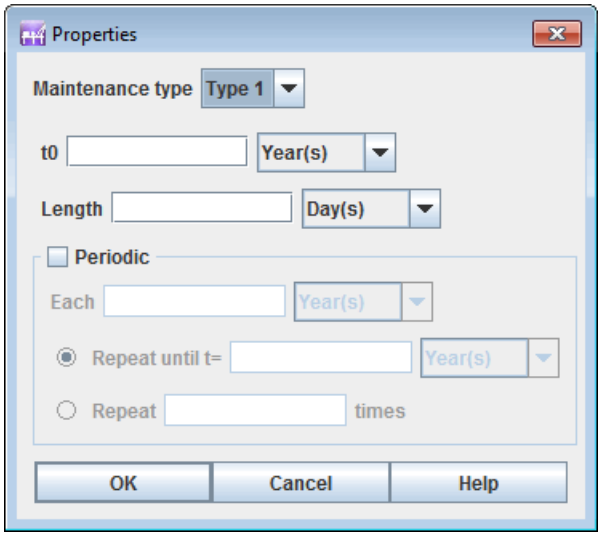
When you select one or more components, right clicking on the selection displays a contextual menu with the following functions:



- New periods: adds new periods. This function is described below (Section 3.11.2, “New periods”)
- Remove the component periods: removes all the component’s periods or only those from a certain date.
- Shift in time: shifts the periods selected in time. Entering a negative number is authorised to bring the period forward in time.
- Add the component periods: copies the periods of the selected component and adds them to the periods of the current component.
- Replace the component periods: deletes the periods of the current component and replaces them with those of the selected component.
- Properties: gives access to the property window of the selected component. This window can also be accessed by double clicking on a line.

3.11.2. New periods

In this window you can add new periods to the selected component(s).

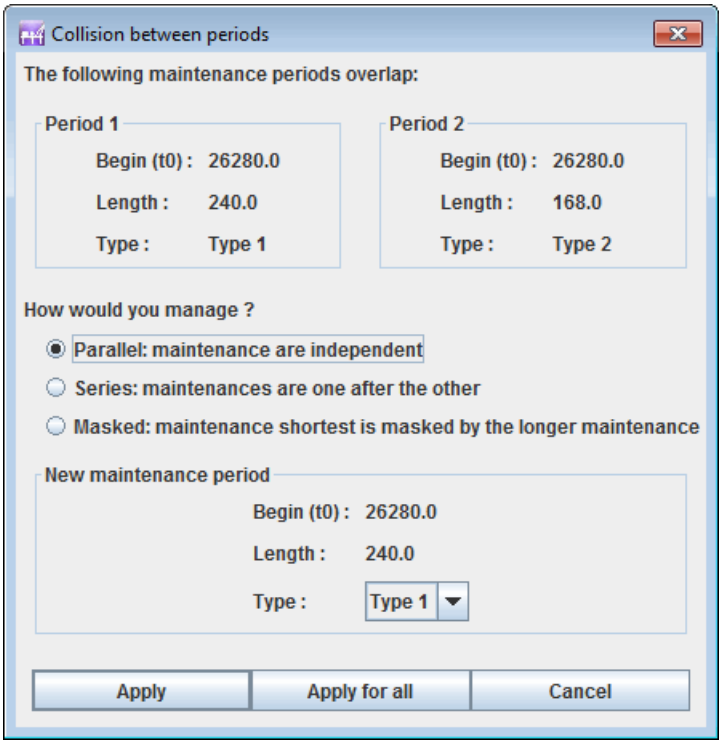


- Maintenance type: defines a specific maintenance type
- t0: start date of the period
- Length: length of the period
- Periodic: generates several maintenance operations of the same length
- Every: defines the t0 of the periodic maintenances
- Repeat until t=: defines the number of periods to be generated
- Repeat X times: defines the number of periods to be generated

If periods conflict (i.e. overlap) with existing periods, a conflict management window will be displayed. Conflict management is explained below.

3.11.3. Conflict management

In this window, you can manage any possible conflicts between two periods. For the model to function properly, it is not possible to have two periods defined at the same time for the same component.



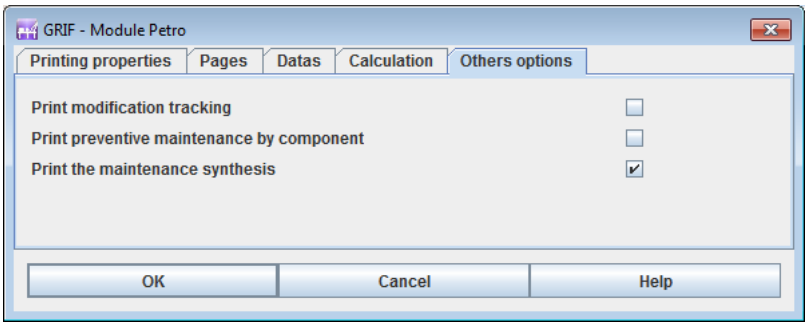
There are three ways of correcting conflicts:

- Parallel: maintenance operations are independent. The period generated is defined as follows: t = min(t0, t1), length = max(t0 + length0, t1 + length1)
- Series: maintenance operations occur one after the other. The period generated is defined as follows: t = min(t0, t1), length = length0 + length1
- Hidden: The shortest maintenance is hidden by the longest maintenance The period generated is defined as follows: if length0 > length1 then period0, if not period1

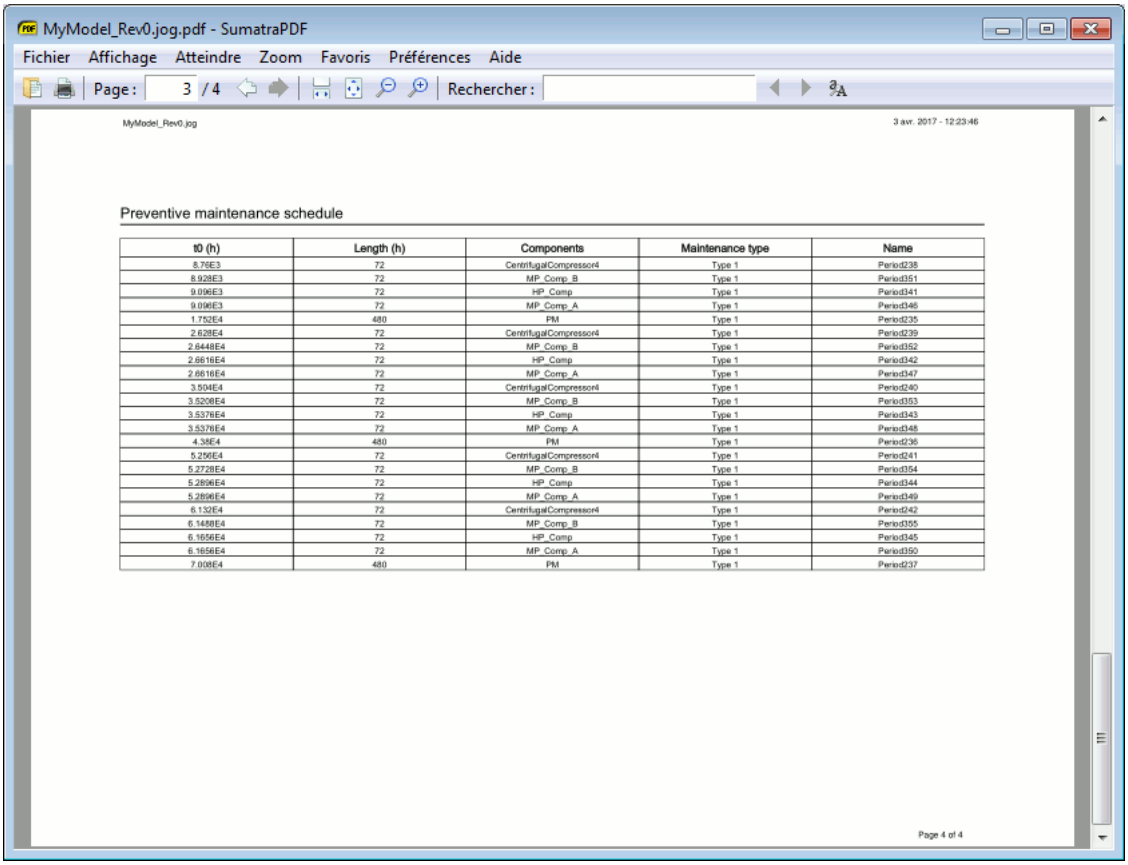
The **Apply** button applies the choices made to the periods currently in conflict. The **Apply for all** button applies the same choice to all the conflicts detected during the operation.

3.11.4. PDF report

During the printing of the report, the maintenance schedule can be inserted into the PDF report. For this, in **File - Print** select **Print the maintenance synthesis** in **others options**.



In the PDF report, the following table is added:



t0 (h)	Length (h)	Components	Maintenance type	Name
8.76E3	72	CentrifugalCompressor4	Type 1	Period238
8.928E3	72	MP_Comp_B	Type 1	Period351
9.096E3	72	HP_Comp	Type 1	Period341
9.096E3	72	MP_Comp_A	Type 1	Period346
1.752E4	480	PM	Type 1	Period235
2.628E4	72	CentrifugalCompressor4	Type 1	Period239
2.6448E4	72	MP_Comp_B	Type 1	Period352
2.6616E4	72	HP_Comp	Type 1	Period342
2.6616E4	72	MP_Comp_A	Type 1	Period347
3.504E4	72	CentrifugalCompressor4	Type 1	Period240
3.5208E4	72	MP_Comp_B	Type 1	Period353
3.5376E4	72	HP_Comp	Type 1	Period343
3.5376E4	72	MP_Comp_A	Type 1	Period348
4.38E4	480	PM	Type 1	Period236
5.256E4	72	CentrifugalCompressor4	Type 1	Period241
5.2728E4	72	MP_Comp_B	Type 1	Period354
5.2896E4	72	HP_Comp	Type 1	Period344
5.2896E4	72	MP_Comp_A	Type 1	Period349
6.132E4	72	CentrifugalCompressor4	Type 1	Period242
6.1488E4	72	MP_Comp_B	Type 1	Period355
6.1656E4	72	HP_Comp	Type 1	Period345
6.1656E4	72	MP_Comp_A	Type 1	Period350
7.008E4	480	PM	Type 1	Period237

3.12. Cost management

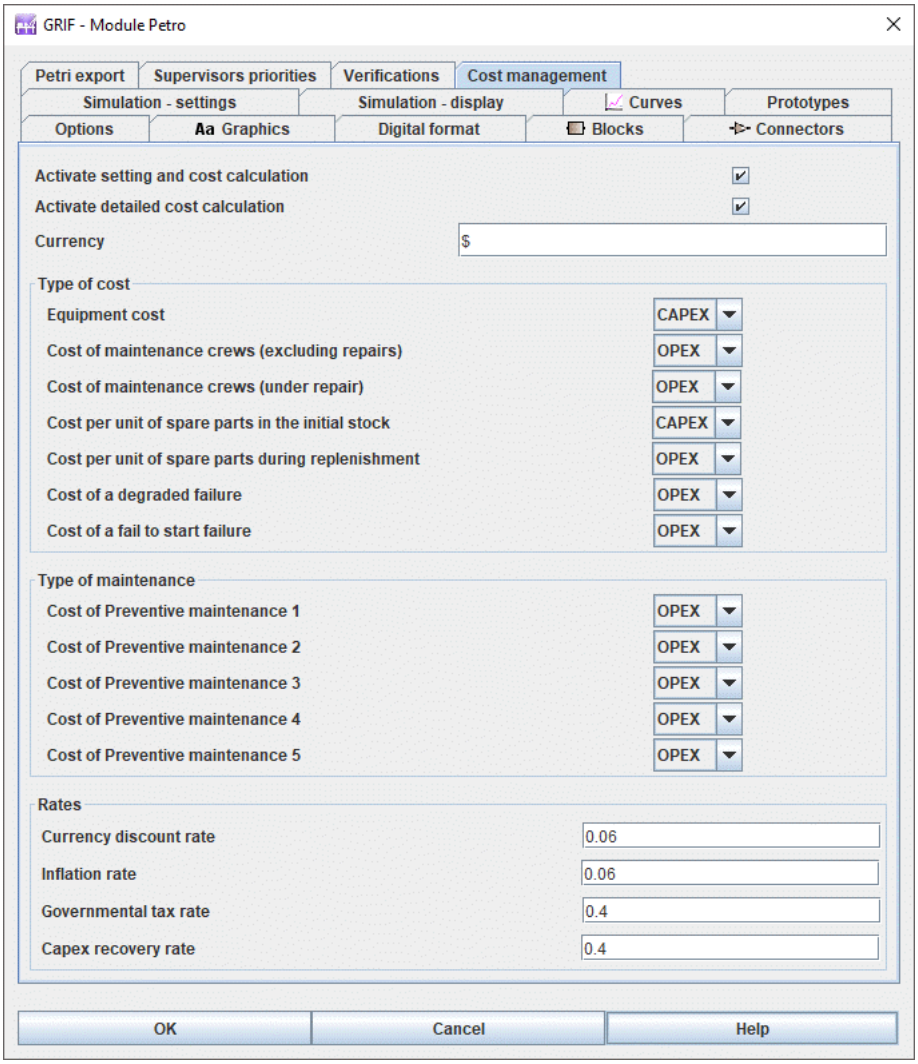
3.12.1. Cost management activation

In Petro cost can be considered in different places:

- for purchasing equipment;
- for mobilisation and cost for maintenance crew;
- for the purchase and replenishment of spare parts;
- for preventive maintenance (hourly and fixed cost) ;
- for environmental taxes related to flaring policies and cost per unit of flared quantities.

These cost can be activated in **Tools - Document (or Application) options - Cost management** menu.

the following window appear to configurate the cost management:



The following options are available:

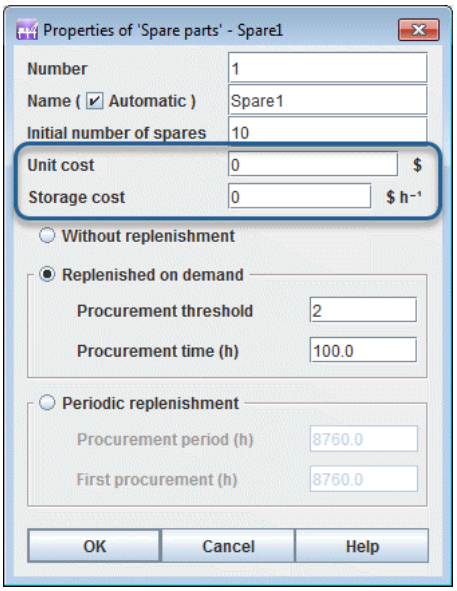
- **Activate setting and cost calculation**: enables to activate cost calculation in the model;
- **Activate detailed cost calculation**: enables to have detailed information about the origins of the costs, in particular results for each maintenance team, for each spare parts stock and for each component;
- **Currency**: enables the user to choose his currency;
- **Type of cost**: for each cost, it is possible to precise if it is **OPEX** operation (OPEX = OPerational EXpenditure) it means operating expenses or **CAPEX** (CAPEX = CAPital EXpenditure) it means investment type.
- **Type de maintenance** : for each maintenance type, it is possible to precise if it is **OPEX** operation (OPEX = OPerational EXpenditure) it means operating expenses or **CAPEX** (CAPEX = CAPital EXpenditure) it means investment type.
- **Rates** used in post-treatment to take into account evolution of some parameters.

3.12.2. Consideration

When this option is checked, it is only necessary to provide the different amounts in the parts intended for this:

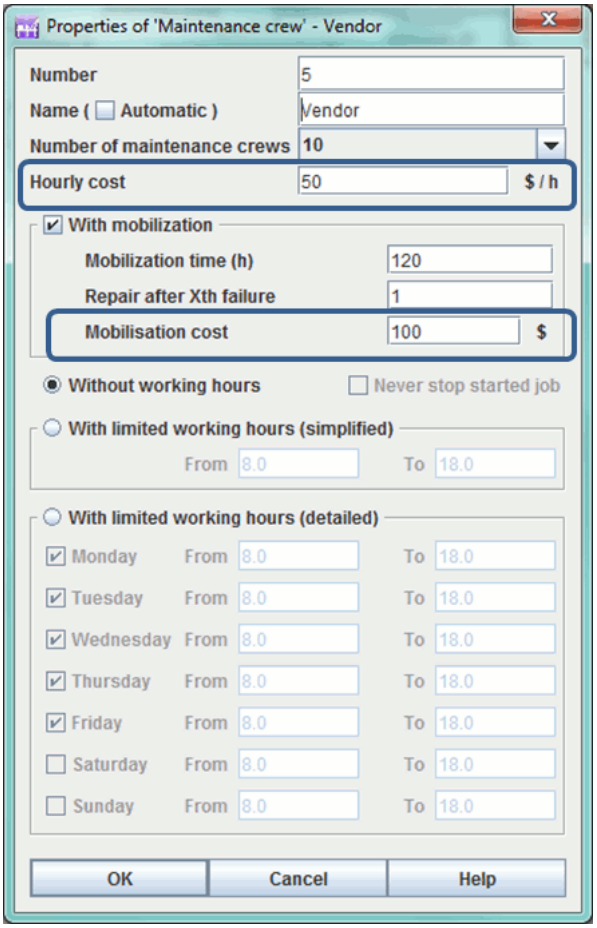
3.12.2.1. Spare parts

For a spare parts, the costs take into account are: procurement price by unit and hourly storage cost.



3.12.2.2. Maintenance team

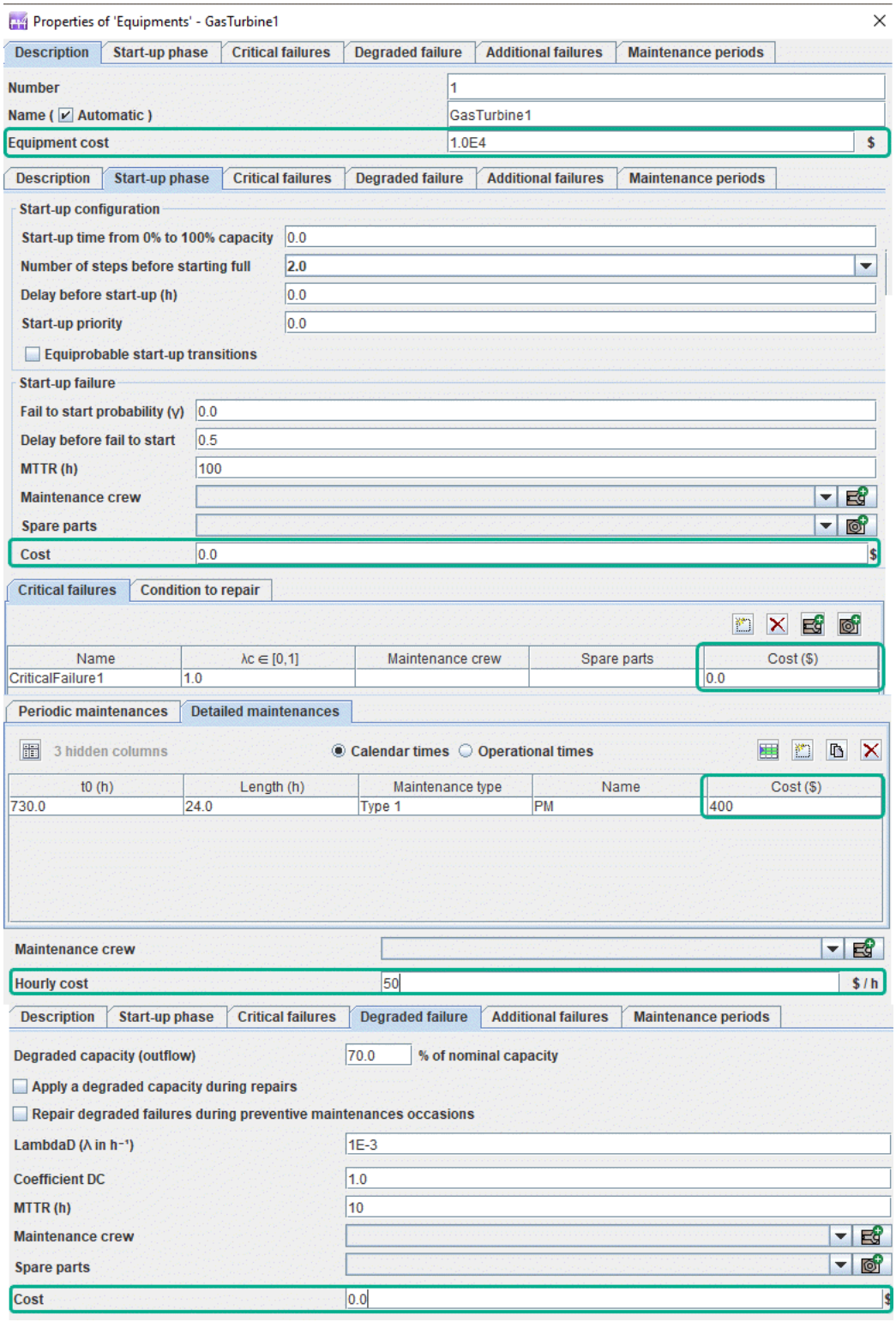
Hourly cost intervention for a maintenance crew is considered. If a mobilisation is necessary, a fixed cost due to the mobilisation can be added.



3.12.2.3. Components

- **Equipment cost**: in **Description** tab with the equipment properties, the equipment price can be considered;

- **Cost**: in **Start-up phase** tab with the start-up failure properties, the price of a start-up failure can be considered;
- **Cost**: in **Degraded failure** tab, the cost of a degraded failure can be considered;
- **Degraded failure**: in **Description** tab with the equipment properties, the equipment price can be considered;
- **Critical failures** and **Additional failures**: in this case a fixed cost due to the failure is considered;
- **Maintenance periods**: It is possible to add an hourly cost related to the preventive maintenance of the current component. The **Cost** column of the component enables to impute a specific cost for maintenance.

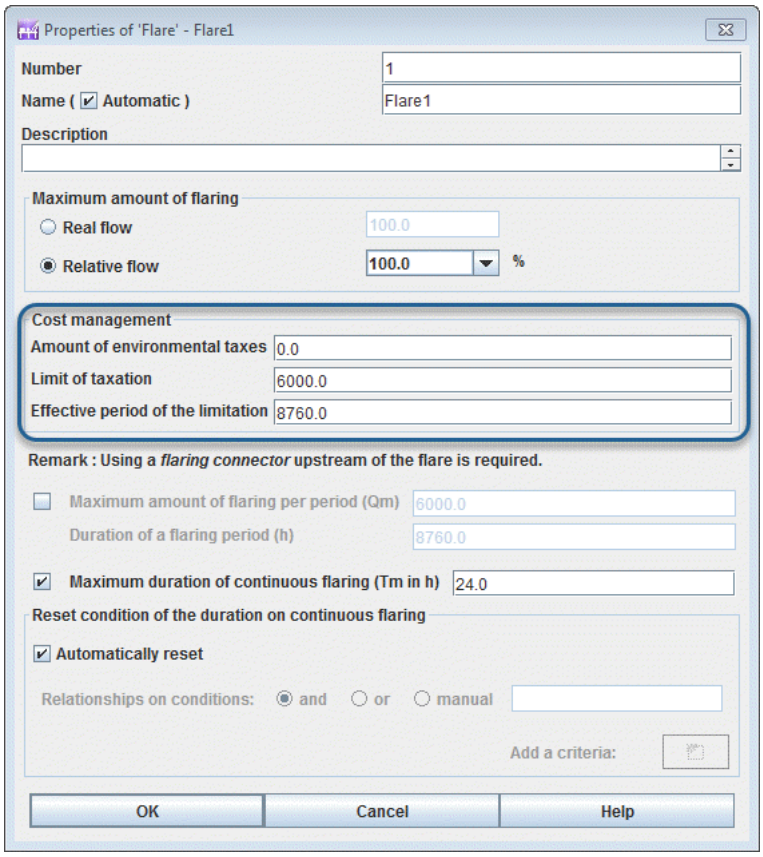


3.12.2.4. Flares

The **amount of environmental taxes** defines a cost which is taken into account if the **Limit of taxation** is exceeded. The final amount depends on the flow rate. The input amount is applied per quantity of gas flared.

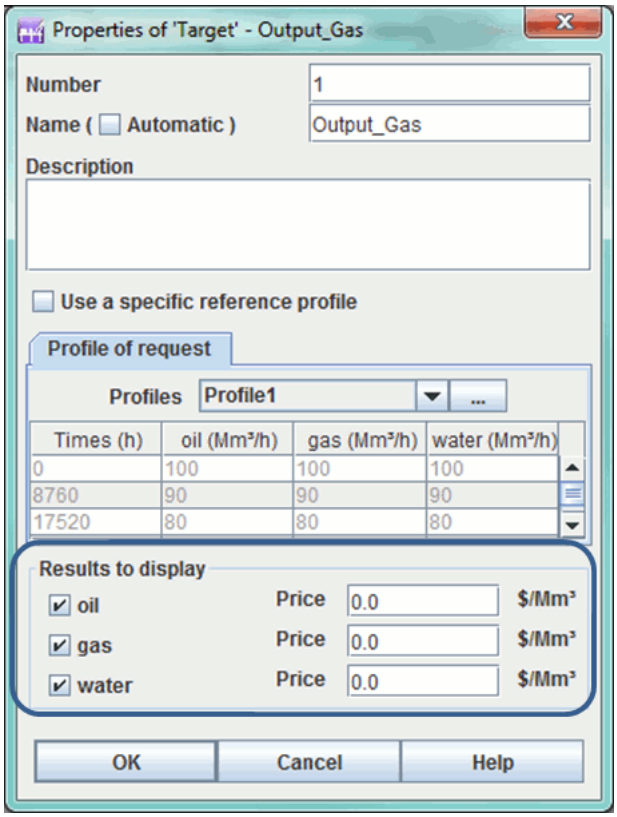
The **limit of taxation** enables to define the quantity that it is possible to burn without tax.

The **Effective period of the limitation (h)** is the period of the tax application limitation - the time interval before the threshold reset.



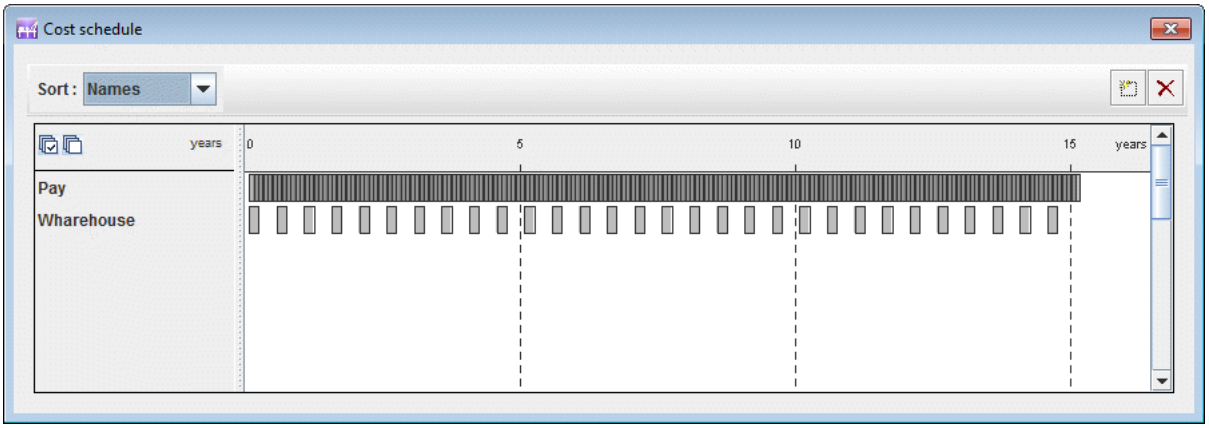
3.12.2.5. Output

For each flow observed in an output, it is possible to define a sale price. Thus, the produced quantities are taken into account and counted in the calculation results.

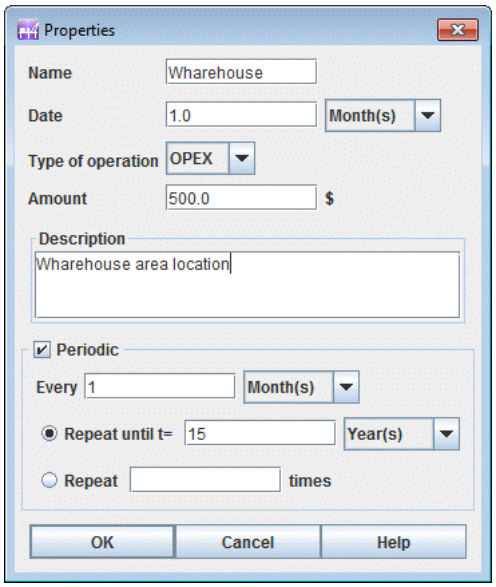


3.12.3. Planned costs

The **Cost schedule** (**Data and Computations** menu) is used to plan the cost during exploitation duration.



Add button add a new line in the schedule and to define a punctual or a recurring operation.



4. Process Diagram Block

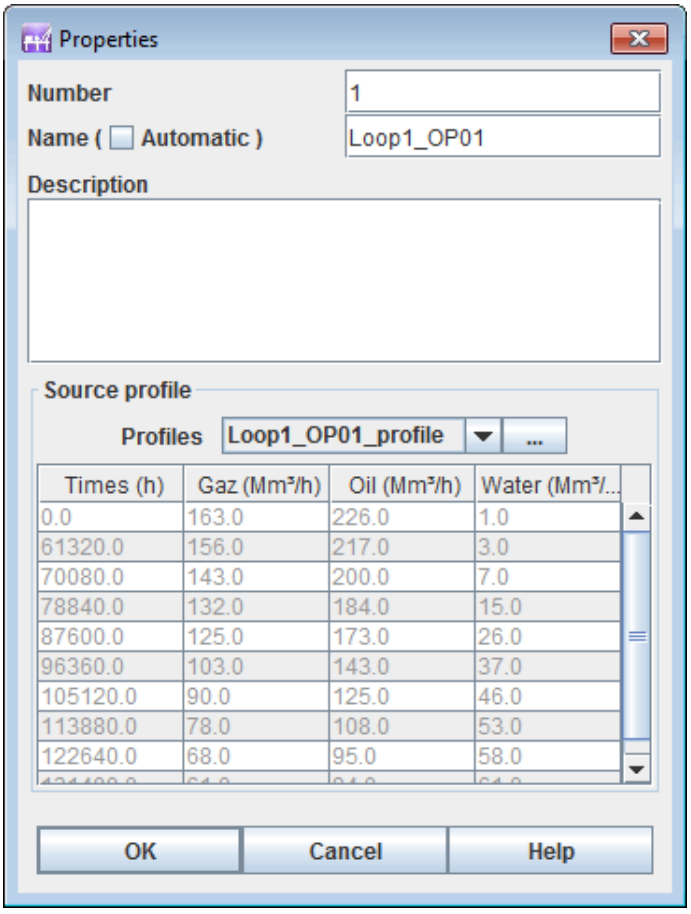
This part describes the components available on the Process pages of the Petro module.

4.1. Input and output nodes

4.1.1. Source

An input node for **process** networks. In the context of an oil and gas installation, it may correspond to an effluent tank (e.g. oil, gas, water).

The source can be configured by double-clicking on the component or by a right click, then selecting the **Properties** menu.



A **Number** and a **Name** can be entered here. Tick the **Automatic** box to generate a name comprising a base name followed by the number.

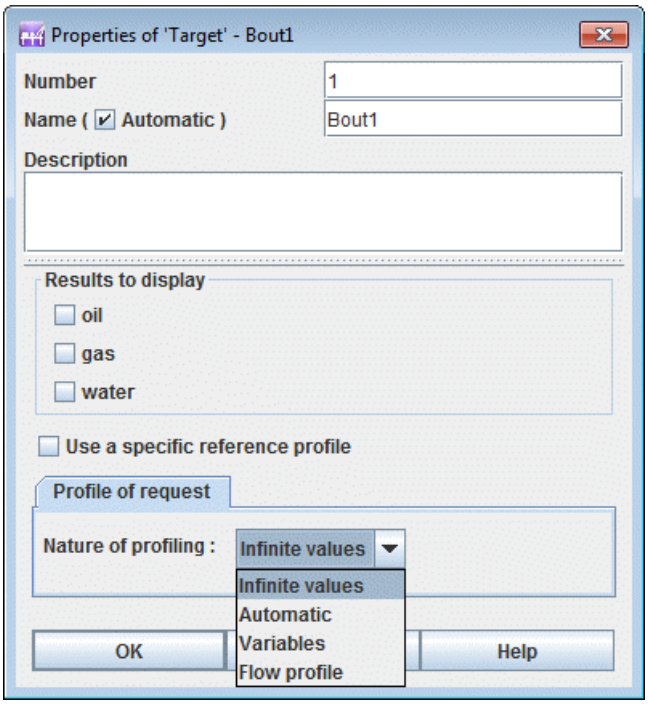
A description of the component can be entered in the **Description** field.

The **Profile** indicates the potential quantities that can be extracted from the well over time, which is called the nominal well profile (this profile is infinite by default).

4.1.2. Target

Output node for **process** networks which gives the produced quantity of each effluent.

The target can be configured by double-clicking on the component or by right-clicking and selecting the **Properties** menu.



A **Number** and a **Name** can be entered here. Tick the **Automatic** box to generate a name comprising a base name followed by the number.

A description of the component can be entered in the **Description** field.

The **Profile of request** indicates the desired production quantities over time. The request is infinite by default. By default, the request is infinite but it is possible to modify the **Nature of profiling**:

- **Infinite values** : (by default) the target will always ask infinity for each flow.
- **Automatic** : The profile will be pre-calculated according to the profiles of the sources present on the upstream branch.
- **Variables** : Allows each flow to be defined by a value or variable expression.
- **Flow profile** : Used to enter the variations over time of the values of the flows.

The **Reference profile** serves as an alternative profile to the profile of request to calculate the production availability percentage. It is not used in the simulation but only when the results are displayed.

The **Results to display** section is used to select the flows that will be observed at the output.

4.2. Rotating machines

There are currently 14 different types of rotating machines, which are grouped into 3 categories in the toolbar:

1. Gas turbine, steam turbine, electric motor, combustion engine, electric generator

2. Centrifugal compressor, screw compressor, reciprocating compressor, turbo-expander, blower

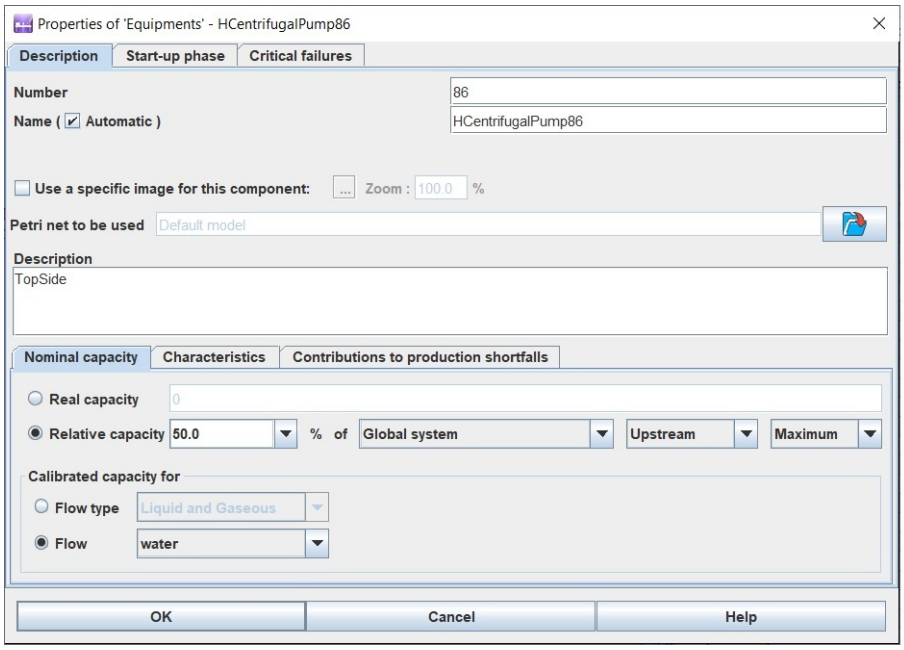
3. Horizontal centrifugal pump, vertical centrifugal pump, reciprocal pump, screw pump

Below are the components associated with rotating machines and their corresponding icons:

Icon	Code	Description
	RotatingMachine	Rotating machine generic
	GasTurbine	Gas turbine
	SteamTurbine	Steam turbine
	ElectricalMotor	Electric motor
	CombustionEngine	Combustion engine
	ElectricGenerator	Electric generator
	CentrifugalCompressor	Centrifugal compressor
	ScrewCompressor	Screw compressor
	ReciprocatingCompressor	Reciprocating compressor
	TurboExpander	Turbo-expander
	Blower	Blower
	ScrewPump	Screw pump
	HCentrifugalPump	Horizontal centrifugal pump
	VCentrifugalPump	Vertical centrifugal pump
	ReciprocatingPump	Reciprocating pump

All these components use the same Petri net and their configuration interface is therefore the same. Note the existence of a 15th type of rotating machine: generic rotating machines. These are used when a prototype is required that differs slightly from the original prototype. This will be discussed in greater detail in the section Section 4.7.1, “The generic component”

A rotating machine can be configured by double-clicking on the component or by a right click, then selecting the **Properties** menu.



By default, the behaviour of a rotating machine is quite simple: it is a component that has only one critical failure with a lambda of 1E-3 and an MTTR of 100.

4.2.1. Description

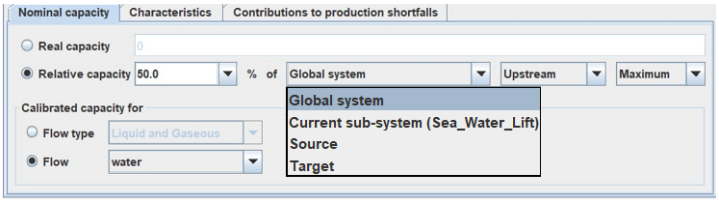
A **Number** and a **Name** for the component can be entered here. Tick the **Automatic** box to generate a name comprising a base name followed by the number.

Using a specific image for the component means that the component icon can be modified. Use the **Zoom>** to crop the image.

A description of the component can be entered in the **Description** field.

4.2.1.1. Nominal capacity

This screen offers the possibility of calibrating the capacity that the component is capable of processing at a given moment. Later on, you will see that the capacity can be changed in different situations. This screen deals with the capacity of the component in its nominal state. You can choose whether to indicate a **Real capacity** or a **Relative capacity**.



The choice of reference for the relative capacity depends on the location of the component in your model. If the component is within a sub-system S1, you can indicate that the capacity is relative to X% of what enters S1. The creation and configuration of sub-systems is described in in the sections Section 3.8, “Sub-systems creation” and Section 3.9, “Sub-system capacity”.

In a general case, the reference is the **Global system**. The relative capacity is then calculated from the well profiles of the **Upstream** branch or the demand profiles of the **Downstream** branch of the component.

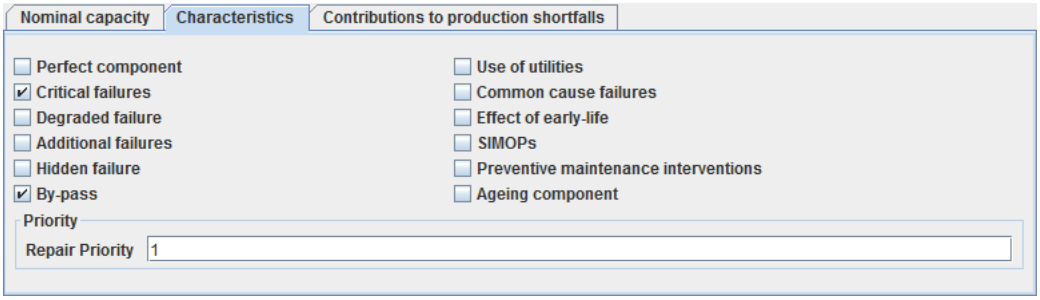
It is also possible to define the capacity by a specific source or target (Menu **Source** or **Target**). In these cases, the capacity will follow the profiles indicated on the source or target.

Two capacity calculation strategies have been defined (Menu **Maximum** or **Dynamic**). With the **Maximum** strategy, the relative capacity is pre-calculated statically, by summing all the profiles considered. The value retained is the maximum over time of the "sum" profiles. This strategy is interesting for macroscopically defining the capacity of the component. With the **Dynamic** strategy, relative capacity is a variable expression of the current values of profiles. The nominal capacity of the component is then variable over time.

The use of this capacity can be restricted to a specific **Flow** or **Stream type**. The capacity of the other flows will be adjusted automatically, in proportion to what flows through the component. For example, if a real capacity of 100 is defined, calibrated on the "oil" flow, and if a flow vector of (200, 100, 50) circulates with (oil, gas, water), then the restriction (100, 50, 25) will be obtained at the component output.

Note that the real capacity can be variable.

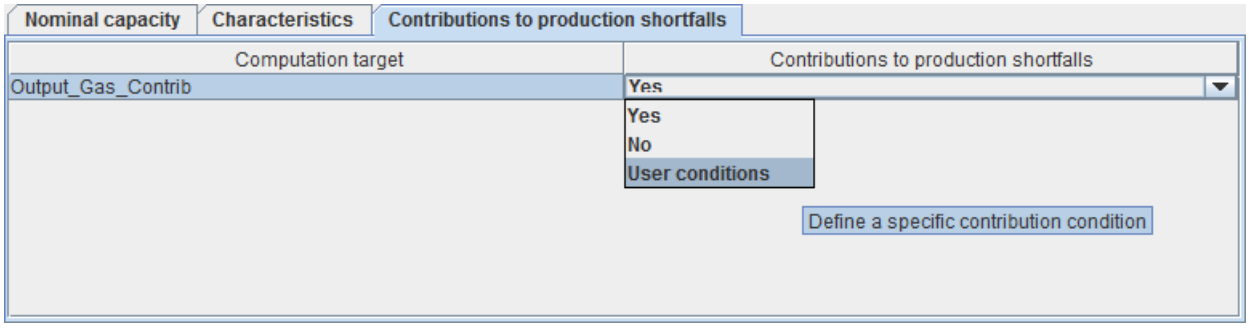
4.2.1.2. Characteristics



The behaviours of the rotating machine can be selected on this screen. The characteristic **critical failures** is selected by default. Below is a list of the possible characteristics and their effects:

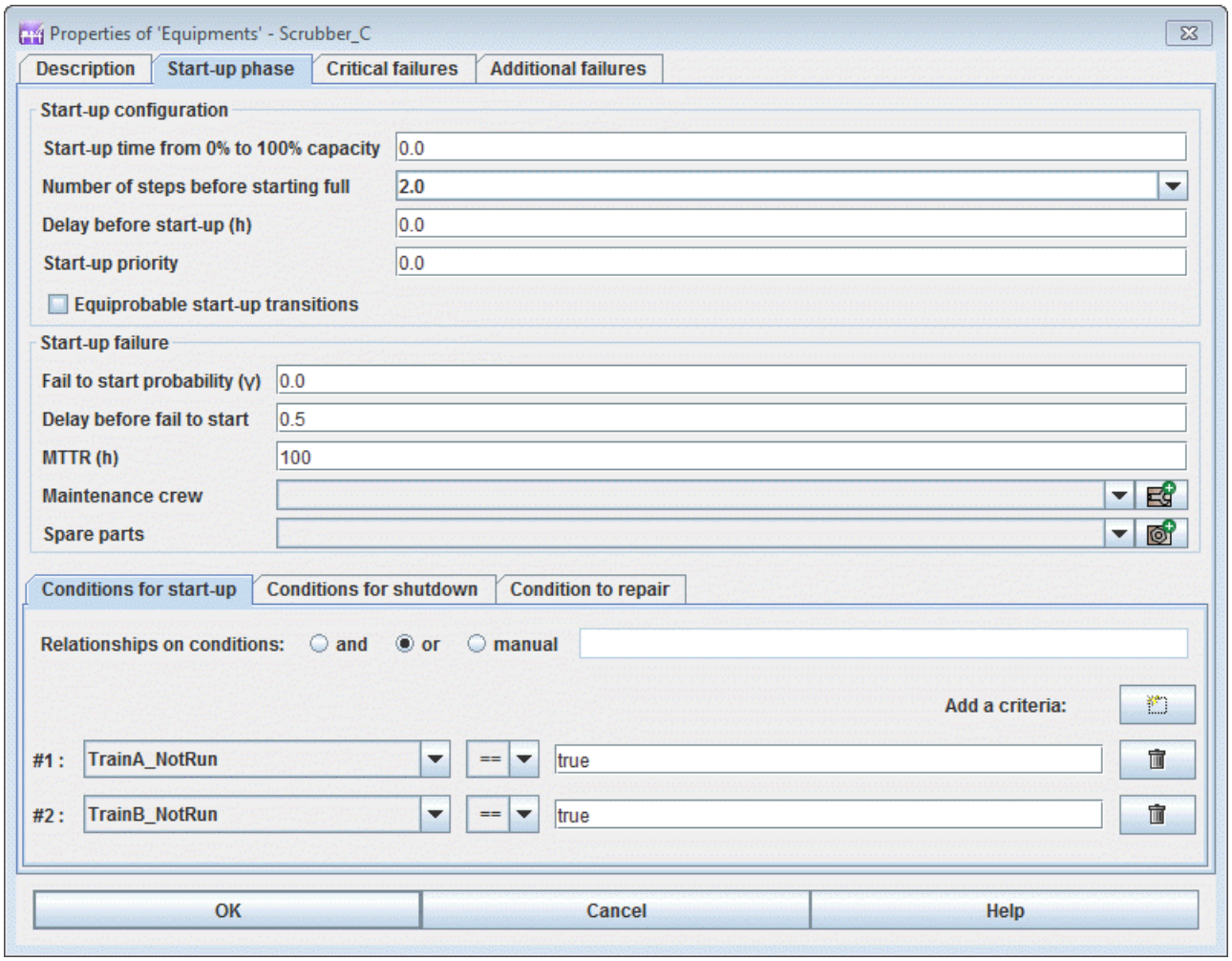
- **Perfect component**: the component cannot fail;
- **Critical failures**: up to six critical failures can be defined;
- **Degraded failures**: defines a degraded mode of the component which may become critical;
- **Additional failures**: defines failures;
- **Hidden failures**: adds failures Defined by Full-Periodically-Tested law;
- **By-pass**: programs the by-pass of a component in the event of a failure;
- **Use of utilities**: adds start-up conditions for the component according to the outputs on the utilities network;
- **Common cause failures**: the component is subject to common cause failures;
- **Effect of early life**: increases the failure rate of the component in early life;
- **SIMOPS**: defines groups of components that will be shut down in the event of an approach or intervention of a maintenance crew;
- **Preventive maintenance interventions**: defines maintenance periods and the conditions for carrying out maintenance interventions;
- **Ageing component** : critical and degraded failures uses Weibull law, repairs are not necessarily As Good As New;
- **Repair priority**: manage repair priority for repair teams. Repair team repair in priority failures with hight priority.

4.2.1.3. Contributions to production shortfalls



This screen is used to enter a condition for the contribution of the component to production loss. By default, all components contribute to loss. Adding a condition restricts the contribution of the component. Note that the contributions to loss will be calculated only if they are requested in the model configuration (Section 3.2, “Flows configuration”).

4.2.2. Start-up



The first part enables to configure the start-up of a component.

- **Start-up time from 0% to 100% capacity (Time before full start (hrs))**: this is the time in hours that the component requires to move from zero to full capacity.
- **Number of steps before starting full**: models a notion of progressive start-up of the component.

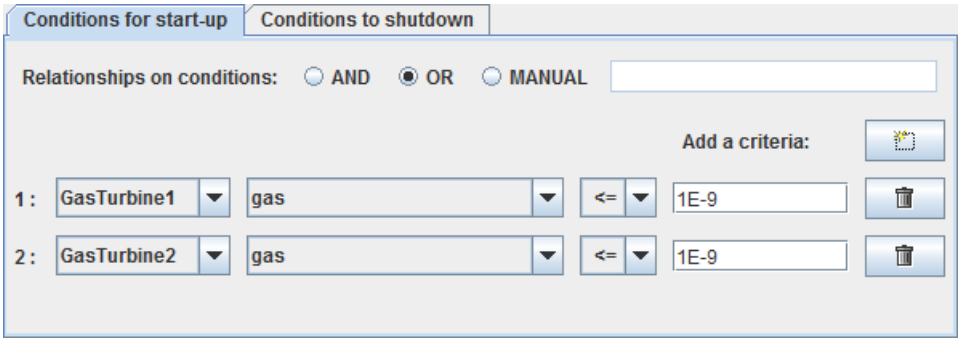
- **Delay before start-up (h)**: length of time as from which a shutdown component must factor in the **Start-up time from 0% to 100% capacity**. If the component is restarted before this time, it will start immediately at 100% capacity.

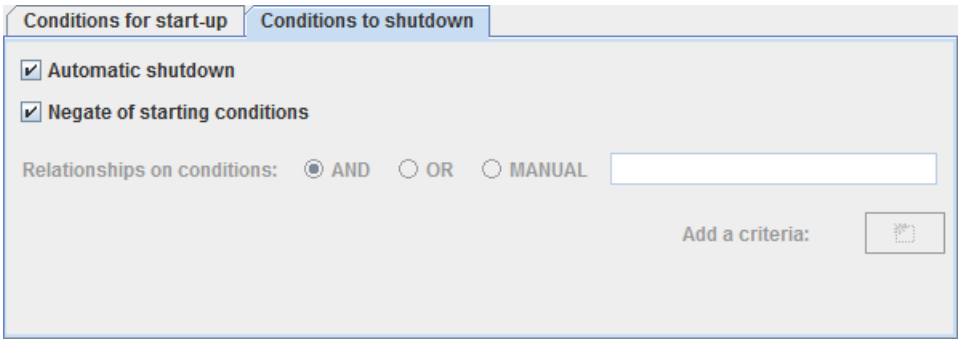
  Must be zero if **Start-up time from 0% to 100% capacity** is zero.

- **Start-up priority**: to define the start-up priority.
- If **Equiprobable start-up transitions** is checked all the start-up transitions will be fired with an equiprobable way. Otherwise creation order will be taken into account to start equipments.

The second part is for Start-up failure:

- **Fail to start probability (γ)**: probability that the component will fail to start each time it is started or restarted.
- **Delay before fail to start (h)**: inactivity time beyond which the \u03B3 is factored in when the component is restarted.
- **MTTR (h)**: Mean Time To Repair the component for a start-up failure.
- **Maintenance crew**: used to assign a maintenance crew to repair the start-up failure.
- **Spare parts**: allocates spare parts for start-up failures.
- **Conditions for start up** and **Conditions for shutdown**:

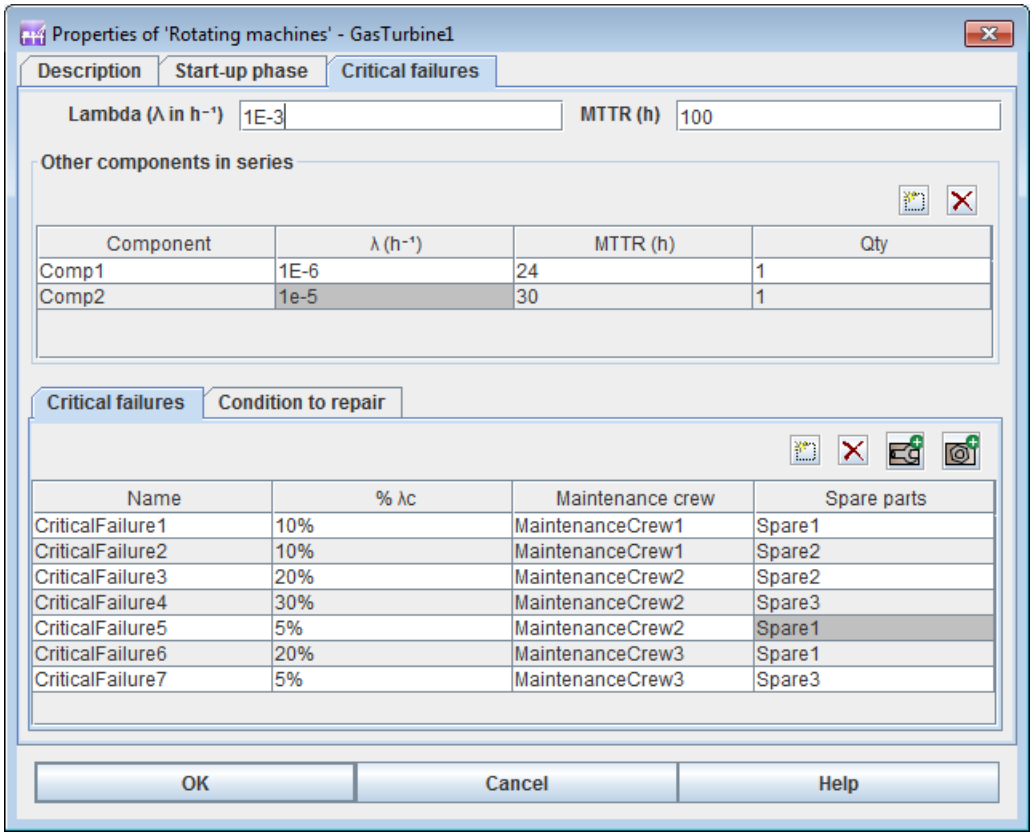




The **Conditions for start up** and **Conditions for shutdown** tabs are used to define the rules for starting up and shutting down a component. By default, there are no rules. The component starts up automatically as soon as it is affected by a flow request. It will shut down automatically if no more flow is requested at its output. **Automatic shutdown** can be deactivated by unticking Automatic shutdown in the **Conditions for shutdown** tab.

- **Condition to repair** : enables you to define a rule for component repair after a start-up failure

4.2.3. Critical failures



The **Critical failures** tab is used to separate the critical failures into several parts. The equation settings can be entered in the upper part of the screen:

- **Lambda(λ)(h^{-1})**: the component failure rate per hour.
- **MTTR (h)**: Mean Time To Repair the component for its critical failures.

The **Other components in series** section can be used to add components assimilated to the component being edited. The following criteria can be entered:

- **Component**: the name of the component in series.
- **λ (h^{-1})**: component failure rate
- **MTTR (h)**: Mean Time To Repair component.
- **Qty**: the number of occurrences of this type of component.

These components will influence the **Lambda** and **MTTR** settings by the calculation of equivalents. Therefore, in the case below, the lambda equivalent used will not be 1E-3 but:

$$\lambda_{eq} = \lambda + \lambda_{C1} * qte_{C1} + \lambda_{C2} * qte_{C2}$$

$$\mu_{eq} = \lambda_{eq} / (\lambda * MTTR + \lambda_{C1} * MTTR_{C1} * qte_{C1} + \lambda_{C2} * MTTR_{C2} * qte_{C2})$$

In the lower part, critical failures can be defined by entering:

- **Name**: name of the critical failure

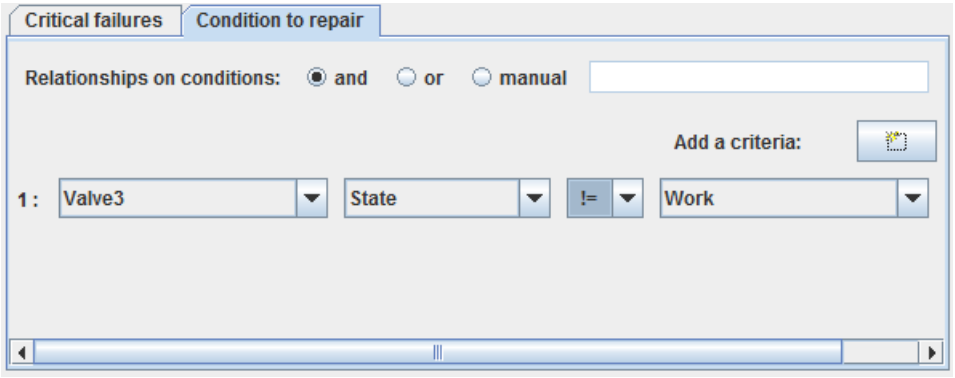
  This name appears during simulation to identify the different critical failures.

- **%λc**: the proportion of all critical failures that this critical failure represents.This percentage can be given in the form of variable expression or with parameters.

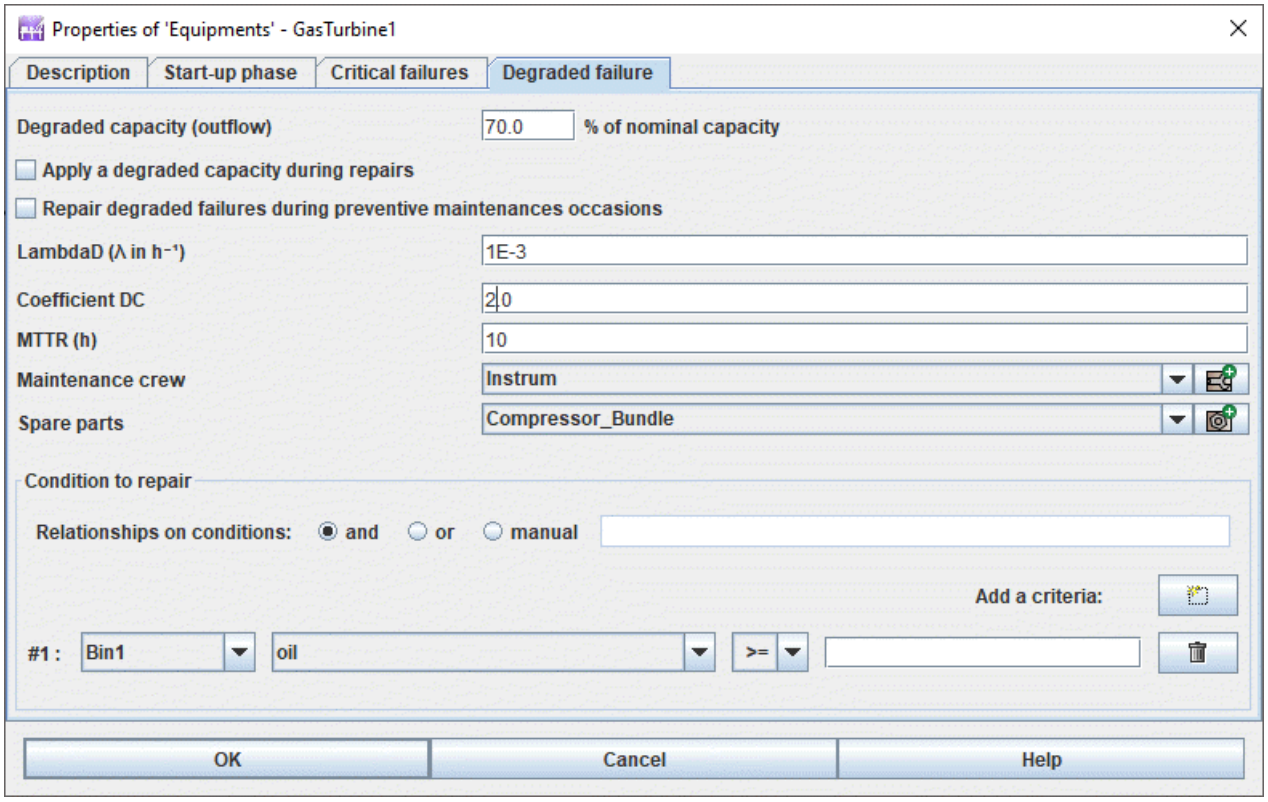
- **Maintenance crew**: the maintenance crew assigned to the failure. The button creates a new repair team.
- **Spare parts**: the spare parts allocated to the failure. The button creates a new spare part.

Note that the use of a maintenance crew or spare parts is optional.

- **Condition to repair**: used to define the rules for repair a component. By default, there are no rules. The component is repaired as soon as repair team and spare parts are available.



4.2.4. Degraded failures



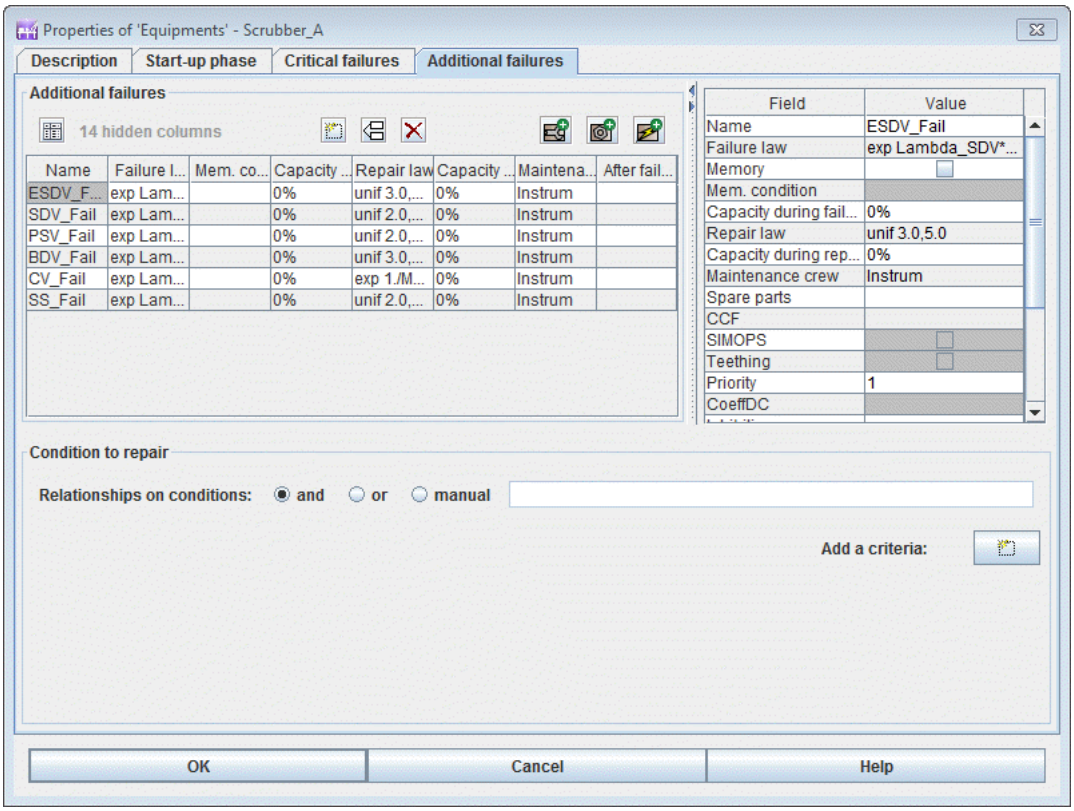
Degraded modes can be defined for components by completing the following fields:

- **Degraded capacity (outflow)**: percentage of the nominal capacity. Capacity that will be applied during the degraded mode.
- **Apply a degraded capacity during repairs**: indicates whether the capacity of the component during repairs is the degraded capacity. Otherwise, its capacity is set to 0.
- **Repair degraded failures during preventive maintenances occasions** : The component in default during a degraded failure will not be overhauled by the preventive maintenance without consuming the resources nor counting the cost of repair.

- **LambdaD (λ en h^{-1})**: failure rate of the degraded failure.
- **Coefficient DC**: coefficient to modify the failure rate in case of degraded failure.
  This coefficient will be applied to the critical failure rate and the critical additional failure rates (with zero capacity during repairs and with an exponential failure law).
  For aging components, the scale parameter (η) will be multiplied by $CoeffDC^{-1/\beta}$
- **MTTR (h)**: Mean Time To Repair a degraded failure.
- **Maintenance crew**: the maintenance crew assigned to the degraded failure.
- **Spare parts**: the spare parts allocated to the degraded failure.
- **Condition to repair**: condition before mobilisation of the maintenance crew.

4.2.5. Additional failures

If the exponential laws used in the critical failures and degraded failures tabs do not suit the case in hand, or if the user wishes to add other failures to the component, the **Additional failures** tab serves this purpose.



The additional failures are listed in the left table and the selection detail is displayed on the right. If multiple rows are selected, a change in the detail table will affect the entire selection.

Below, it is possible to enter a **Condition to repair** that affects all additional failures. This condition can be fine-tuned for each failure (column **Condition to repair**) and a logical AND is performed between the global repair condition and the fault repair condition.

 icon enables to choose the options to be display for the additional failures:

- **Name**: name of the additional failure.
- **Failure law**: failure law used for the additional failure.
- **Memory** : check the box to indicate if the failure transition is with memory.

- **Mem. condition** : enter a boolean expression that indicates under which criteria the remaining time must be stored. For more information on memory transitions, refer to User manual Moca14.10.3 [2.7.8. Transitions with memory]

- **Capacity during failure (%)**: percentage of nominal capacity. Capacity that will be applied during failure.

- **Repair law**: law used to define the mean repair time for the component.

- **Capacity during repair (%)**: percentage of nominal capacity. Capacity that will be applied during repair.

- **Maintenance crew**: the maintenance crew assigned to the failure.

- **Spare parts**: the spare parts allocated to the failure.

- **CCF** : Assigns a common cause failure to this failure

- **SIMOPS** : if failure is subjected to SIMOPS determined in SIMOPS characteristics.

- **Teething** : if failure is affected by effect of early life determined in Early life characteristics.

- **Priority** : define the repair priority for each additional failures.

- **CoeffDC** : if the capacity during repair in not 0 (degraded failure), this coefficient is applied to the failure rates of other critical additional failures (having zero capacity during repair) and having an exponential failure law, as well as the critical failures defined in the **Critical failures** tab.

- **Inhibition** : enables to enter a Boolean expression that indicates whether the failure is inhibit or not.

- **After failure assignment** : enables treatment when the failure occurs. This processing can be a simple variable assignment or a more complex code. For more information on the syntax and usable functions, refer to User manual Moca14.10.3 [2.7.4. Assignments]

- **DYN (Failure)** : (for advanced user) Failure transition is a dynamic transition.

- **After repair assignment** : enables treatment when the repair is complete.

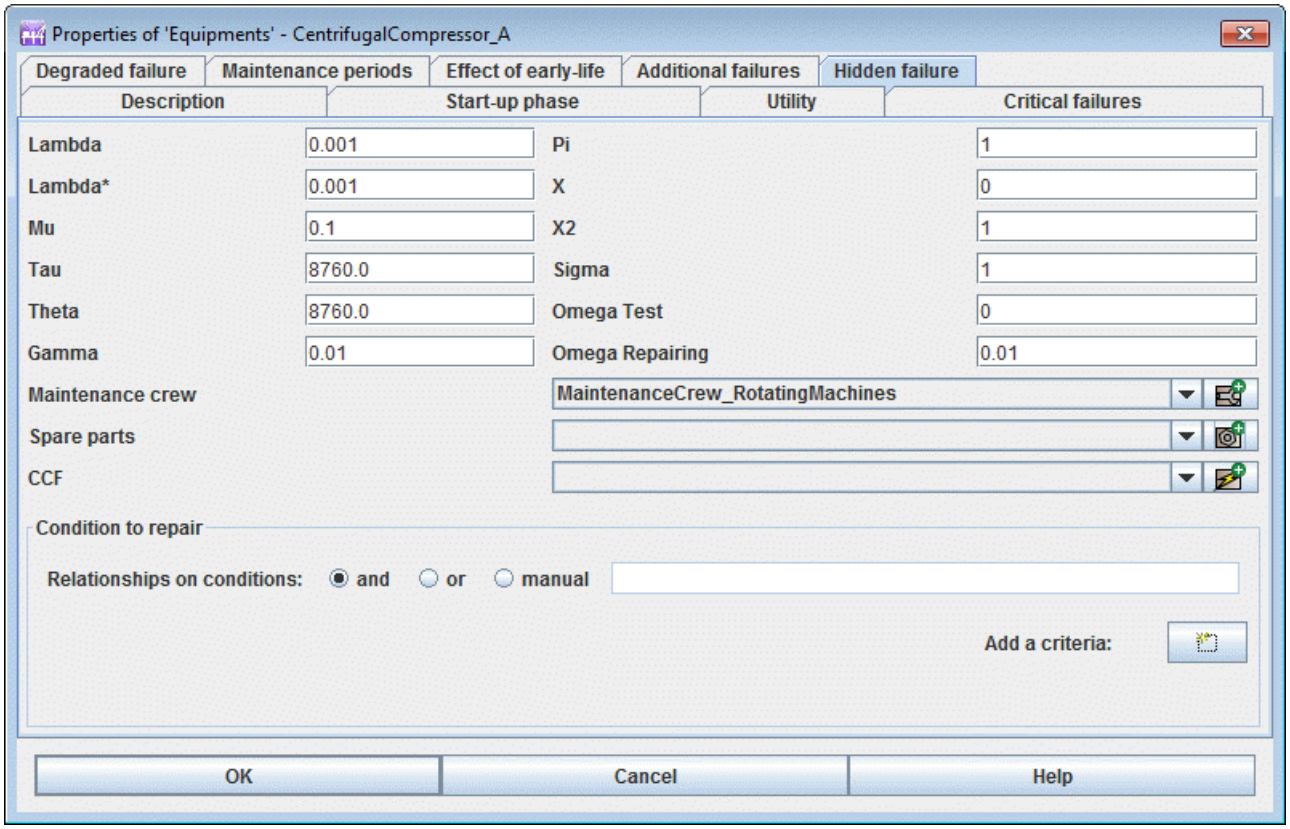
- **Condition to repair** : additional condition to repair failure.

- **DYN (Repair)** : (for advanced user) Repair transition is a dynamic transition.

- **SEQGEN** : check the box to indicate that the failure can be included in the list of transitions leading to the dreaded event (sequence generation calculations).

4.2.6. Hidden failure

This mode makes it possible to add a hidden failure defined by a complete periodic test law.



The following parameters are then available:

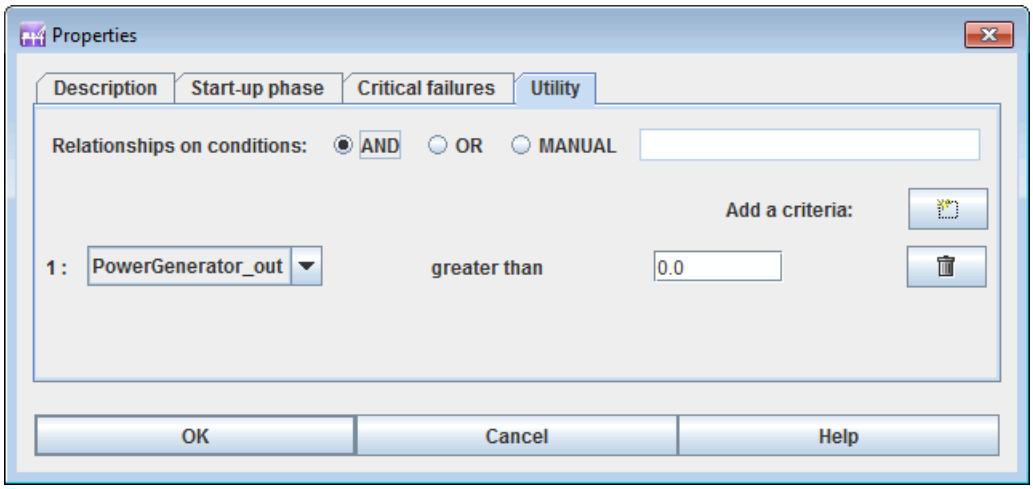
- **Lambda** : Failure rate in h^{-1}
- **Lambda*** : Failure rate during the test in h^{-1}
- **Mu** : Repair rate in h^{-1}, Mu = 1 / MTTR
- **Omega Test** : probability [0,1] of bad reconfiguration after the test. 0 = no problem of reconfiguration, the operator always puts the system back on properly.
- **Omega Repair** : probability [0,1] of bad reconfiguration after the repair. 0 = no problem of reconfiguration, the operator always puts the system back on properly.
- **Pi** : test duration in h. 0 means that the test is instantaneous.
- **Sigma** : Test coverage rate, probability [0,1] that the component failure is detected during the test. 1 = the test is perfect and covers all possible faults.
- **Tau** : Test period in h (time interval between two consecutive tests). For example 43800 h = 5 years.
- **Theta** : Date of the first test in h (often equal to the test period)
- **X** : Availability indicator during test {0,1}. 0 => component not available, 1 => component available.
- **X2** : Availability indicator for production during hidden failure {0,1}. 0 => component not available, 1 => component available.
- **Spare parts** : Stock of spare parts to be used for hidden failure

- **Maintenance crew** : Maintenance crew assigned for hidden failure
- **CCF** : Common cause failures for hidden failures
- **Repair condition** : Hidden failure repair condition

4.2.7. Perfect component

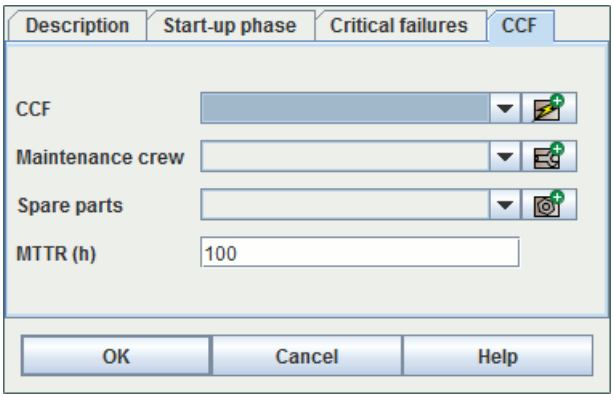
A **Perfect** component is one that has no failures. All the characteristics of the component concerning failures become inaccessible. For example, this type of component can be used for a valve that would open and close depending on external conditions or events.

4.2.8. Utility



In this tab, a criterion that affects the utility network output can be entered. If the criterion is not valid, the component shuts down.

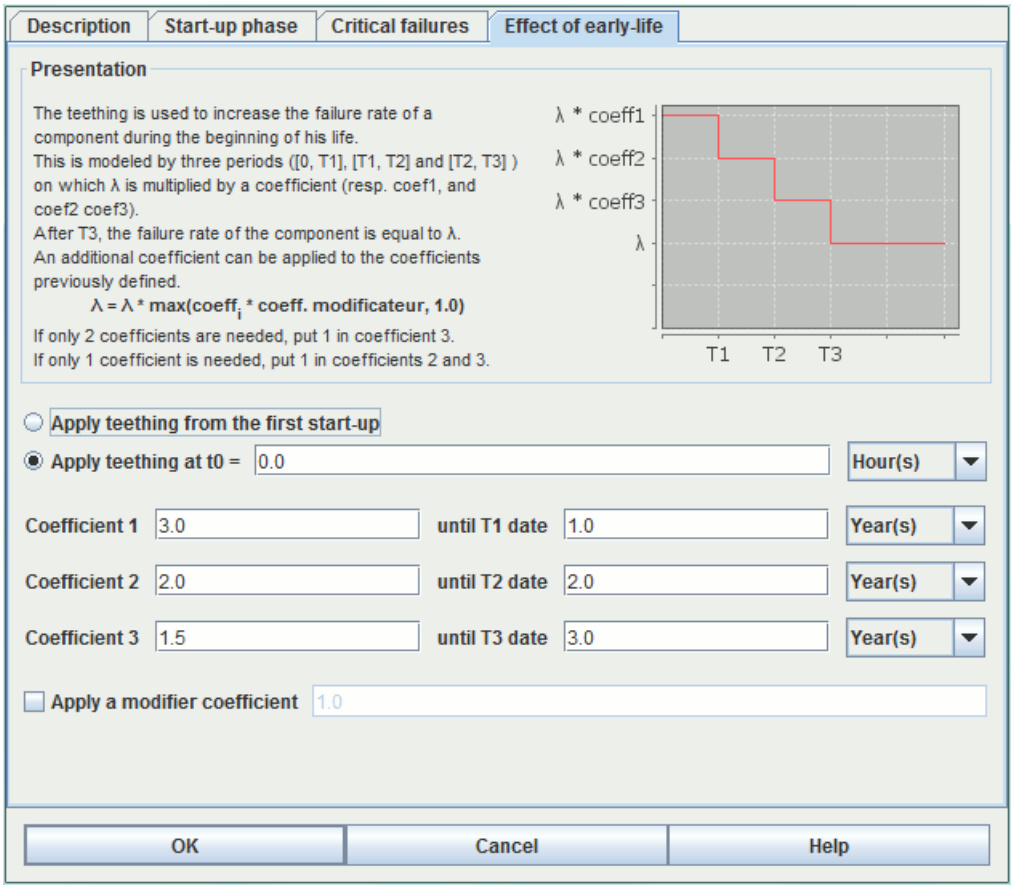
4.2.9. Common cause failure



Common cause failures (CCF) for the component can be added under this tab. The **Maintenance crew** and **Spare parts** fields can be used to allocate the maintenance crew and spare parts for the component for the CCF.

Repair time can be specified in the **MTTR (h)** field.

4.2.10. Effect of early life



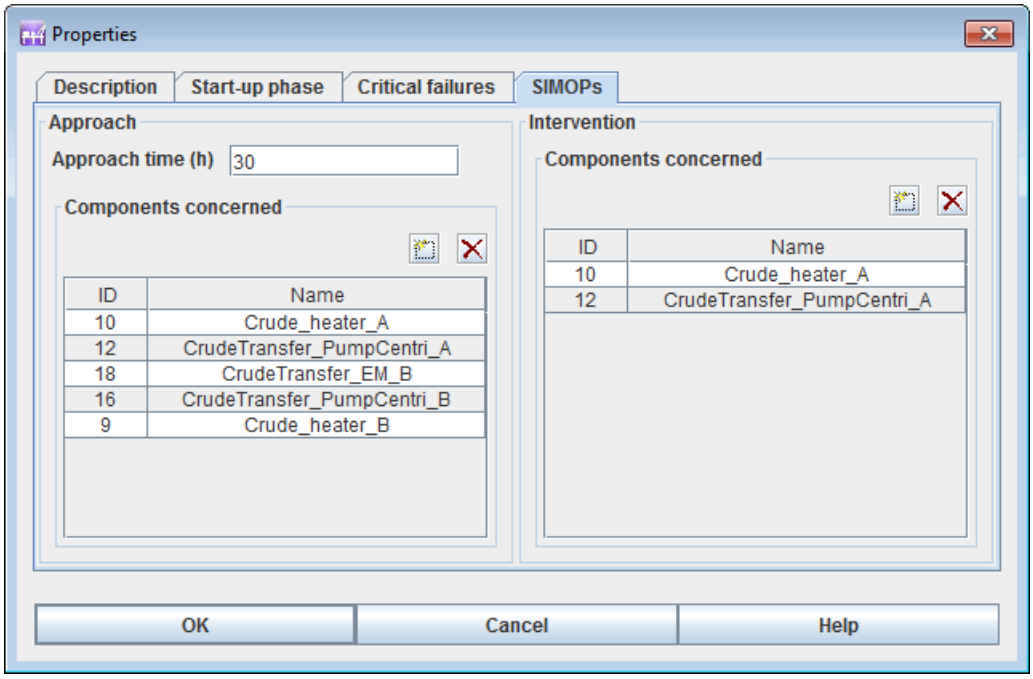
The effects of early life are used to increase the failure rate of a component during its early life. This is modelled in 3 periods ([0, T1], [T1, T2] and [T2, T3]) in which λ is multiplied by a coefficient (resp. coef1, coef2 and coef3). After T3, the component failure rate is equal to λ.

Following options can be checked:

- **Apply teething from the first start-up** allow to consider that t0 for the teething problem is when component starts and not the beginning of the simulation.
- **Apply teething at t0** moves the t0 for all components.
- **Apply a modifier coefficient** to all the coefficients entered previously

If this option is checked the coefficients which the value is 1 will also be modified.

4.2.11. SIMOPs



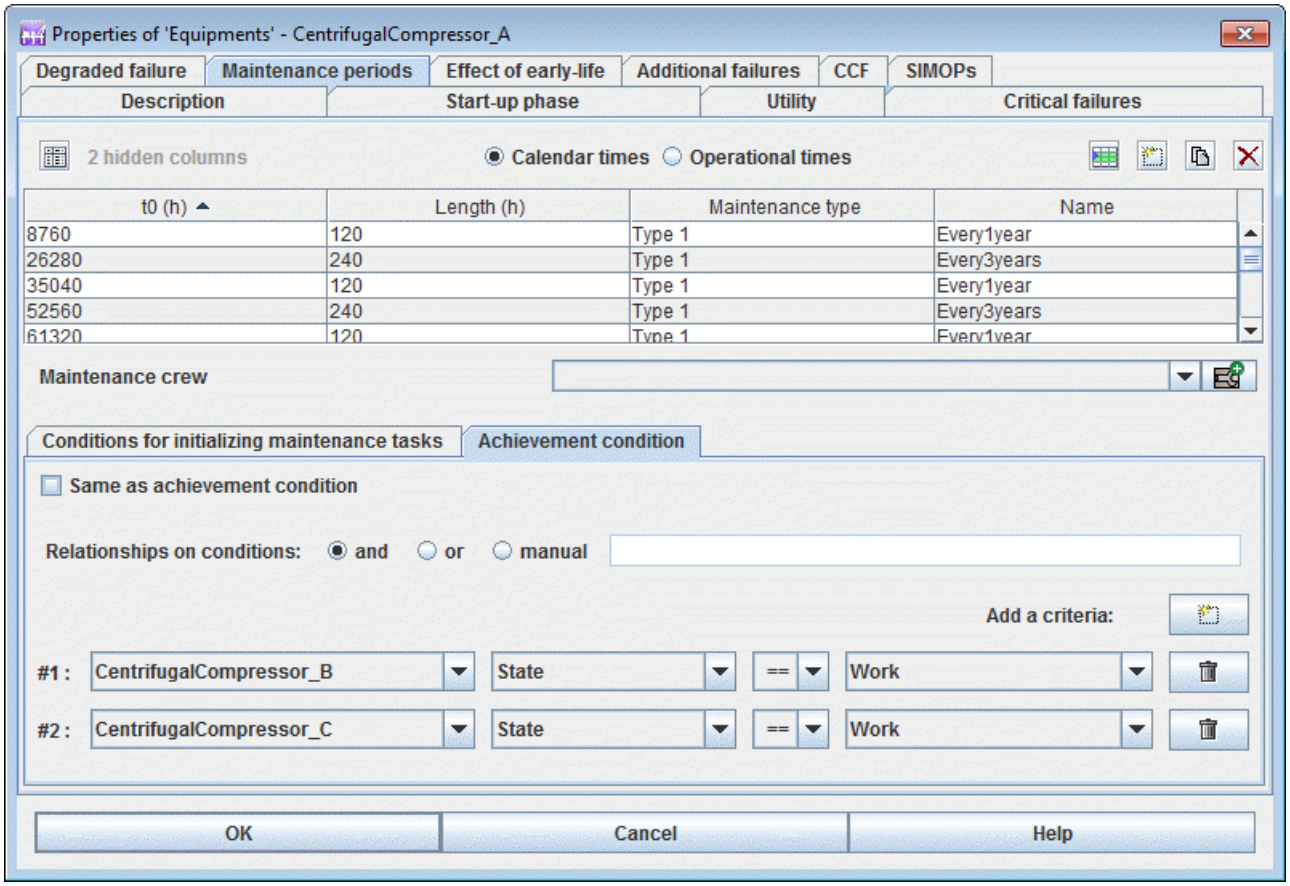
SIMOPs (SIMultaneous OPerations) are used to configure the shutdown of a set of components for maintenance operations. For example, when a component fails, it may be necessary to shut down a component so that the maintenance crew can intervene.

SIMOPs are used to define the components concerned during the maintenance crewâ##s approach phase.

The **SIMOPs Approach time (h)** is the time for which the components are shut down owing to the proximity of a maintenance crew.

The list of components concerned and listed in the **Approach** section are the components that will be shut down during this phase. The list of components affected in the **Intervention** section includes the components that will be shut down while the failed component is being repaired.

4.2.12. Maintenance periods



This tab is used to define preventive **Maintenance periods** for the component. They can be defined as **Calendar times**, in which case the dates are absolute as from t=0, or in **Operational times**, in which case the maintenance dates correspond to an operational duration. This avoids frequent maintenance operations on components that are regularly on stand-by.

	**Edit several**: user can modify one or more selected periods at the same time.
	**New**: user can add new periods to the component. The add period window is explained in the section Section 3.11.2, “New periods”.
	**Import periods from another component**: user can import periods from another component.
	**Delete**: deletes selected periods.

Repair team can be selected or added using the button  .

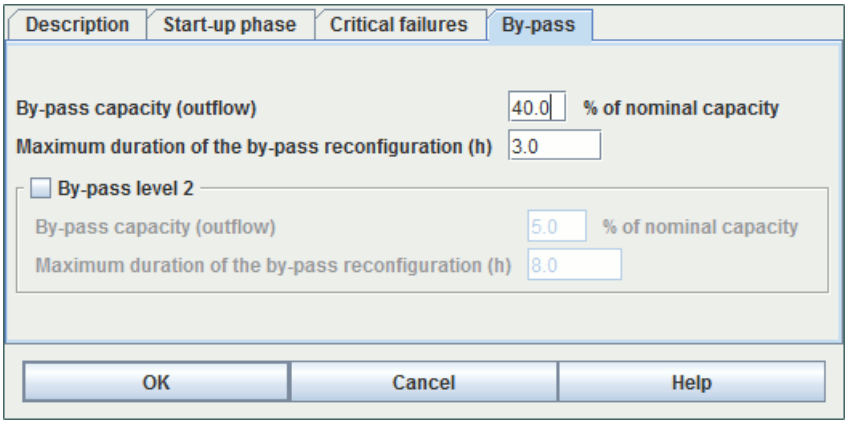
The lower part of the tab is used to enter conditions on the maintenance of the component.

- **Condition for initializing maintenance tasks** : Condition triggering the component maintenance process. When a maintenance is planned and the triggering condition is true, Resources are being mobilized, the component is not yet in the PM state. The PM_init variable is true.

- **Achievement condition** : Condition to perform maintenance (and change state to PM) when resources are available. When the component change to the PM state, PM_Init is set to false.

- **Same as initializing condition** : The achievement condition is the same as initializing condition.

4.2.13. By-Pass



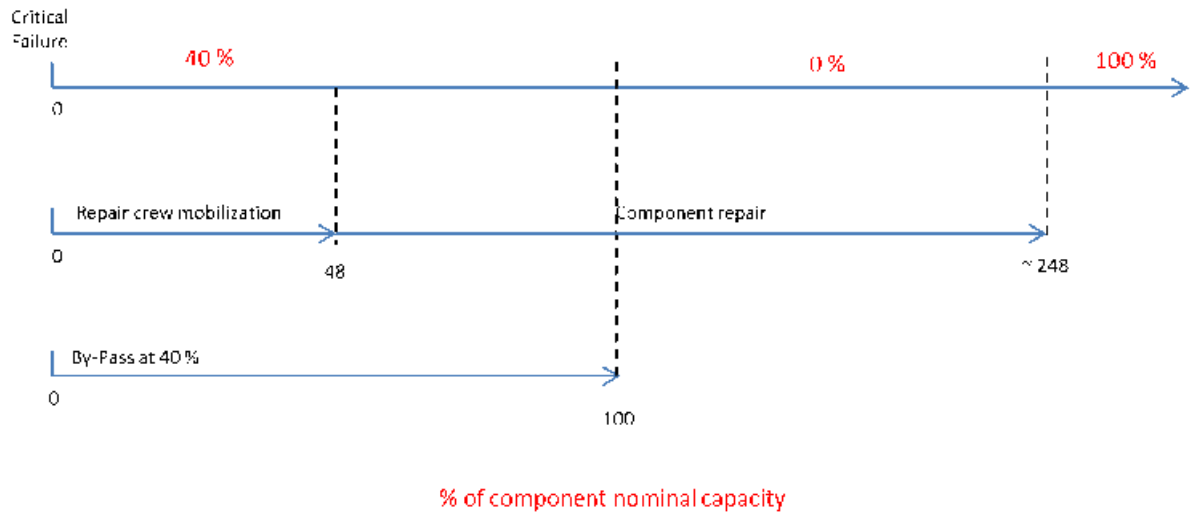
By-pass is used to by-pass a component when a failure occurse. The **By-pass capacity (outflow)** field is used to give a new capacity to the component while it is being by-passed. This value is a percentage of the nominal capacity. The **Maximum duration of the by-pass configuration** is the time in hours for which the component will remain in by-pass mode.

N.B. The capacity of the by-pass mode is only really used when the capacity of the component should be zero.

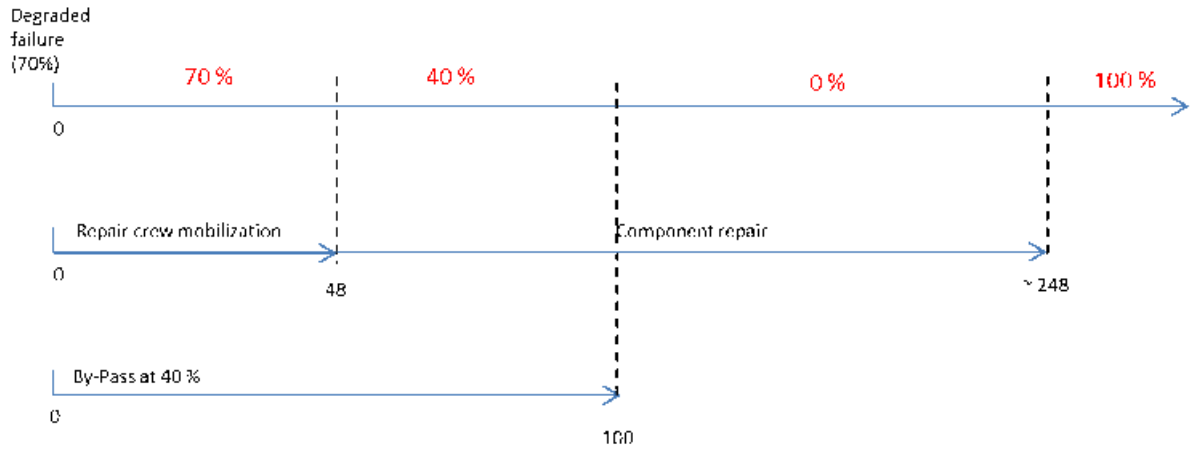
Below are different scenarios that may occur on a component that has a:

- Degraded failure
- Critical failure
- By-pass
- MTTR of 500 hrs
- Maintenance crew with a mobilisation time of 48 hours
- Degraded capacity at 70% of its nominal capacity
- A by-pass capacity at 40% of its nominal capacity
- A by-pass duration of 100 hrs.

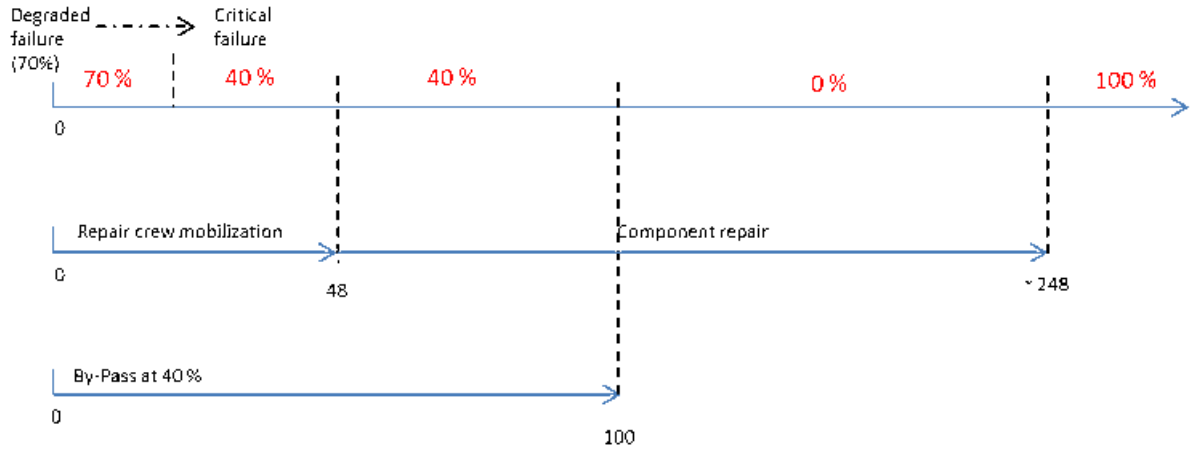
In the first scenario, the component suffers a critical failure, the maintenance crew is contacted immediately and the by-pass is activated. For the entire duration of the by-pass, the capacity of the component is not zero, but effectively 40%.



In the second scenario, the component suffers a degraded failure, the maintenance crew is contacted immediately and the by-pass is activated. The component can use its degraded capacity (70%) for the entire time the crew is mobilised. When the component is being repaired and is shut down, the capacity used is the by-pass capacity (40%).



In the final scenario, the component suffers a degraded failure, the maintenance crew is contacted immediately and the by-pass is activated. Then after a certain time, the component switches from degraded mode to critical failure. Here again, even when the capacity of the component should be zero, the by-pass means that a capacity of 40% can be achieved.



4.3. Static machines

There are currently 14 different types of static machines which are grouped into 3 categories in the toolbar:

1. Air cooler, electrical heater, plate heat exchanger, tube heat exchanger, boiler, electrical boiler

2. Filter, air filter, horizontal vessel/separator, vertical vessel/column, hydrocyclone.

3. Valve, control valve, choke valve.

Below are the components associated with static machines and their corresponding icons:

Icône	Code	Description
	StaticMachine	Machine statique générique
	ShellTubeHeatExchanger	Echangeur de chaleur à tube
	ElectricalHeater	Réchauffeur électrique
	PlateHeatExchanger	Echangeur de chaleur à plaque
	AirCooler	Refroidisseur à air
	Boiler	Chaudière
	ElectricalBoiler	Chaudière électrique
	VerticalVessel	Citerne verticale/colonne
	HorizontalVessel	Citerne horizontale/séparateur
	HydroCyclone	Hydrocyclone
	AirFilter	Filtre à air
	Filter	Filtre
	ChokeValve	Soupape d'étranglement
	ControlValve	Vanne de régulation
	Valve	Vanne

All these components use the same Petri net and their configuration interface is therefore the same. Note the existence of a 15th type of static machine: generic static machines. These are used when a prototype is required that differs slightly from the original prototype. This will be discussed in greater detail in the section Section 4.7.1, “The generic component”

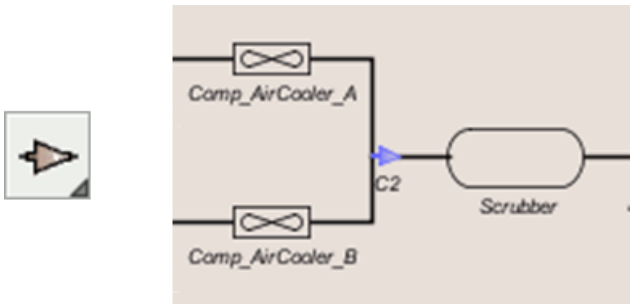
A static machine can be configured by double-clicking on the component or by a right click, then selecting the **Properties** menu.

Properties and configuration are identical to rotating machines.

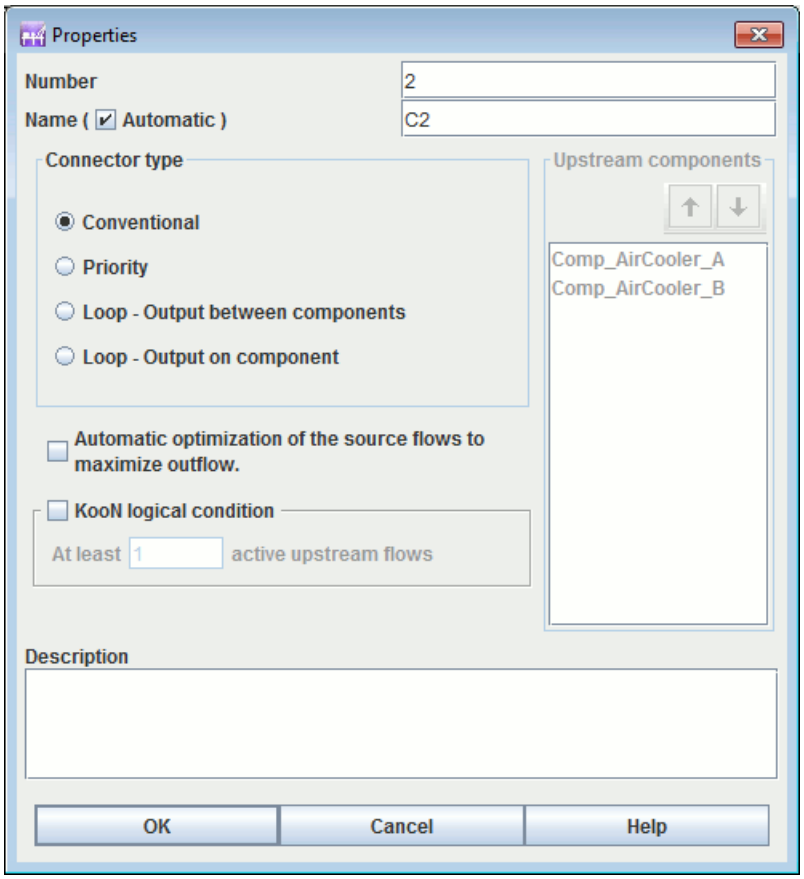
4.4. Connectors

4.4.1. Convergent connectors

These are used to run several branches of a network into a single branch.



A convergent connector can be configured by double-clicking on the component or by right-clicking and selecting the **Properties** network.

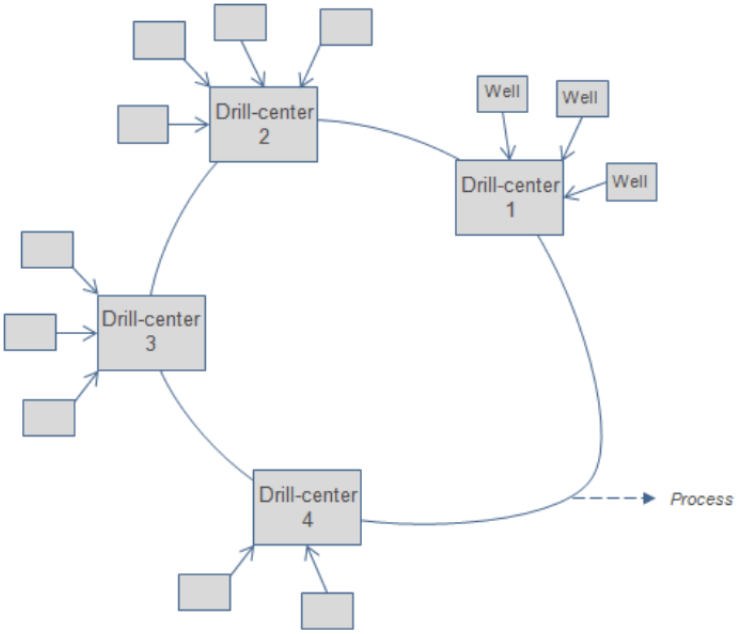


A **Number** and a **Name** can be entered here. Tick the **Automatic** box to generate a name comprising a base name followed by the number.

There are 4 different **types of connector**:

- **Conventional**: adds up all the input flows. The proportion taken from each branch may vary if the Automatic optimization of the source flows to maximize outflow box is ticked.
- **Priority**: runs priority consumption according to the input order. The priority ranking respects the order in the Upstream components table.
- **Loop – output between components**: models a component loop in which the output is located between two components. The connector output will be located on the loop between the first and last components in the **Upstream components** table.

In the case of Drill-Centers for example:

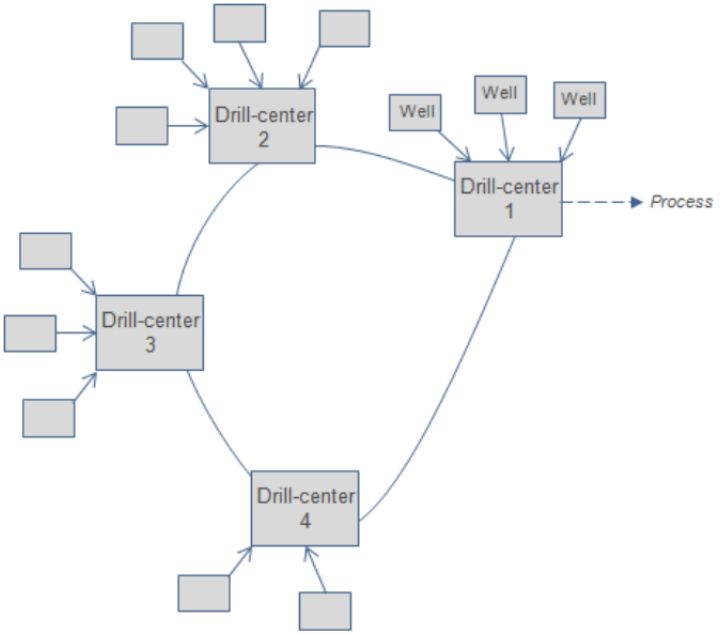


Make sure that the following order is respected when modelling the diagram above:

1. Drill-Center1
2. Drill-Center2
3. Drill-Center3
4. Drill-Center4

- **Loop – output on component**: models a component loop in which the output is located on a component. The connector output is located on the first component in the **Upstream components** table.

  In the case of Drill-Centers for example:



  Make sure that one of the following orders is respected when modelling the diagram above:

1.	Drill-Center1	or	1.	Drill-Center1
2.	Drill-Center2		2.	Drill-Center4
3.	Drill-Center3		3.	Drill-Center3
4.	Drill-Center4		4.	Drill-Center2

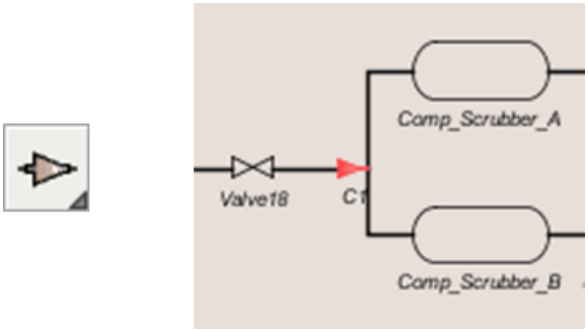
When the tick box **Automatic optimization of the source flows to maximize outflow** is ticked, it optimizes the management of the different flows to privilege the export of the top priority flow. For memory, the top priority flow is the first flow defined in the model configuration (see Section 3.2, “Flows configuration”)

The **KooN logical condition** section is available only for **conventional** connectors. It does not allow connector output flows to be propagated unless there are at least K upstream branches active.

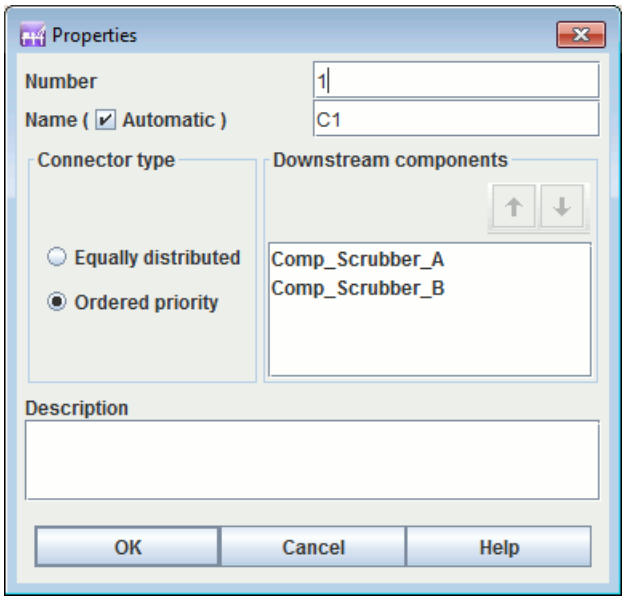
A description of the connector can be entered in the **Description** field.

4.4.2. Divergent connectors

These are used to split one branch of a network into several different branches.



A divergent connector can be configured by double-clicking on the component or by right-clicking and selecting the **Properties** network.



A **Number** and a **Name** can be entered here. Tick the **Automatic** box to generate a name comprising a base name followed by the number.

There are 2 different types of divergent connector:

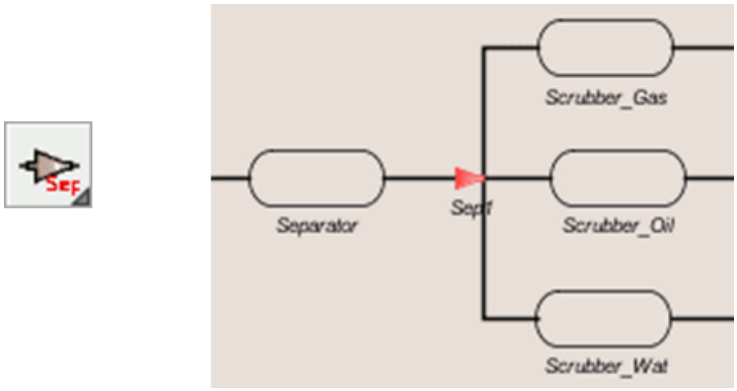
- **Equally distributed**: the flows are equally distributed.
- **Ordered priority**: the flows are distributed in priority according to the output order. The priority ranking follows the order in the **Downstream components** table. The first branch will receive the maximum possible flow, then the rest will flow into the second, etc.

A description of the component can be entered in the **Description** field.

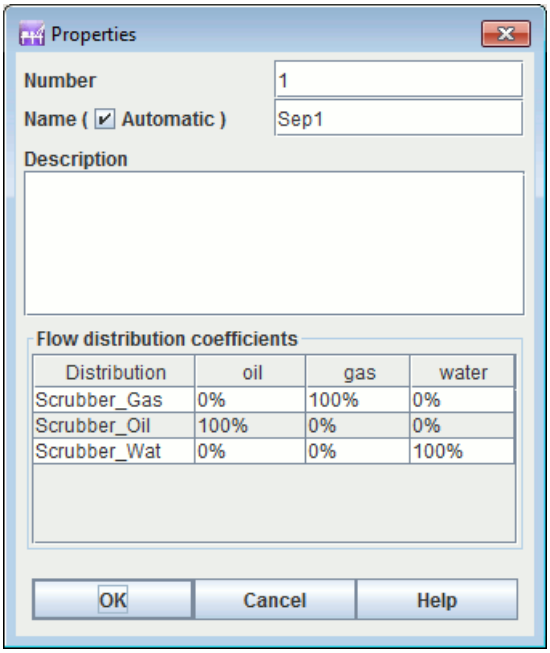
Warning: a divergent connector **Ordered priority** must be associated with a priority convergent connector.

4.4.3. The connector-separator

This separates the flows from one branch into several other branches.



A connector-separator can be configured by double-clicking on the component or by right-clicking and selecting the **Properties** network.



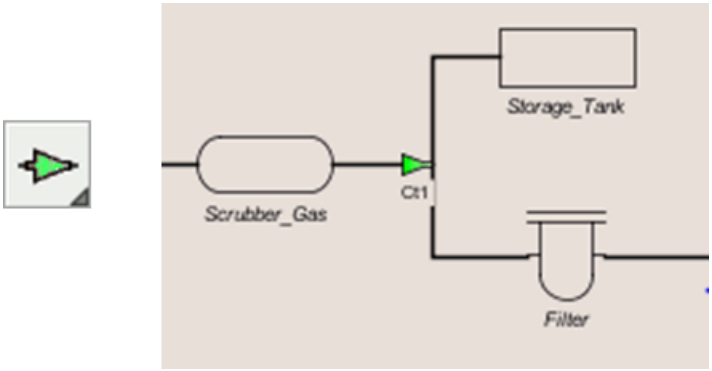
A **Number** and a **Name** can be entered here. Tick the **Automatic** box to generate a name comprising a base name followed by the number.

A description of the connector can be entered in the **Description** field.

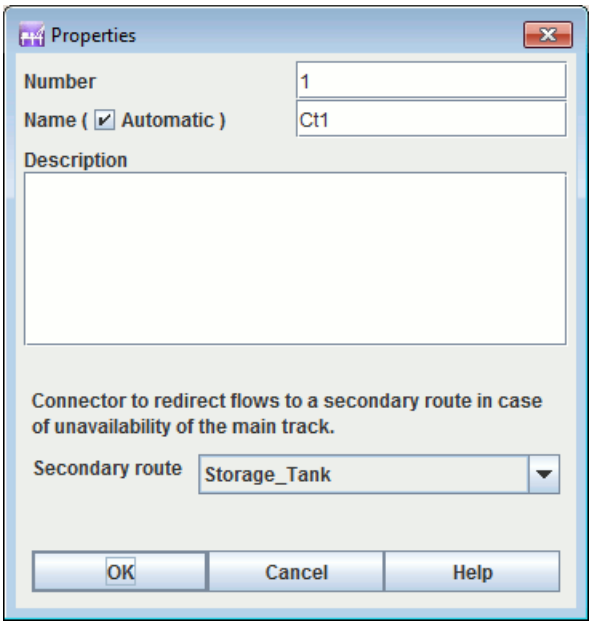
The **Flow distribution coefficients** table indicates how and in what proportions the input flows are to be separated. The left-hand column **Distribution** lists the components downstream of the connector. Then there is a column corresponding to each flow in the model for which the total sum of the coefficients must be equal to 100% (there can be no loss of flow).

4.4.4. The flare connector or discharge connector

This redirects flow to a secondary route if the main one is unavailable. This connector can have only one input and only two outputs. Note that the flare connector constantly sends an epsilon flow into the secondary route.



The flare connector can be configured by double-clicking on the component or by a right click, then selecting the **Properties** menu.



A **Number** and a **Name** can be entered here. Tick the **Automatic** box to generate a name comprising a base name followed by the number.

A description of the connector can be entered in the **Description** field.

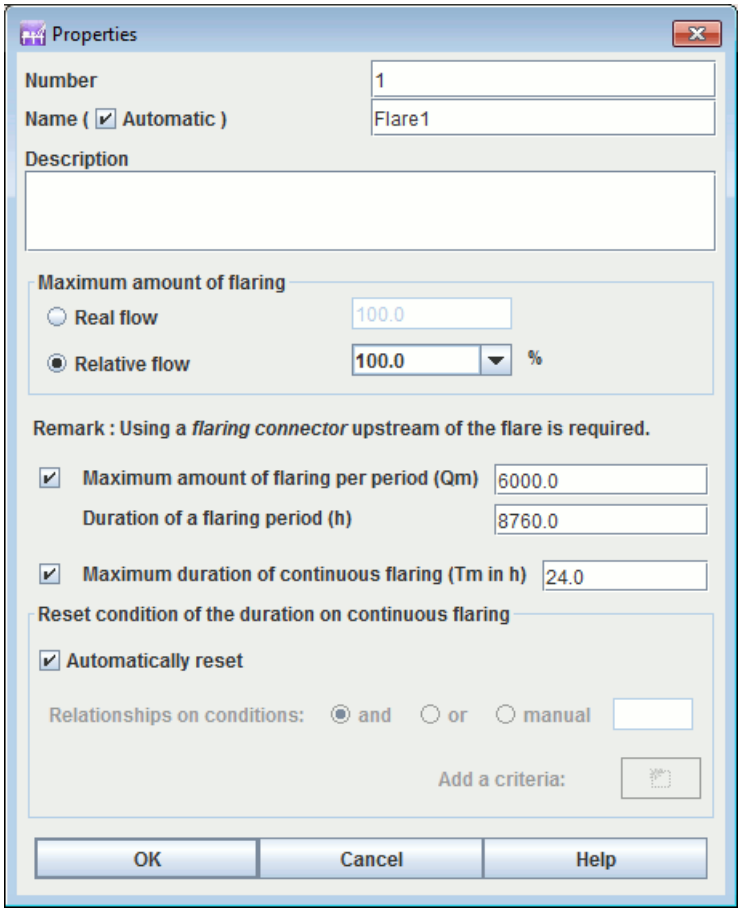
The **secondary route** indicates the branch to be used if the main one is unavailable.

4.5. Other specific components

4.5.1. The flare

This component is a network output and is used to consume input flows provided that certain flaring policies are respected. The flare is usually used after a **flare connector** but this is not mandatory. The theoretical request for flows to be flared is infinite.

The flare can be configured by double-clicking on the component or by a right click, then selecting the **Properties** menu.



A **Number** and a **Name** can be entered here. Tick the **Automatic** box to generate a name comprising a base name followed by the number.

A description of the component can be entered in the **Description** field.

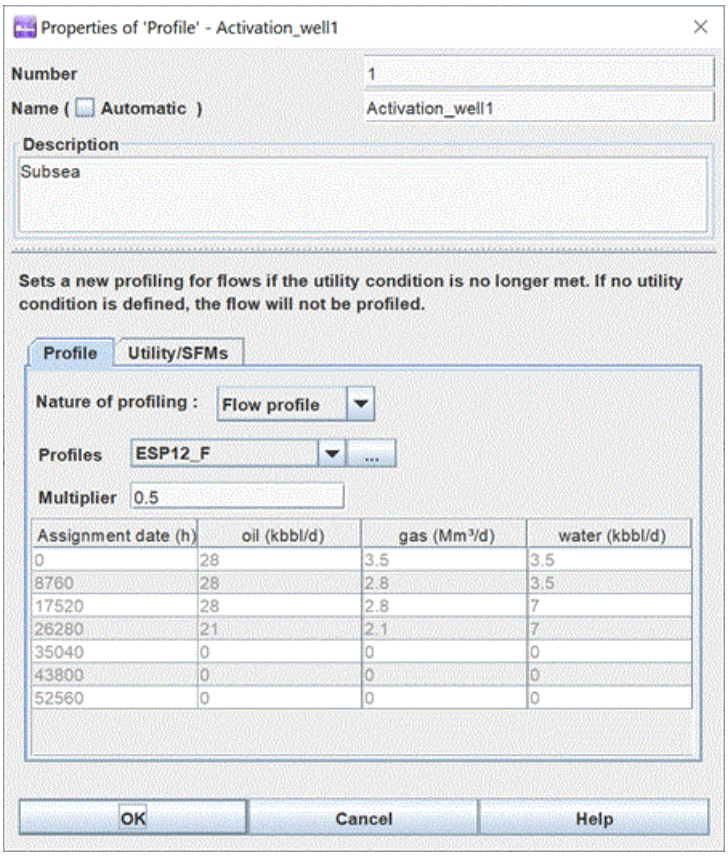
The **Maximum amount of flaring** defines the flare capacity.

The **maximum amount of flaring per period** is used to restrict the use of the flare for a certain quantity of flow and for the time defined in the **Duration of a flaring period**.

The **maximum duration of continuous flaring** is the duration in hours as from which the flare can no longer burn. If this duration is exceeded, the flare switches into critical failure mode. The duration is zeroed as soon as the main branch becomes available again.

4.5.2. Profiling block

This is used to define a new profile for the flows if the utility condition is no longer respected. If no utility condition is defined, the flows will not be profiled.



A **Number** and a **Name** can be entered here. Tick the **Automatic** box to generate a name comprising a base name followed by the number.

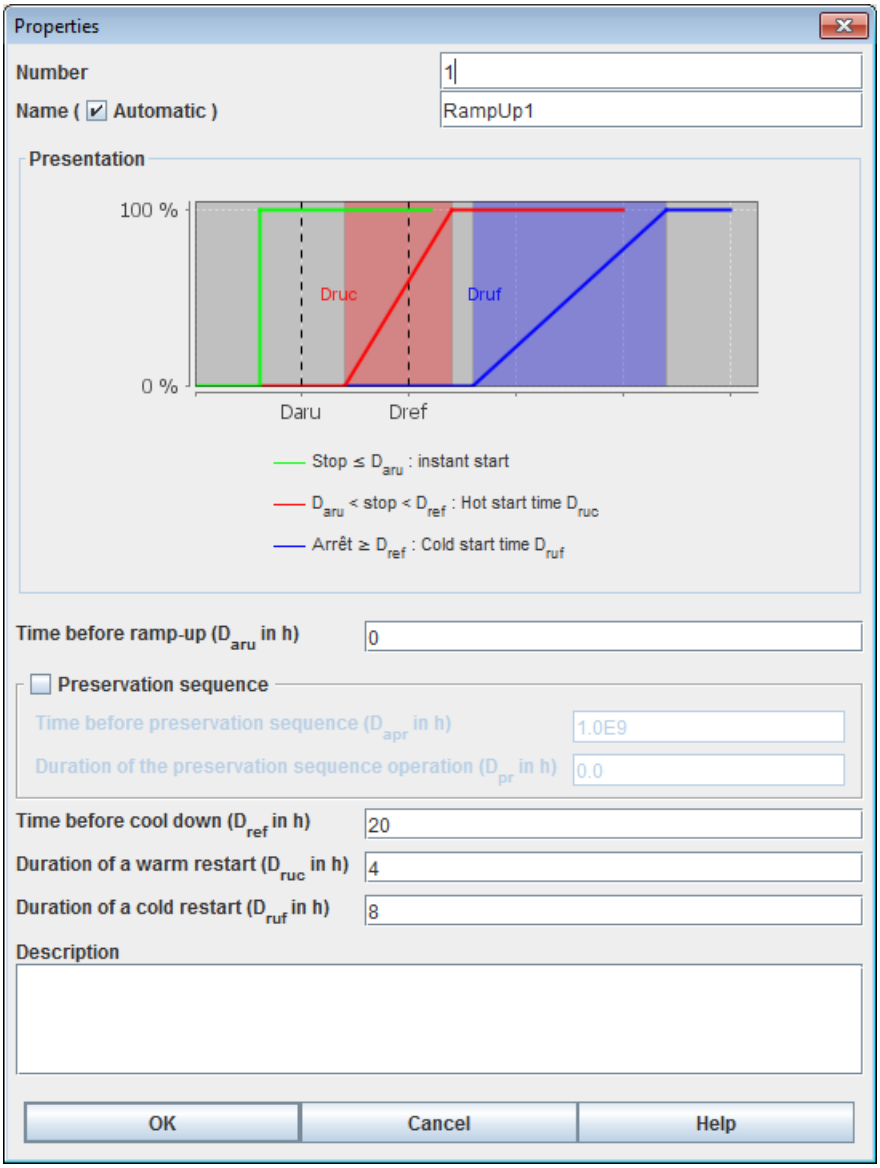
A description of the component can be entered in the **Description** field.

The **Profile** tab defines the profile to be used. The "Multiplier" case enables to apply coefficient on all selected profil values, which limits edition of multiple profils.

The **Utility** tab is used for entering the utility condition.

4.5.3. Ramp-up block

Ramp-Up is the progressive start-up of all or part of an installation. When the system is shut down for long enough after a specific event, e.g. a shutdown, the installation requires a transition period during which production will gradually increase until it reaches its nominal value.



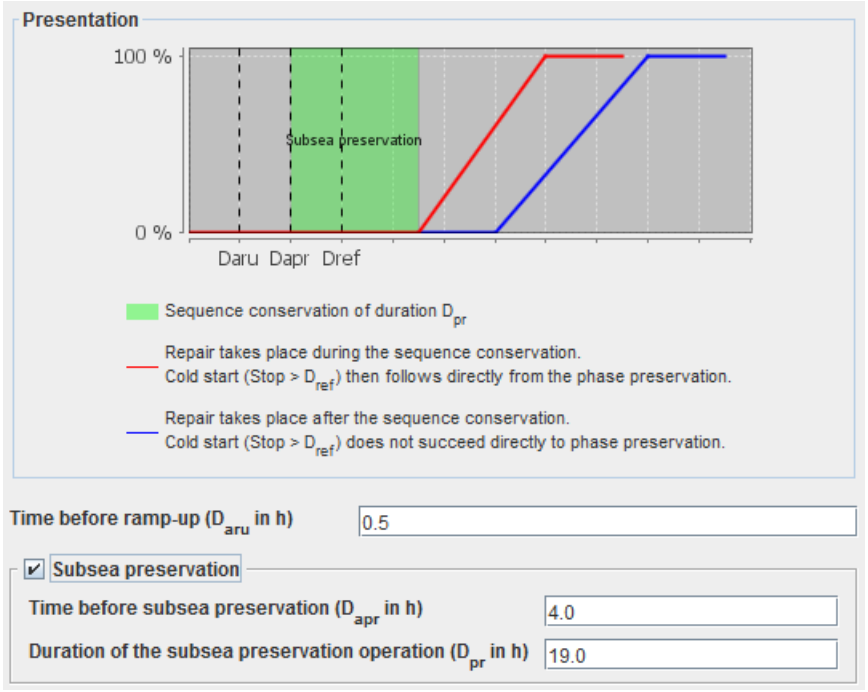
A **Number** and a **Name** can be entered here. Tick the **Automatic** box to generate a name comprising a base name followed by the number.

A description of the component can be entered in the **Description** field right at the bottom of the screen.

The fields to be completed are used to set the durations that will define the component start-up graph.

- **Time before ramp-up**: period of inactivity beyond which a restart/start-up phase is considered.
- **Time before cool down**: length of time after which the shutdown system is considered as cold and will therefore require a longer restart/start-up phase.
- **Duration of a warm restart**: length of time needed by the system to increase from 0% to 100% of its production capacity when it is still warm.
- **Duration of a cold restart**: time taken by the system to increase from 0% to 100% of its production capacity when it has completely cooled down.

In the case of an extended shutdown, a preservation sequence may be launched which in practice is an extension of the shutdown period.



Configuration of the preservation sequence:

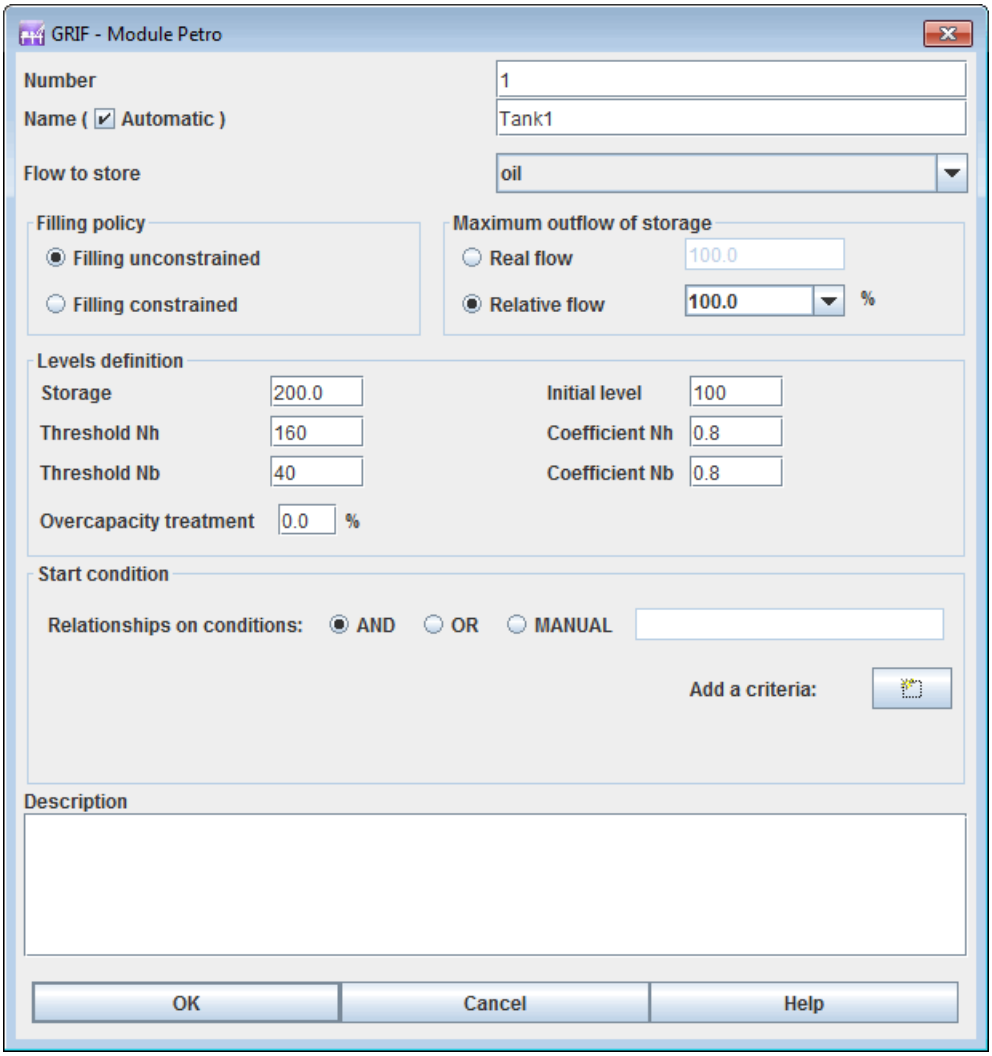
- **Time before preservation sequence**: period of inactivity beyond which the preservation phase is considered.
- **Duration of the preservation sequence**: the system cannot restart during this period.

In tabs in the right of the input area, *Ramp-up* describes all the ramp-up block in the model.

4.5.4. The Tank

This component serves/acts as a buffer, i.e. when the system is not working perfectly, this component can either be filled or emptied to reduce the impact of a failure on production as much as possible.

The tank can be configured by double-clicking on the component or by a right click, then selecting the Properties menu.

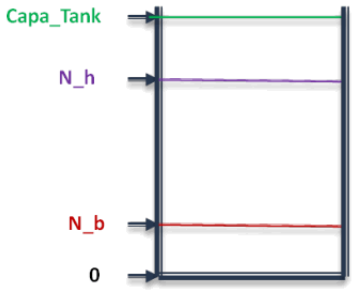


A **Number** and a **Name** can be entered here. Tick the **Automatic** box to generate a name comprising a base name followed by the number.

The **Flow to store** is the flow that will be managed by the tank. The other flows should not have to run through the tank. Use a connector-separator upstream so that only the desired flow circulates within the component.

The **maximum outflow of storage** is the tank processing capacity.

The tank has 4 levels:



- **0**: the tank is empty.

- **N_b**: the tank has reached its lower threshold at the time it is emptied. Between **[0, N_b]**, the user can reduce the outflow rate (**Coefficient Nb**) so that it takes longer for the tank to empty.
- **N_h**: the tank has reached its upper threshold at the time it is filled. Between **[N_h, Capa_tank]**, the user can reduce the inflow rate (**Coefficient Nh**) so that it takes longer for the tank to fill.
- **Capa_Tank**: the tank is full.

The **initial level** is the quantity of flow present in the tank at t=0.

The **Processing overcapacity** is the percentage increase of the request. When the system is running as well as it possibly can, the tank is given a processing overcapacity so that it can be filled to its initial level.

Two filling policies are possible when the tank has reached its limits (empty or full).

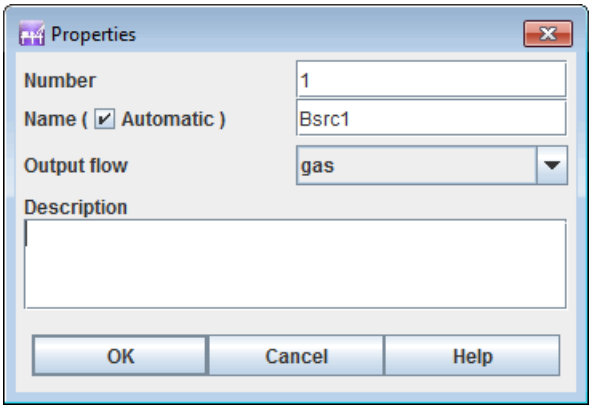
- **Unconstrained filling**:
  - If tank is full (resp empty) when new failure occurs it is possible to full (resp empty) again without delay.
- **Constrained filling**:
  - If tank is full (resp empty) when new failure occurs it is possible to full (resp empty) again only if level is lower than Nh (resp upper than Nb).

A **start condition** can also be defined to put the component on standby if the condition is not met.

A description of the tank can be entered in the **Description** field.

4.5.5. Connecting utilities

This component is used to define a point of entry for utility networks.



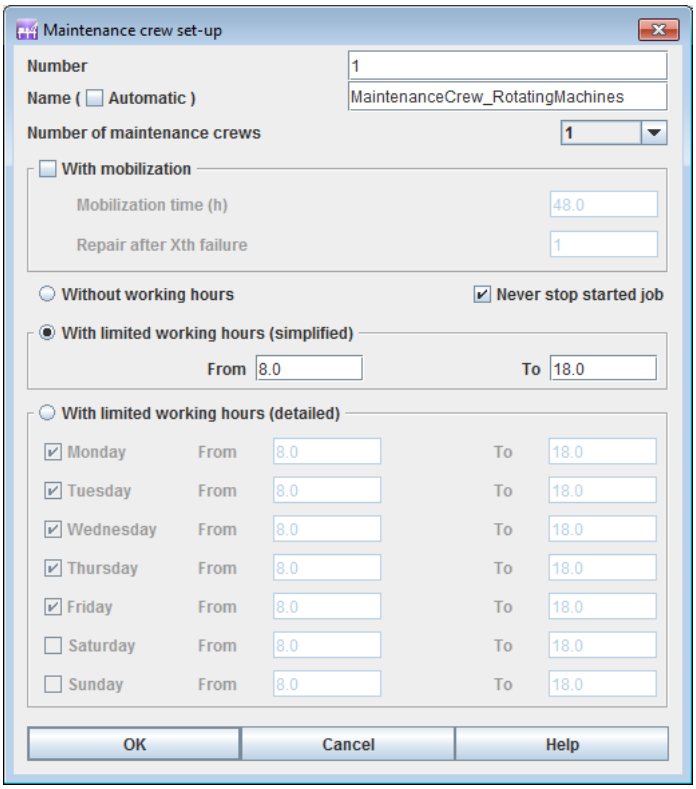
A **Number** and a **Name** can be entered here. Tick the **Automatic** box to generate a name comprising a base name followed by the number.

A description of the component can be entered in the **Description** field.

The **output flow** is the flow that will be used by the utility networks, but it is not consumed by them. It simply enables the user to determine whether or not the requested flow is circulating at a given point on the utility network.

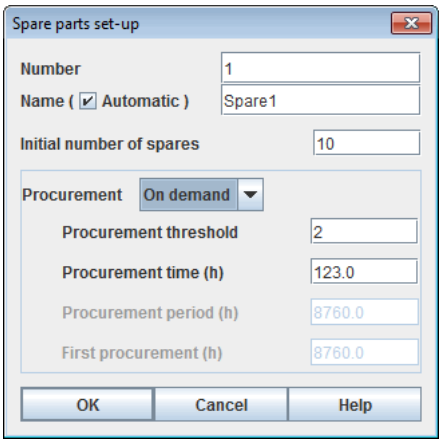
4.6. Resources

4.6.1. Maintenance crew



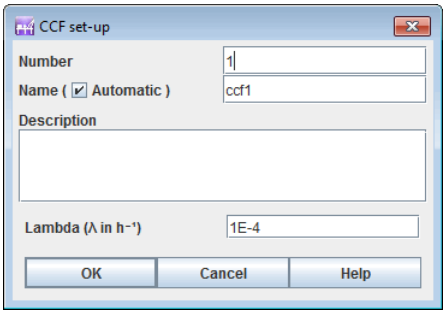
- **Name**: maintenance crew block name.
- **Number**: maintenance crew block number.
- **Number of maintenance crews**: number of maintenance operators in the crew (up to 100).
- **With mobilization**: specifies whether or not the crew is to be mobilized.
- **Never stop started job**: Even if working day is ended, the current reparation is not stopped.
- **Mobilization time**: time as of which the crew can intervene on the failure.
- **Repair after Xth failure**: number of items of equipment in failure mode as of which the crew will be mobilized.
- **Working hours**: specifies the intervention window within which the crew can work on the failure.
- **From**: start time for the intervention window.
- **To**: end time for the intervention window.
- **Detailed**: specify start and time for each day.

4.6.2. Spare parts



- **Name**: spare parts block name.
- **Number**: spare parts block number.
- **Initial number of spares**: number of parts comprising the initial stock.
- **Procurement**: type of spare part procurement **(None, periodical, on demand)**.
- **Stock after procurement**: number of spare parts in the stock after procurement.
- **Procurement threshold**: size of stock as from which procurement is triggered (applicable only in the case of on-demand procurement).
- **Procurement time**: time taken to replenish stock once the threshold has been reached (applicable only in the case of periodical procurement).
- **First procurement**: date of first procurement (in hours) (applicable only in the case of periodical procurement).
- **Time between 2 procurements**: time between 2 procurement operations (in hours) (applicable only in the case of periodical procurement).

4.6.3. Common cause failures

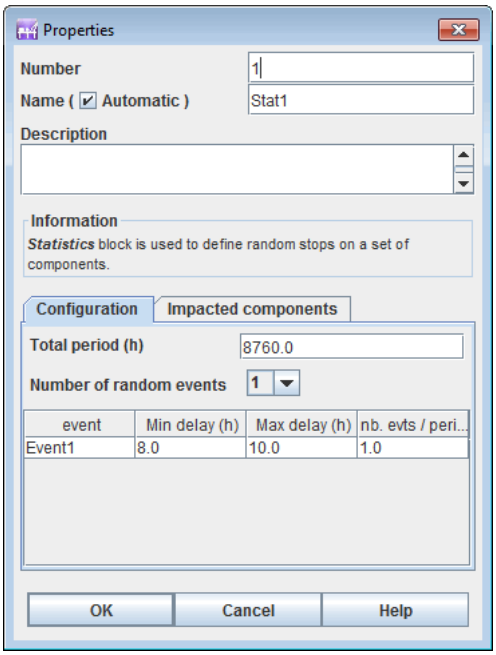


Mean time to repair is specified when CCF is linked to a component.

- **Name**: Common cause failure block name.
- **Number**: Common cause failure block number.
- **Lambda**: definition of the lambda used in the law governing the common cause failure.

4.6.4. The Statistics block

This block is used to manage random events (characterized by a frequency) that cause the partial or global shutdown of the installation.



It includes:

- **Total period** period in hours over which an event may occur.
- **Number of random events** that can happen on average over the given period (maximum of 4 possible events)
- List of **components impacted** by the random events.

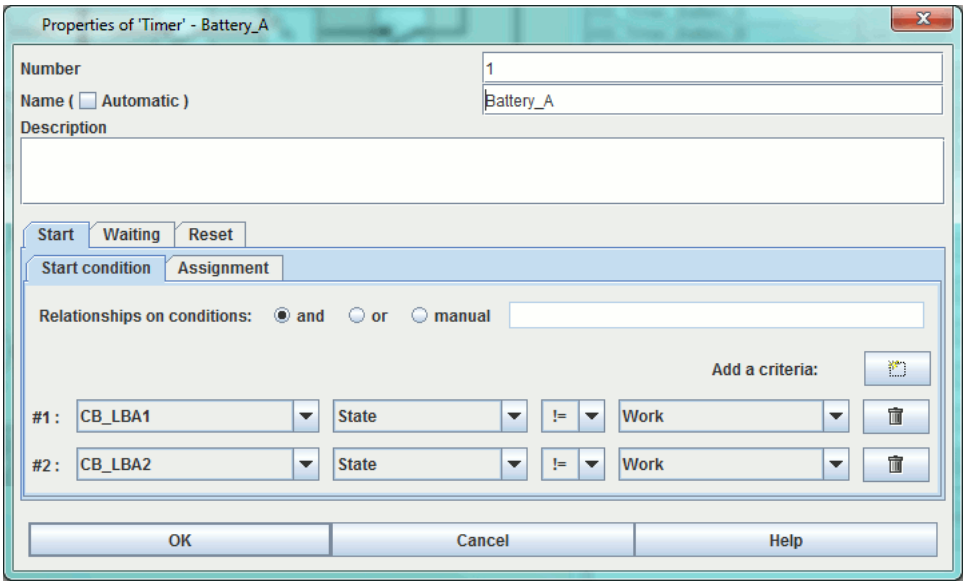
The repair time varies according to the type of event, and follows a triangular law included in an interval [delay_min ; delay_max] where the optimum corresponds to the average (delay_min + delay_max) /2.

To summarize the behaviour of this block, if an event occurs, it sends a signal (true shutdown) to the system impacted by the shutdown. Each component in the system then switches to standby until the event is repaired and consequently, until the signal stops.

In tabs in the right of the input area, *Statistical data* describes all the statiscal data block in the model.

4.6.5. Timer

This block is enables to activate a chronometer if the conditions of initialization are respected.



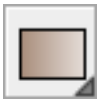
It is made of 3 tabs :

- **Start** enables :
  - to indicate in the **start condition** tab under each condition the timer is activated ;
  - to modify variable in **Assignment** tab.
- **Waiting** indicates the **Waiting delay** it means the duration of the activation of the timer if reset conditions are not respected. It is also possible to save the duration already spend in this state with checking **Transition with memory**.
  - **Waiting condition** enables to add condition at the start of the wainting delay.
  - **Assignment** enables to modify a variable.
  - **Reset condition for waiting** indicates the conditions of the timer stop even if the waiting delay is not completely sold.
- **Reset** enables to define **Reset condition** possible change of variables in **Assignment** tab.

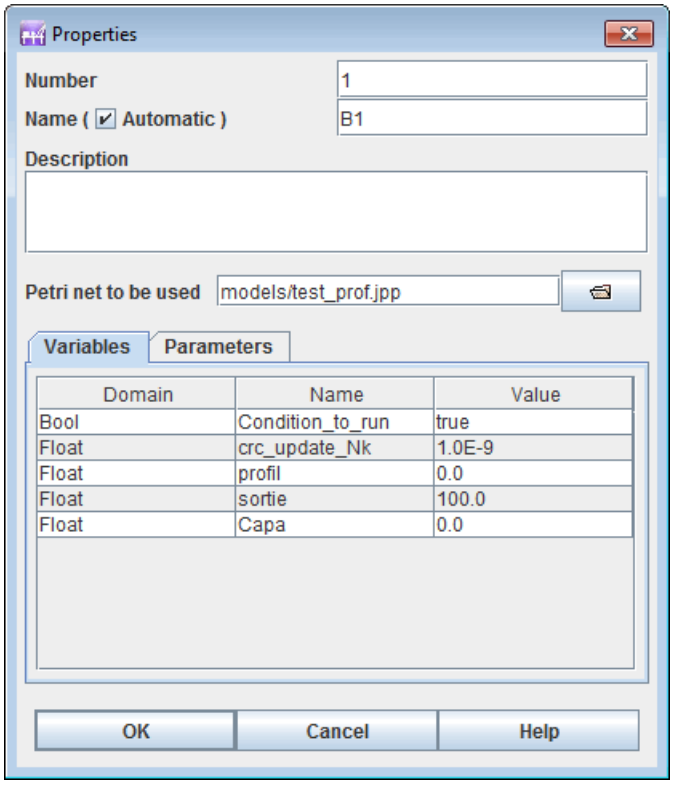
4.7. Generic blocks

4.7.1. The generic component

If you are missing a component, you can create your own Petri nets (using the GRIF-Workshop Petri module), and access it through the **generic** component.



The net can be configured by double-clicking on the component or by a right click, then selecting the **Properties** menu.

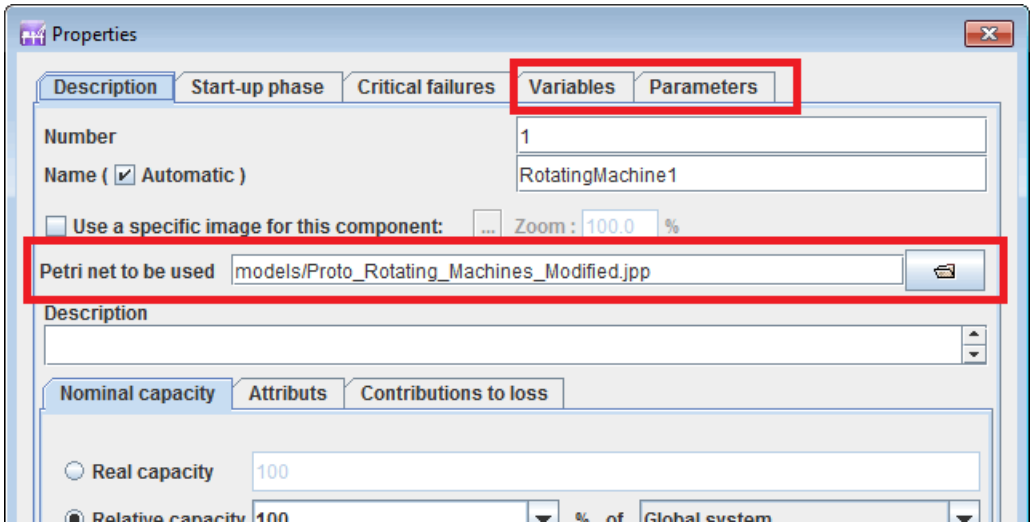


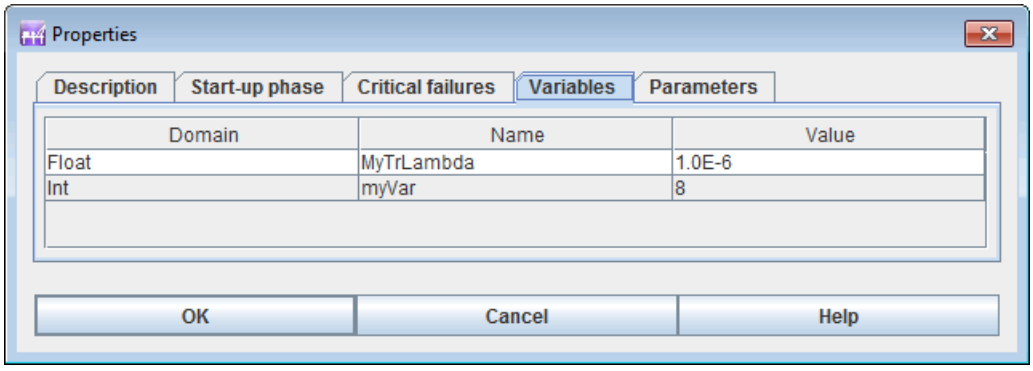
Select the **Petri net to be used**. The variables and parameters defined in the Petri net are accessible via the Petro interface. Note that the modifications made to a component do not affect the other components that are using the same net.

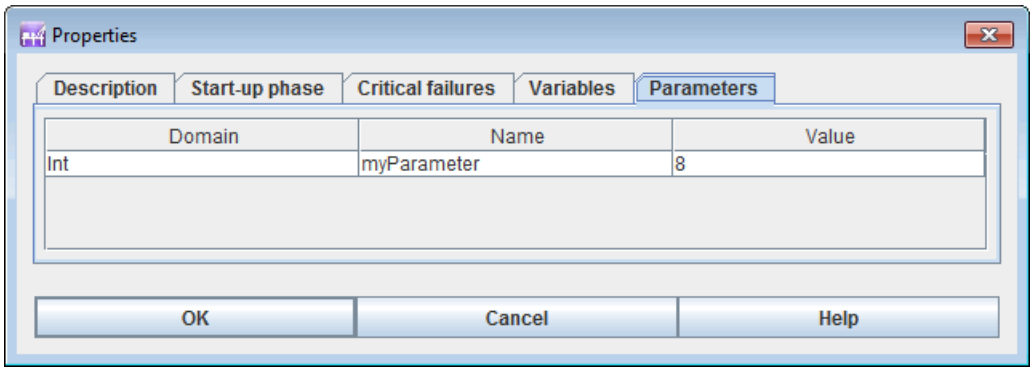
4.7.2. Generic rotating and static machines

It is possible to make slight changes to the models used by rotating and static machines. The modified Petri nets can then be used via the **generic rotating machine** and **static machine** blocks. The user then accesses the

configuration interface for rotating machines (resp.static), with the additional tabs **Variables** and **Parameters** to customize the nets.







N.B. Not all modifications are tolerated. For example, a user cannot delete a transition or data from initial nets. This function must be used to enhance the initial net or to modify a transition firing rule.

5. Utility Diagram Block

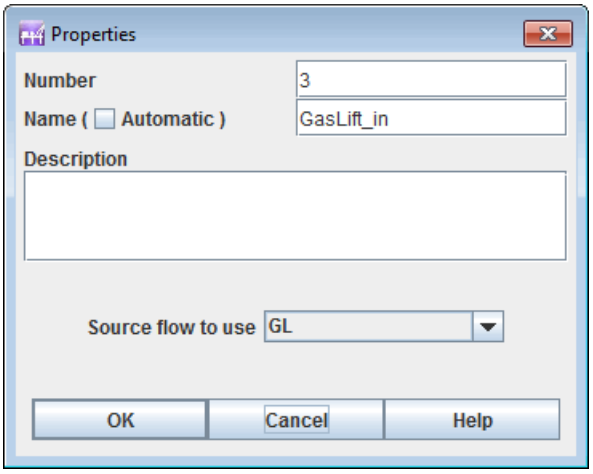
The utility networks condition the operation of rotating and static machine components. The components of Process networks are not all available in the utility network, but the static and rotating machines, the tank the generic components and the resources blocks do appear.

5.1. Input and output nodes

5.1.1. Utility source

The input node for utility networks. The entry node produce infinitely provided that the Process source to which it is possibly linked is active

The source can be configured by double-clicking on the component or by a right click, then selecting the **Properties** menu.



A **Number** and a **Name** can be entered here. Tick the **Automatic** box to generate a name comprising a base name followed by the number.

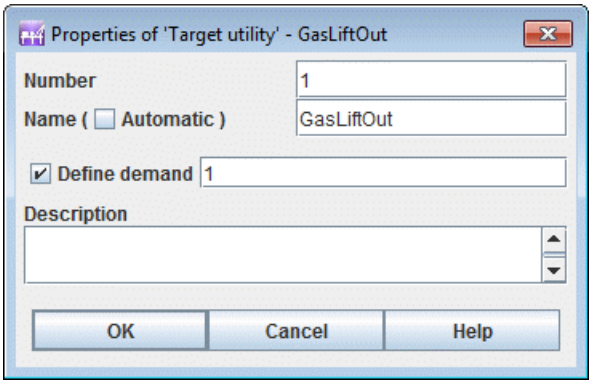
A description of the component can be entered in the **Description** field.

Source flow to use: used to indicate the Process source (**Utility connection** component in the Process pages) that conditions the operation of the utility network.

5.1.2. Utility target

The output node for **utility** networks that can be reused in the utility conditions of components to condition their operation.

The source can be configured by double-clicking on the component or by a right click, then selecting the **Properties** menu.



A **Number** and a **Name** can be entered here. Tick the **Automatic** box to generate a name comprising a base name followed by the number.

A description of the component can be entered in the **Description** field.

The **Define demand** field is used to modify the value requested by the utility target on its upstream branch. By default this value is infinite.

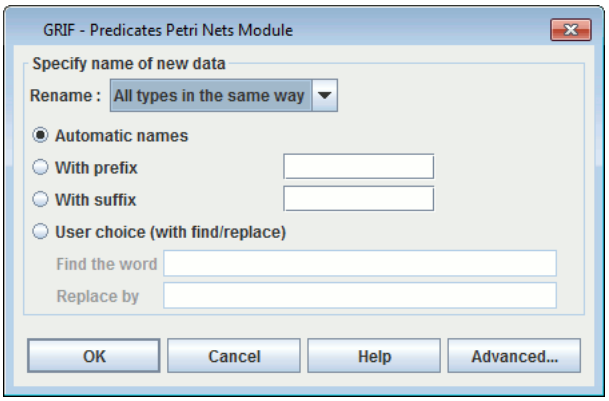
6. Data Entry Aids

To simplify model creation the Petro module has different data entry aids to automate time-consuming operations.

6.1. Copy / Paste / Renumber (without shortcut)

To assist with the entry of the repeated parts of the Petro "Copy / Paste and Renumber" mechanisms have been provided. This operation is carried out in 6 steps:

1. Select the part to be copied.
2. Click the **Copy** icon, or use menu **Edit - Copy** or the shortcut Ctrl + C.
3. Click the **Paste and Renumber** icon, or use menu **Edit - Paste and Renumber** or the shortcut Ctrl + R.
4. A window appears where you choose the way to rename the elements.

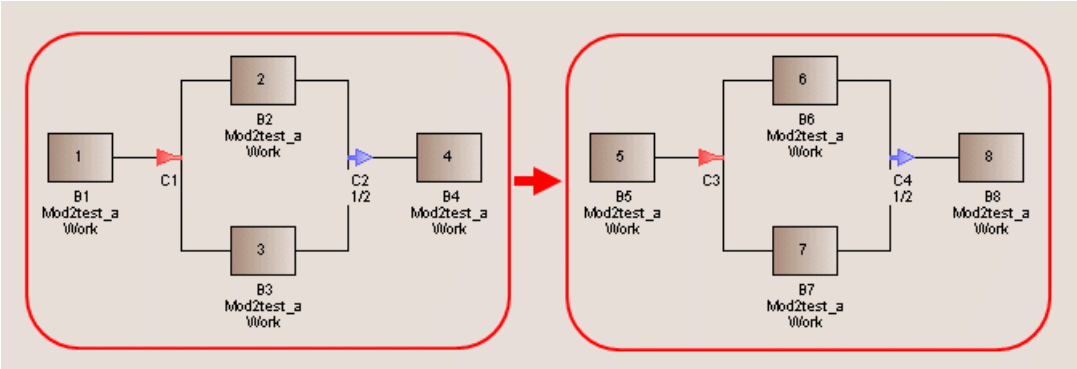


Automatic names choose allows to add a number of the name of the place.

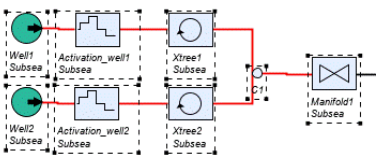
5. The previously selected part is copied and the copy is selected.
6. Move the copy to the desired location.

We then obtain the diagram shown in the figure below:

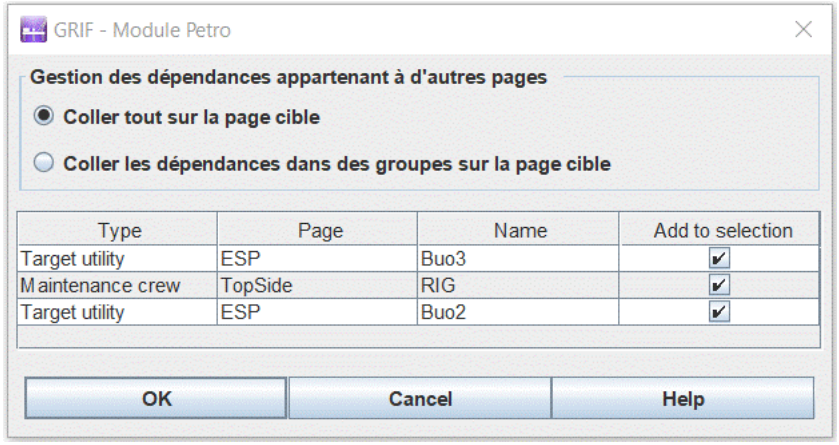
- Blocks 1,2,3 and 4 of diagram are became 5,6,7 and 8 for the copy;
- Connectors **C1** and **C2** are became **C3** and **C4** for the copy.



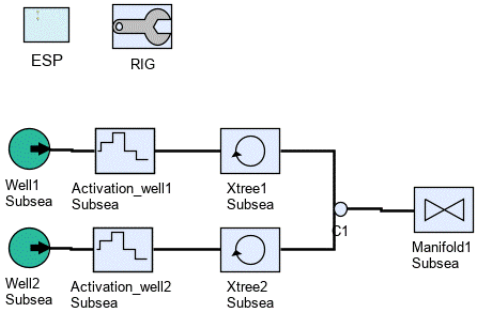
When copying to a new document, There may be dependencies with the copied data.



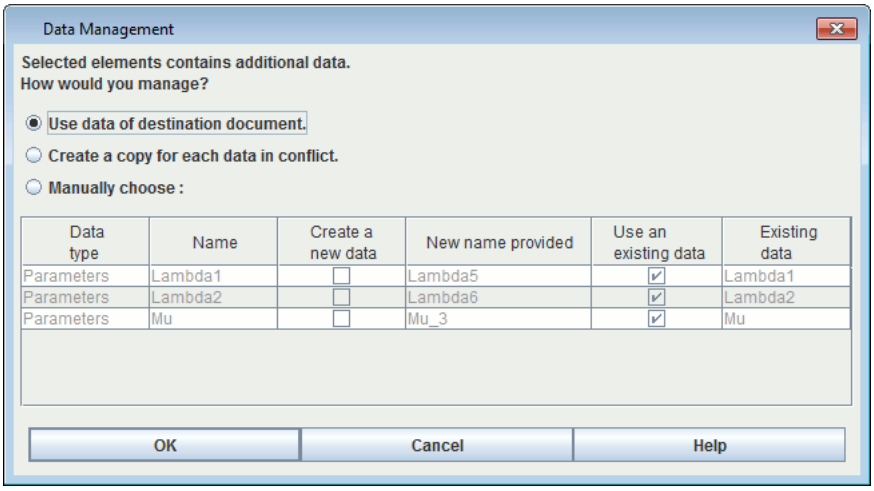
The following window opens and enables to manage indetified dependencies by GRIF :



After managing of dependencies, these are pasted with the selected data



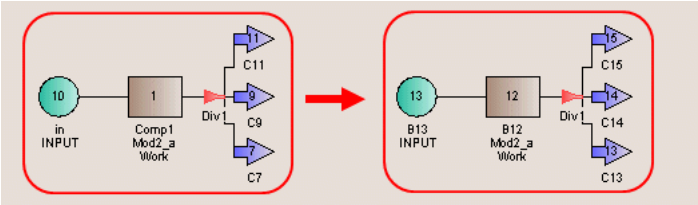
When copying to a new document, any data conflicts are handled in the following window:



This window shows all the data which has the same name in the source document and the destination document. There are three choices:

1. Use data of destination document, this will replace the occurrences of the data in the source document by the data with the same name in the destination document.
2. Create a copy for each data in conflict, this will replace the occurrences of the data in the source document by a copy with a name with the suffix "copy".
3. Manually manage conflict, this allows you to choose whether you use the existing data or not, depending on the data. You can also specify the name of the copy by double clicking on the box in the "destination document" column. The names in this column are normally masked when the **Use existing** check box is selected, since it is the data which is already in the destination document which will be used.

If the selected part is made up with a shortcut, the shortcut refers always at the same source.



6.2. Ordinary Copy/Paste

In addition to the "Copy / Paste and Renumber" command there is an ordinary "Copy / Paste" function. It is used to make a single copy without renumbering. We thus obtain double elements which, from a formal viewpoint, is incorrect but which must be temporarily tolerated to simplify data entry.

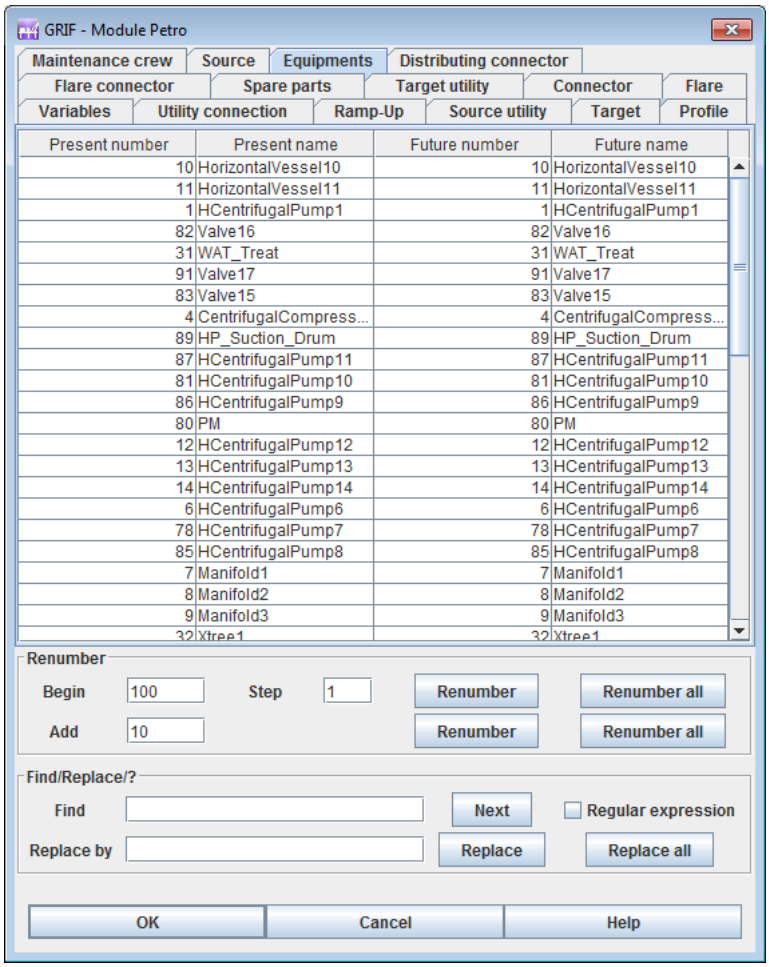
Where possible, the "Copy / Paste and Renumber" function must be used in preference to the simple "Copy / Paste" function to minimise the risk of errors. But when it is used you must take the necessary precautions to re-establish the correct numbering to eliminate the duplicates.

6.3. Overall change

When creating the Petro it may be necessary to change a large part of the elements in the models: changing the names, numbers, etc. The "Replace all" function in the **Edit** menu enables you to perform overall changes:

- Use the **Edit / Overall changes** function.
- Choose the type of elements to be modified among available tabs.
- The "Find / Replace" part changes a character string present in one or more variable labels, place labels or transition labels. It is replaced by the string entered in the "Replace" part.
- The "Renumber" part only concerns the places. It is used to change place numbers. You indicate a **Start** number then specify a constant **Step**, or **Add** a constant value to the current numbers.
- Click **OK** to return to the chart. The changes are validated.

The name changes and renumbering can be done manually if the necessary precautions are taken (avoiding duplicates, etc.). You click the **Future number** or **Future name** column and enter the change. Do not forget to validate it with the "ENTER" key.



6.4. Selection change

The "Replace selection" function is equivalent to a "Replace all" but only applied to the selected elements.

Only the selected blocks and connectors can be replaced.

The "Replace selection" function does not allow the model's parameters to be replaced.

6.4.1. Graphical editing

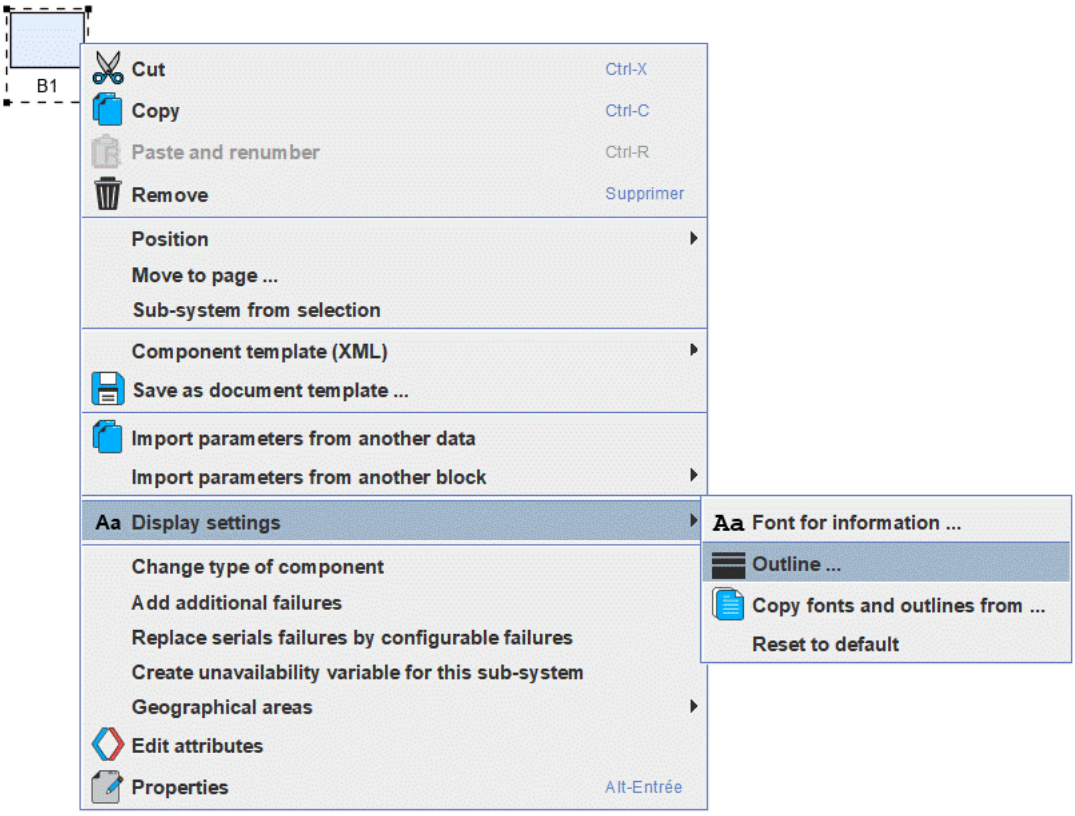
By default, blocs have a blue background, a black border and text is written in black.

Using the shortcut bar or by a right click on a block, you may edit the text apperance, the background color and borders color of the selected block:

- Select the **gate** you want to edit.
- You are now able to use the text configuration shortcut bar:

– From the tool bar.

– Right click on the block and then **display settings**.



Bellow an edited **block**:

6.5. Alignment

To improve the legibility of the model the selected elements can be aligned vertically or horizontally. To do this, use the **Align** command in the **Tools** menu.

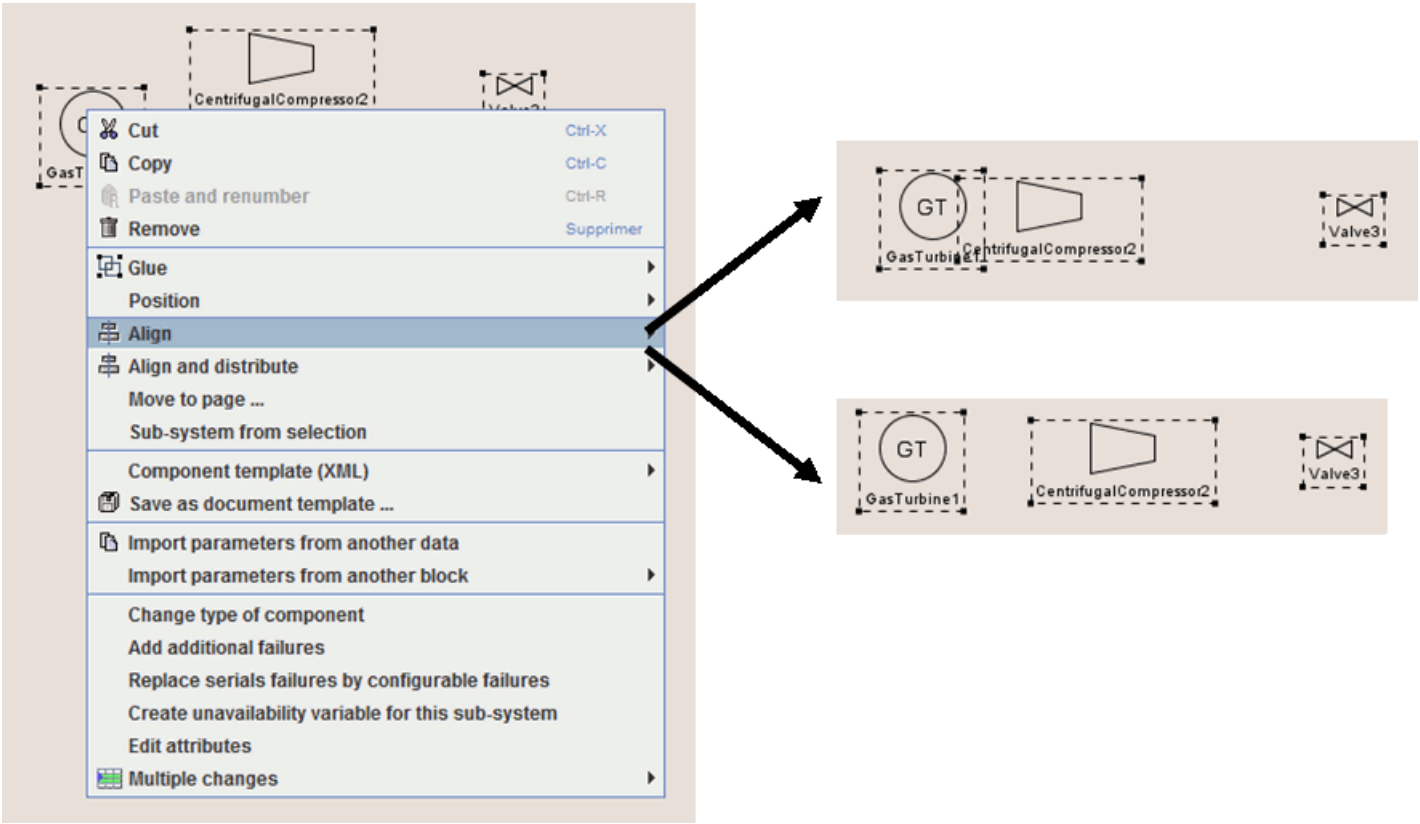
The following figure shows how the command works. For example, to align selected places and transitions vertically, proceed as follows:

1. Select the elements (places, transitions, comments, etc.) to be aligned;

2. Go into the **Tools** menu and select the **Align** function;

3. Choose the type of alignment: **Align center**;

4. Click left on the mouse.



Similarly, to align elements horizontally select the type **Align middle** which aligns the ordinates while keeping the abscissa constant. The principle is the same as that described above.

6.6. Multiple selection

It may sometimes be useful to select several elements located in the four corners of the input zone. To simplify this type of selection click on each of the desired elements one by one while holding down the Shift key on the keyboard.

6.7. Selecting a connected section

It is sometimes difficult to select a connected section of a model. A number of shortcuts can be used to select connected parts of a graphical element. Select part of the graphical element, then:

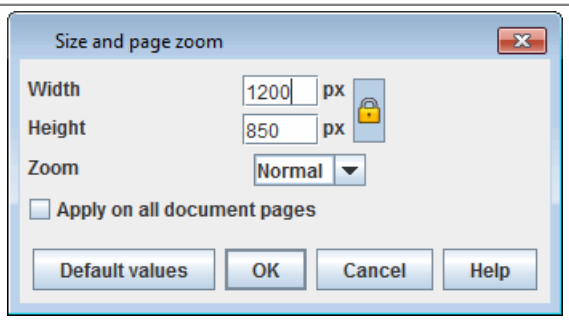
- To select the connected part: hold down **Control**+**Maj**+**A** or use the menu **Edit/Select a connected part**.
- To select the upstream part: press **F4**or use the menu **Edit/Select the upstream part**.
- To select the downstream part: press **Maj**+**F4** or use the menu **Edit/Select the downstream part**.

The connected part can be selected directly by clicking on the element while holding down the Ctrl key.

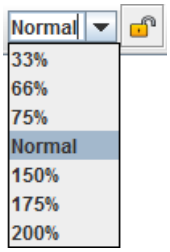
6.8. Zoom and page size

When creating a model, if the page size is not big enough, it can be changed using the menus: **Increase page size** (**Control**+**Keypad +**), **Reduce page size** (**Control**+**Keypad -**), **Page size** (**Control**+**Keypad /**) under the **Tools** menu.

The **Page size** menu enables the user to edit the page dimensions directly.



Page zooms can be modified either by using the toolbar menu:



Or by selecting the display and using **Control**+**mouse wheel scroll up** to zoom or **Control**+**mouse wheel scroll down** to zoom out.

The padlock on the toolbar is used to apply the zoom to the current page or to all pages in the document.

	The zoom applies to all pages in the document.
	The zoom is applied only to the current page.

Note that if an element is selected on the page, the zoom will centre the page on that element.

6.9. Cross hair

To be able to create an ordered and legible model quickly, the **cross hair** can be used to align the different elements with each other (but less accurately than the **Align** function in the Tools menu). The **cross hair** is enabled (or disabled) in the **Graphics** tab of the **Option** menu.

The following picture show how to quickly align two elements of the model.

In order to align horizontally, select **Align au middle** which align keeping constant abscissa.

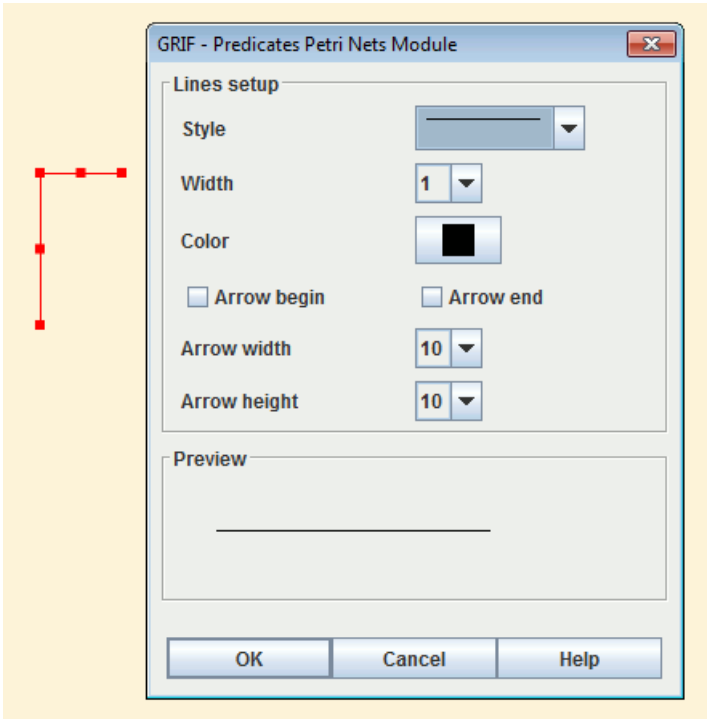
6.10. Gluing/Associating graphics

When objects are where you want, you can glue a set of objects by right-clicking and selecting **Glue**. This command creates a group (a graphical one, not a hierarchical one) with selected objects, so that moving one moves the others.

A double click on an element in the glue group opens the properties window of the element.

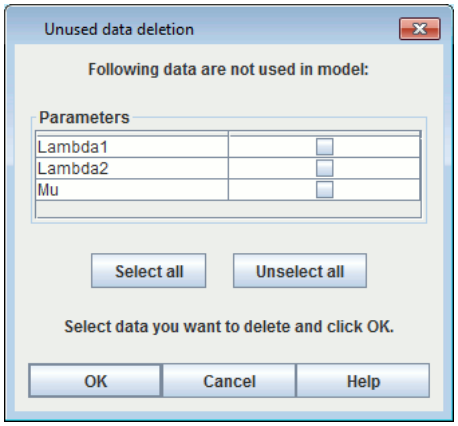
6.11. Line

To be able to draw a line, polyline or arrow, the **Line** can be used. Draw the line and edit properties of line to make an arrow.



6.12. Table Cleaning

Data may not be used anymore, it can be used useful to delete every unused data. To facilitate removal, use **Data and Computations / Unused data deletion** menu.



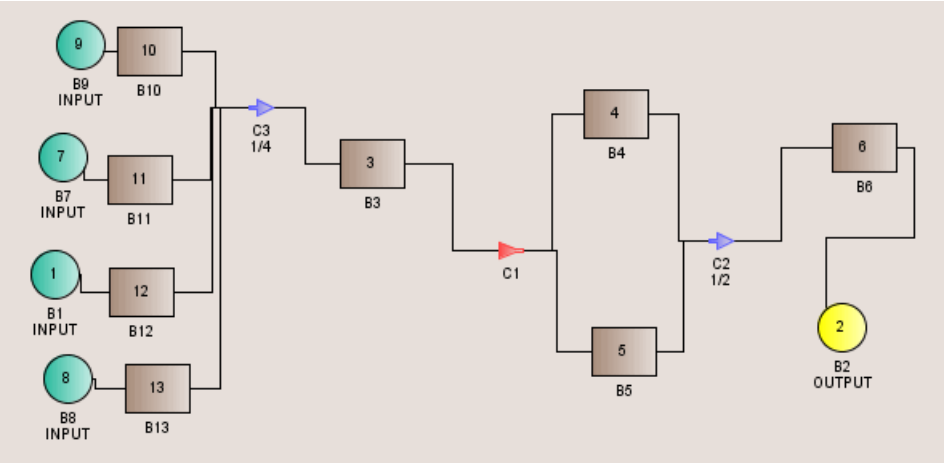
This window displays unused data. Select data you really want to delete and click OK.

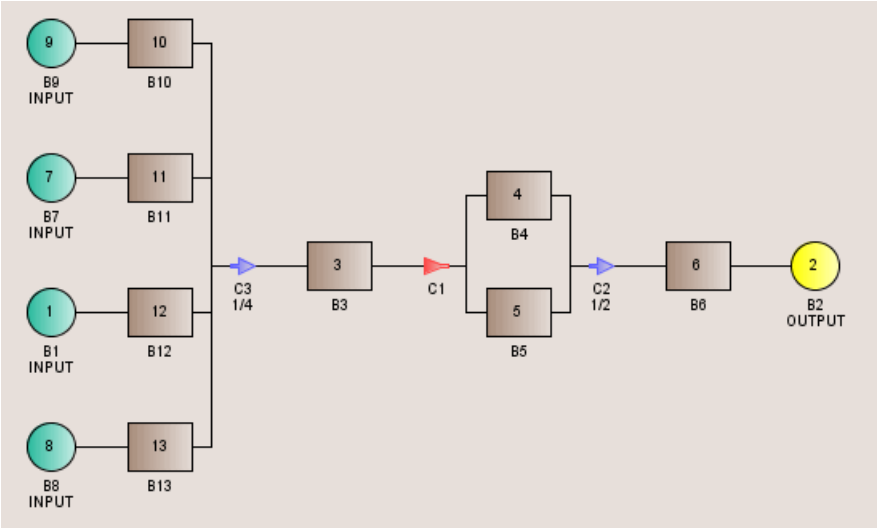
6.13. Automatic layout

The automatic layout tools can be used to make blocks easier to organize. This function is available for:

- Selection: press **Shift+F7** or use the menu **Edit/Automatic layout/Layout selection**.
- The current page: press **F7** or use the menu **Edit/Automatic layout/Layout current page.**.
- The document: press **Control+F7** or use the menu **Edit/Automatic layout/Layout document**.

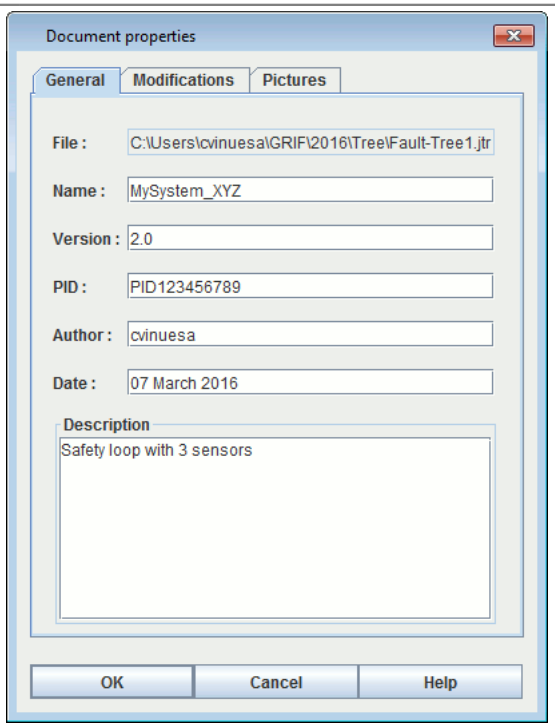
An example of layout is shown below:





6.14. Document properties / Track change / Images management

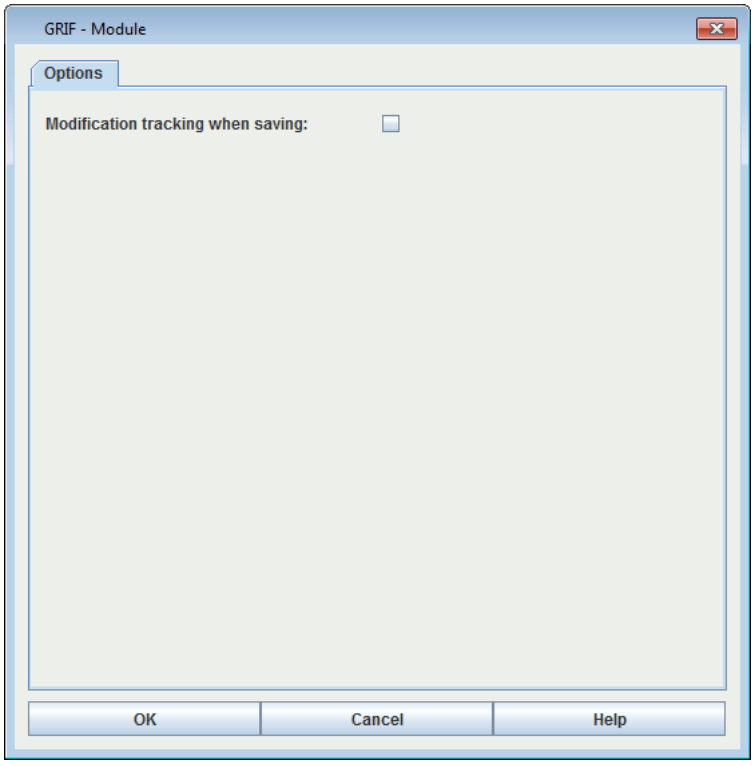
File - Document properties menu enable to save information about document: name, version, comment, ... These information are available in **General** tab.



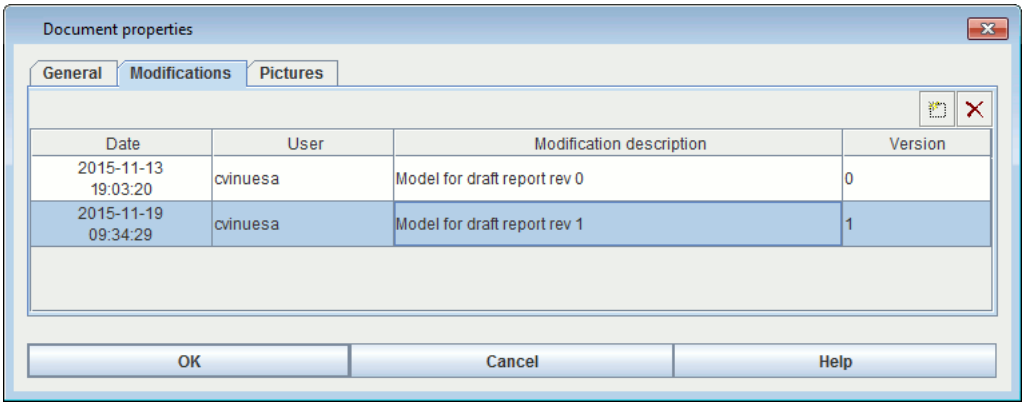
Modification tab enables to save A history of the modifications.

There are two different ways to save modifications:

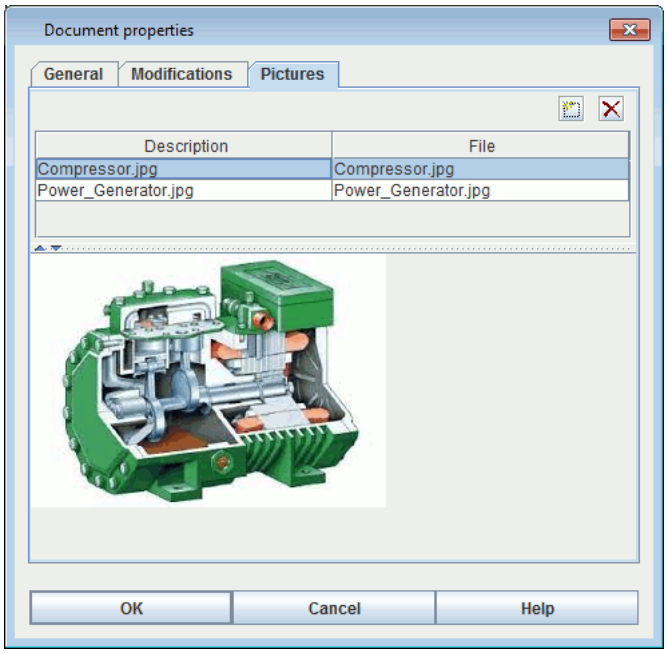
- At each saving by checking: **Modification track when saving** dans **Tools - Document (or Application) options** .



- When the user wants directly in **Modification** tab of the properties using the button 



Images may be very useful to represent sub-system. GRIF 2022 enables to save images that can be used in different parts of software (groups, prototypes, ...). Images management is made in **Images** tab.



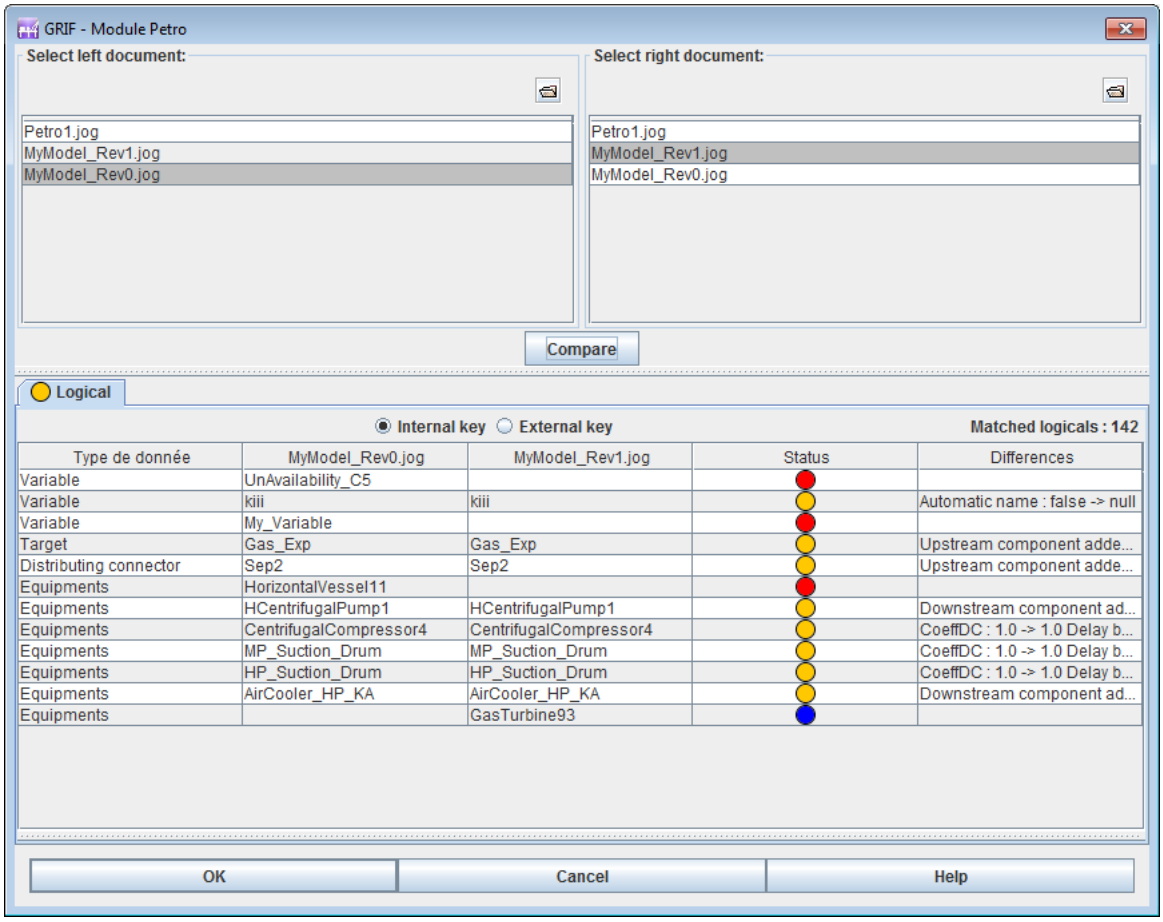
To add a new picture into document, use  icon. A double click in **File** column enables to select a picture (jpg, gif or png). A double click in **Description** column enables to give a name or a description to selected image.

Once in document, picture can be linked to a groupe with **Group - Picture change** menu.

Images are saved inside document, pay attention to picture size. Because images are inside document, you have to re-add picture if picture is modified erternaly.

6.15. Compare 2 documents

This function is accessible using **File / Compare 2 documents**. The following window appears:



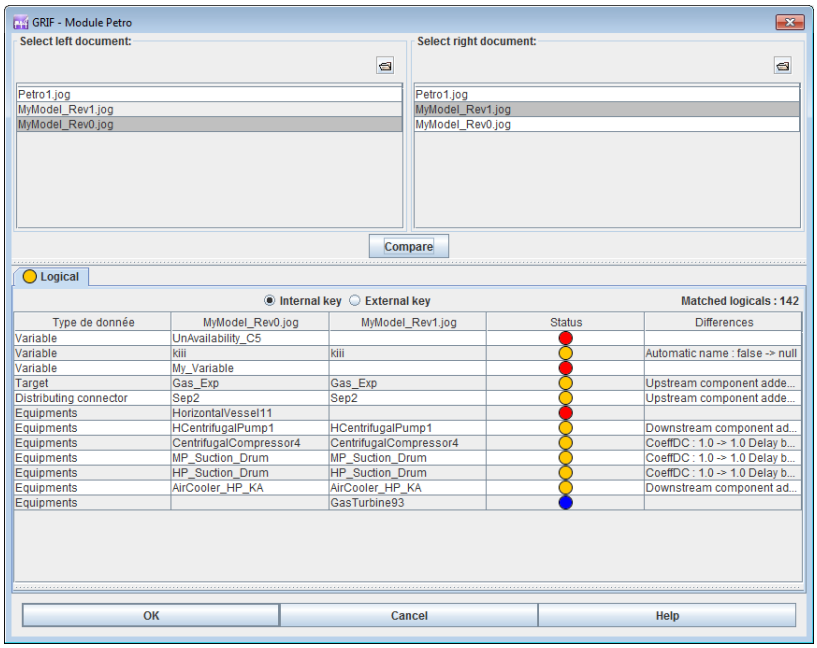
Icon enables loading of the files to be compared.

Click on Compare to launch the comparison.

Difference can be sorted using 3 criteria: internal key, external key or name for nodes

- **Internal key** enumerates the differences according to internal elements of the model for example identifier, creation index, etc...
- **External key** differentiates elements according to the names of the elements of the model.

- **Name for nodes** differentiates nodes according to their names. The external key comparison will be used for others elements.

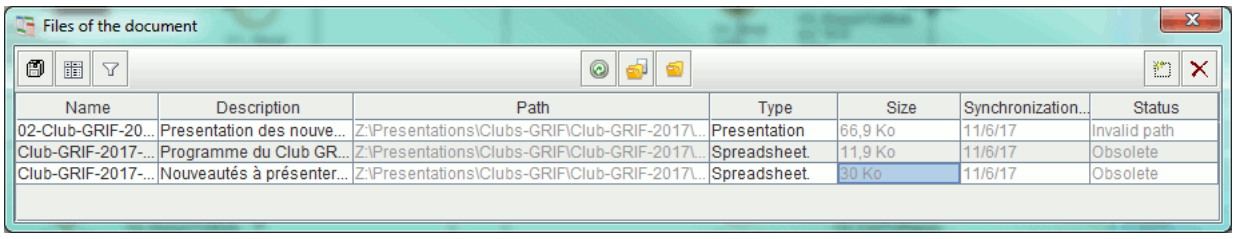


Colour signification is:

- : element is identical;
- : element is added;
- : element is modified;
- : element is deleted.

6.16. Files of the documents

It is possible to associate external file using **File - Files of the document** menu.

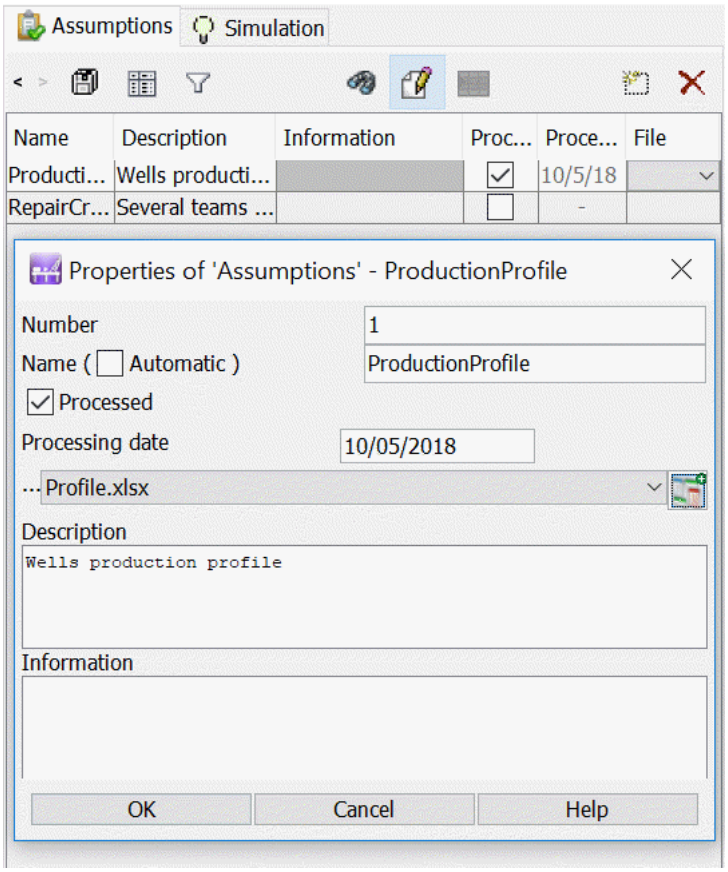


The following icons allow to:

- reload files;
- open files;
- open directory where file is saved.

6.17. Hypothesis

In the data table, in **Hypothesis** tab, it is possible to follow-up and track the studies hypothesis.

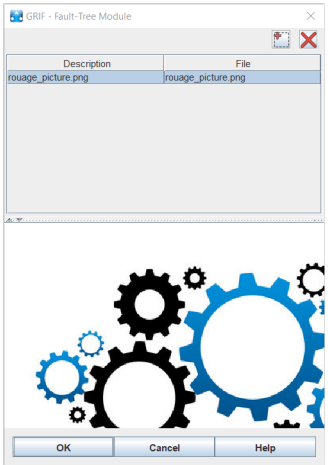


This table enables to take into account the study hypothesis and add file or date to indicate that this hypothesis is taken into account.

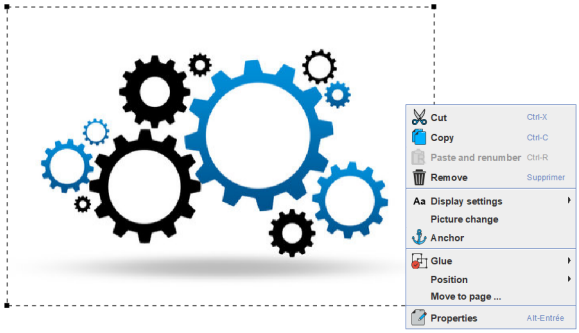
6.18. Picture Anchor

It's possible to anchor a picture in the background. use the action **Picture** of the **Tool barre**.

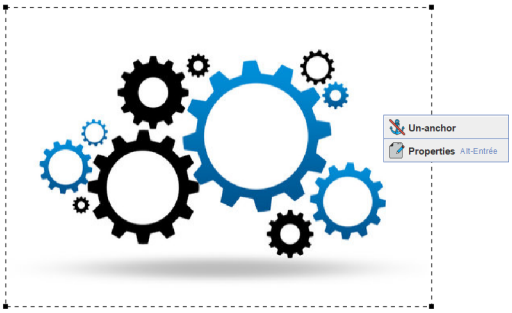
The following screen is display :



It's possible to select a specific picture that is display in background. To anchor the picture, with a right click, select the **Anchor** action. It's possible to change the picture with the **Picture change** action:



To unanchor the picture, with a right click, select the **Un-anchor** action :

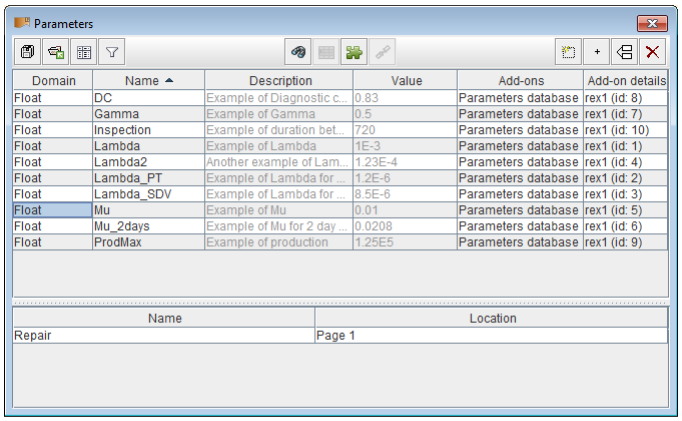


7. The parameters

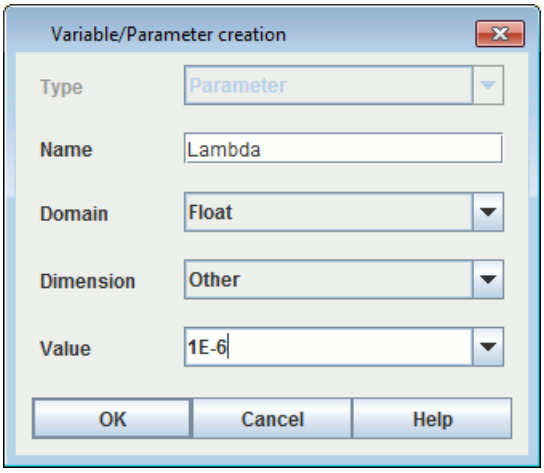
It is possible to create constants which can be booleans, integers or reals. These parameters can then be used for the configuration of different elements of the model (laws, events, transitions, ...)

7.1. Creation

The tab **Parameters** enables the user to define his parameters.



The toolbar enables to do basic operations of the data tables(Section 1, “Description of the Tables”). The button "New" opens the window to create a parameter :



A parameter has a name, a definition domain (Real, Boolean, Integer), a value and a dimension (Failure rate, probability, time, factor, ...) which allow to specify the parameter. This typing is at this moment informative.

Others additional fields are available in the parameters' table.

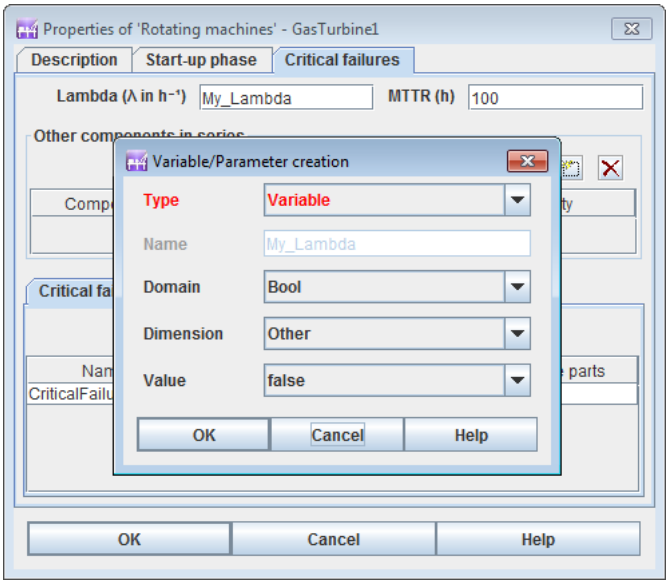
Add-On		enables to define the parameter by a GRIF add-on Petro is delivered by default with 1 add-on for the parameters : **Parameters database** : is an add-on which enables the user to get the data of his parameter in a database or in a CSV or Excel file. This database is more detailed in this section Section 15, “Database of parameters”.
Add-on details		gives a synthesis of the data defined by the add-on. A double-click on the cell enables the user to modify its definition.
Parameters database	**Database**	Displays the database name containing the parameter.
	Identifier	Displays the identifier of the data in the database.
	Update	Displays the date of the last update of the parameter from the database.

8. Variables

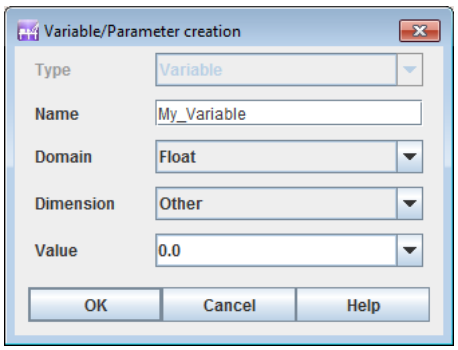
8.1. Variables creation

Variable creation can be made with 2 different ways:

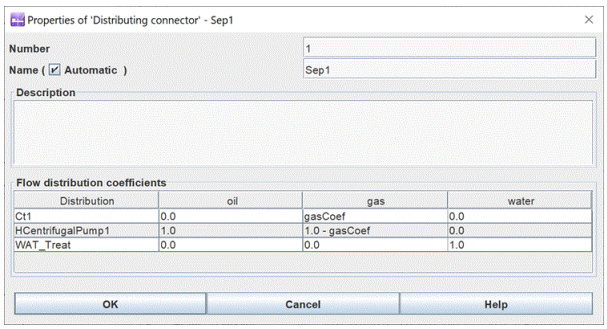
- Using data tab of variables which is at right of the input area thanks to the icon ;
- Or directly in a component by writing the name of the variable wished.



Each variable is defined by a name, a domain (boolean, integer or real), a dimension (boolean, factor, probability, rate, duration or other) and a value.



Variables can be used in properties of modeled elements (equipments, connectors, spare partss, etc)



8.2. Variables edition

When the variable was created, it is then possible to edit it by using the icon  .

Description tab enables to define the variable using 4 different ways:

- With a specific formula;
- With a value of a profile table;
- According to conditions resulting from elements of the model;
- According to data resulting from elements of the models.

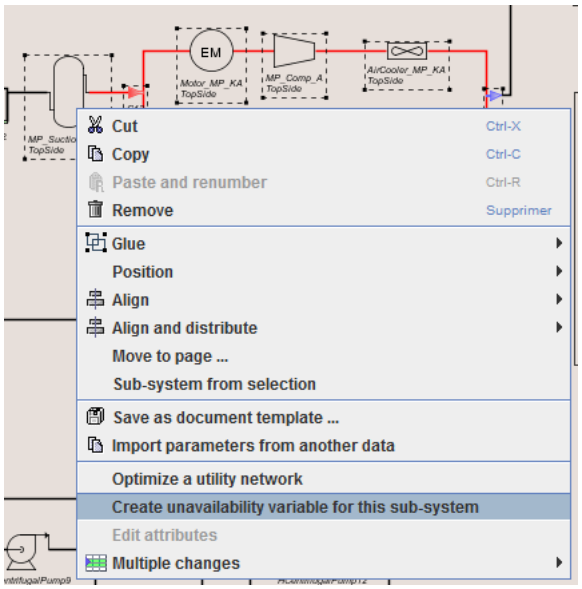
There are two categories of variables:

- States variables: whose definition depends on the model;
- Flow variables: further to an action on the model.

8.2.1. Automatic variables creation

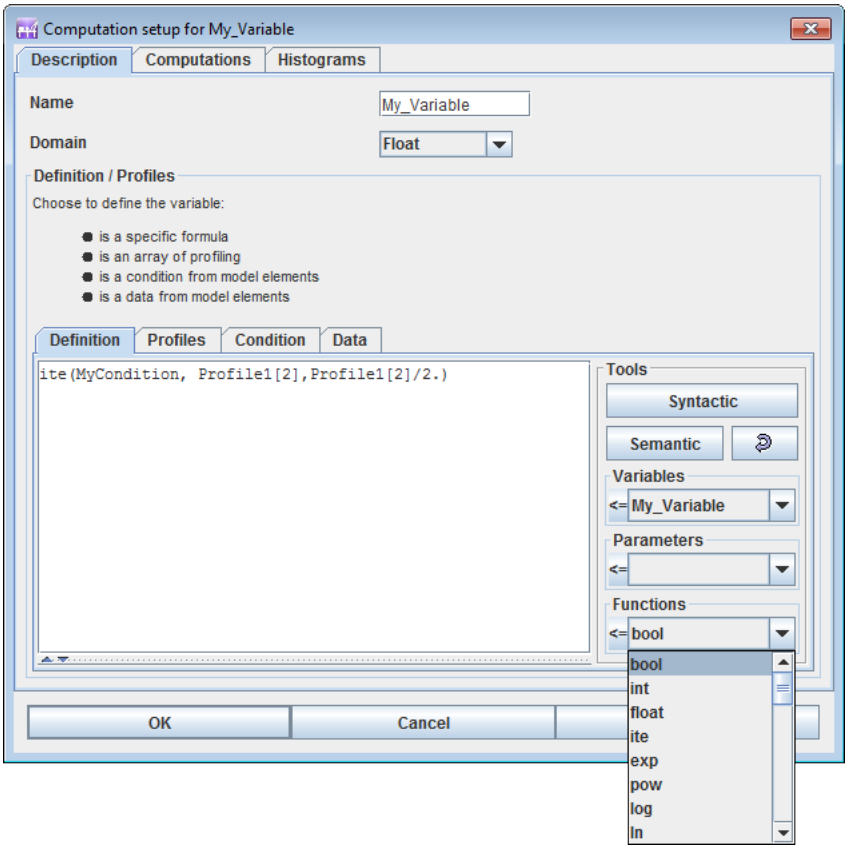
It is possible to create un variable for the unavailability directly in the model. For this:

1. select the elements;
2. in the contextual menu (right click), select **Create unavailability variable for the sub system**;
3. Give a name of the variable.



This variable is a boolean variable which represents the system unavailability. The value is **False** if the system is unavailability and **True** otherwise.

8.2.2. Variables definition



In this part formula, it is possible to use two types of expressions:

- Boolean expression: They are formed from variables, constants and boolean tables (`true` and `false`), brackets and operators `|` , `or` , `&` , `and` , `!` , `no` , `==` , `!=` , `<` , `>` , `<=` , `>=` and `@` .
- or numerical expressions: There are made up with variables, constants and numerical tables (i.e. integer or float) and the operators `+` , `-` (binary), `-` (unary), `*` , `/` , `#` , brackets and mathematical functions.

A boolean expression can be:

- one of the constant `true` or `false` ;
- e_1 `|` e_2 , e_1 `or` e_2 (where e_1 and e_2 are boolean expressions) is the logical OR of e_1 and e_2 ;
- e_1 `&` e_2 , e_1 `and` e_2 (where e_1 and e_2 are boolean expressions) is the logical AND of e_1 and e_2 ;
- `!expr` , `not expr` (where `expr` are boolean expressions) is the logical negation of `expr` ;
- The usual operators of equality and inequality e_1 `==` e_2 , e_1 `!=` e_2 , e_1 `<` e_2 , e_1 `>` e_2 , e_1 `<=` e_2 , e_1 `>=` e_2 (where e_1 and e_2 are two numerical expressions of the same type; its means either integer or float);
- `@(i)(`e_1`,...,`e_n`)` (where `i` is a positif integer and e_1`,...,`e_n are boolean expressions) is equal to true if at least `i` of e_1`,...,`e_n are equal to true;
- `@(i,j)(`e_1`,...,`e_n`)` (where `i` and `j` are positive integers and e_1`,...,`e_n are boolean expressions) is equal to true if at least `i` and more than `j` of e_1`,...,`e_n are equal to true.

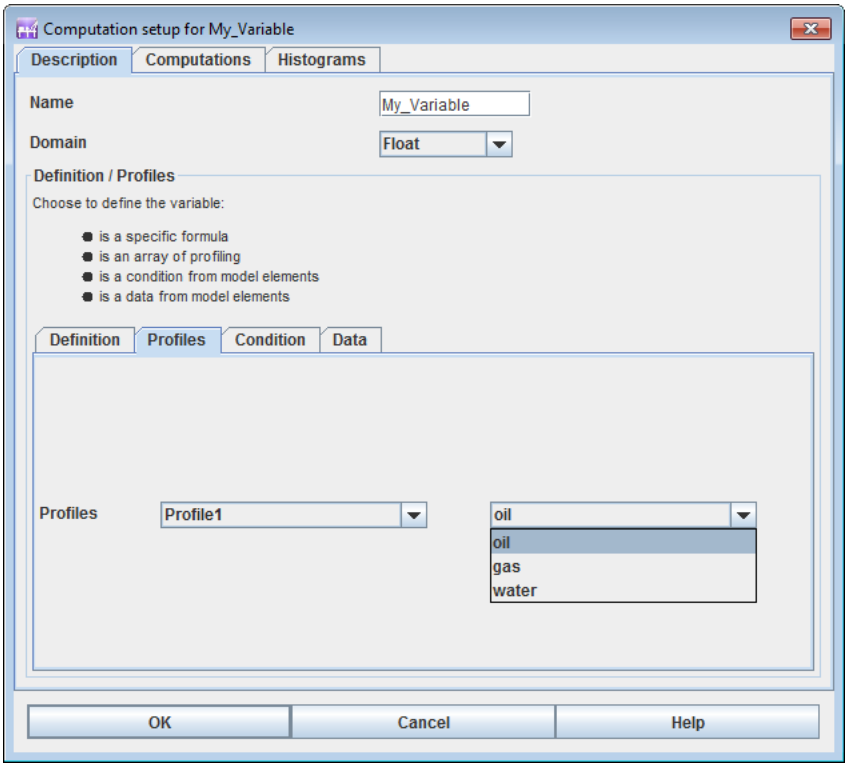
A numerical expression can be:

- `tab.size()` (where `tab` is the table name) which is equal at the number of the table element;
- a float or integer number;
- `int(expr)` (where `expr` is a numerical expression) is a conversion of an expression in an integer expression. The `int(expr)` value is the value of `expr` rounded to its lower integer part;
- `float(expr)` (where `expr` is a numerical expression) is a conversion of the expression `expr` in a float expression;
- `-expr` (where `expr` is a numerical expression) has for value the opposite of `expr` ;
- `(expr)` is a numerical expression with the same value of `expr` ;
- `#i` (where `i` is an integer > 0) which is equal to the marking of the place number `i` in the network;

- The usual arithmetic operations `e1+e2` , `e1-e2` , `e1*e2` and `e1/e2` (where `e1` and `e2` are two expressions of the same type that is either integer, or float);
- Operator `e1%e2` (where `e1` and `e2` are two expressions of the same type that is either integer, or float); has for value the rest of the integer division of `e1` by `e2` ;
- `unif(e1,e2)` (where `e1` and `e2` are two constant float expressions) is a random numbers generator following a uniform distribution in the interval [`e1` , `e2`] (see below) ;
- `nlog(m,e)` (where `m` and `e` are two constant float expressions) is a random numbers generator following a lognormale distribution centered in the average value `m` And being with a 90 % probability in the interval [`m/e` , `m*e`] (see below) ;
- `norm(m,sd)` (where `m` and `e` are two constant float expressions) is a random numbers generator following a normal distribution centered in the average value `m` and the standard deviation `sd` ;
- `min(e1,...,en)` , `max(e1,...,en)` , `mean(e1,...,en)` and `max(e1,...,en)` (where `e1,...,en` are same type expressions) are functions calculating the minimum, the maximum, the average or the sum of the expressions puted in parameter;
- `crc(e1,...,en)` (where `e1,...,en` are float expression) is a function calculating a checksum type CRC on the expressions crossed in parameter (the objective being to verify easily if at least one of the parameters of the function changed);
- An usual mathematical function among (the parameters of these functions being float expressions):
  - `ln(e1)` : natural logarithm
  - `exp(e1)` : exponential
  - `log(e1)` : base-10 logarithm
  - `pow(e1, e2)` : power
  - `sqrt(e1)` : square root
  - `abs(e1)` : absolute value
  - `ceil(e1)` : rounded off the integer inferior
  - `floor(e1)` : rounded off the integer superior
  - `cos(e1)` : cosine
  - `sin(e1)` : sine
  - `tan(e1)` : tangent
  - `acos(e1)` : arccosine
  - `asin(e1)` : arcsine
  - `atan(e1)` : arctangent
  - `cosh(e1)` : hyperbolic cosine
  - `sinh(e1)` : hyperbolic sine
  - `tanh(e1)` : hyperbolic tangent
- `ite(e1, e2, e3)` (where `e1` is a boolean expression and `e2` and `e3` two same type expression) has for value `e2` if `e1` (`e1 == true` or `e1 != 0`) then has for value `e3` . The following syntax `e1 ? e2 : e3` defined in C language C also exists for this operator ;
- `time()` is the current time in the simulation.
- `story()` is the number of current simulation.
- `delay()` is the duration of a history (equivalent to a constant equal to the option `duration` in Petri net)
- `random()` is a random number between 0 and 1.
- `@()(e1,...,en)` (where `e1,...,en` are boolean expressions) counts the number of « true » expressions in the list.

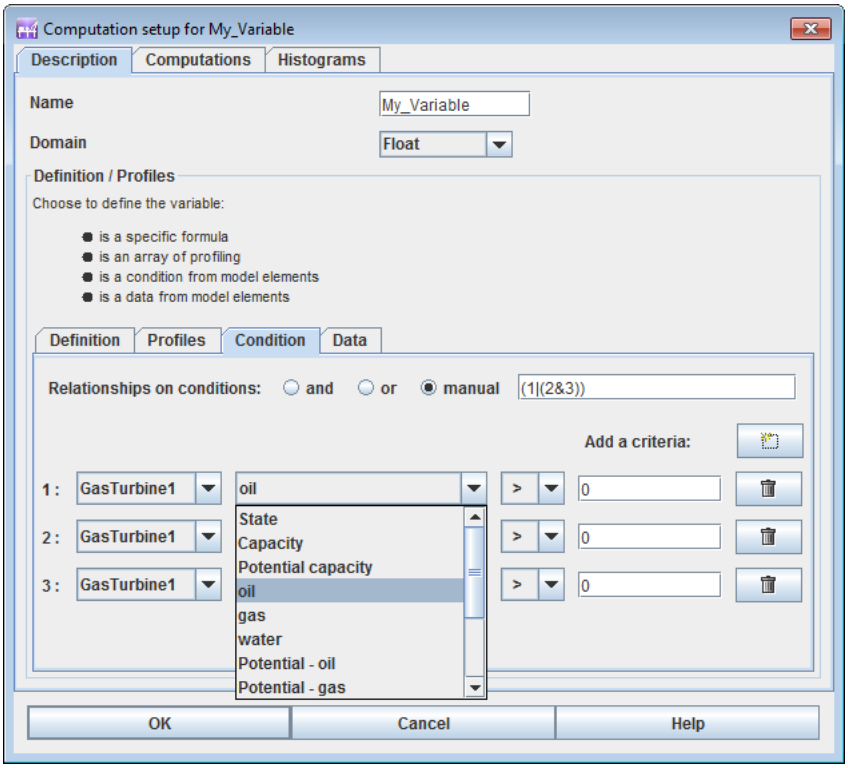
For more information consult the user manual Moca-RP

8.2.3. Profiles



A variable bound(connected) to a profile has for value the current profile chosen by the user.

8.2.4. Condition



This tab enables to link a variable with 1 or several conditions from elements of the model. This variable can be based on:

- component state:

  - 1 : Work
  - 2 : By-Pass
  - 3 : Degraded Failure
  - 4 : Restart
  - 5 : Stand-By
  - 6 : Preventive Maintenance
  - 7 : Critical Failure
  - 8 : Repair
  - 9 : SIMOPS
  - 10: Preservation
  - 13: Hidden Failure
- on a **capacity** : real current capacity of production;
- on a **potential capacity** : current potential capacity of production;
- on a **flow** (oil, gas, water, etc...): real flow in the component output;
- on a **potential flow** (huile, gas, eau, etc...): potential flow in the component output;
- on a **demand** (oil, gas, water, etc...): real component demand;
- on a **potential demand** : potential component demand;
- on a **current level of the tank** : current level of the tank.
- on a **PM_init** : preventive maintenance initialization.

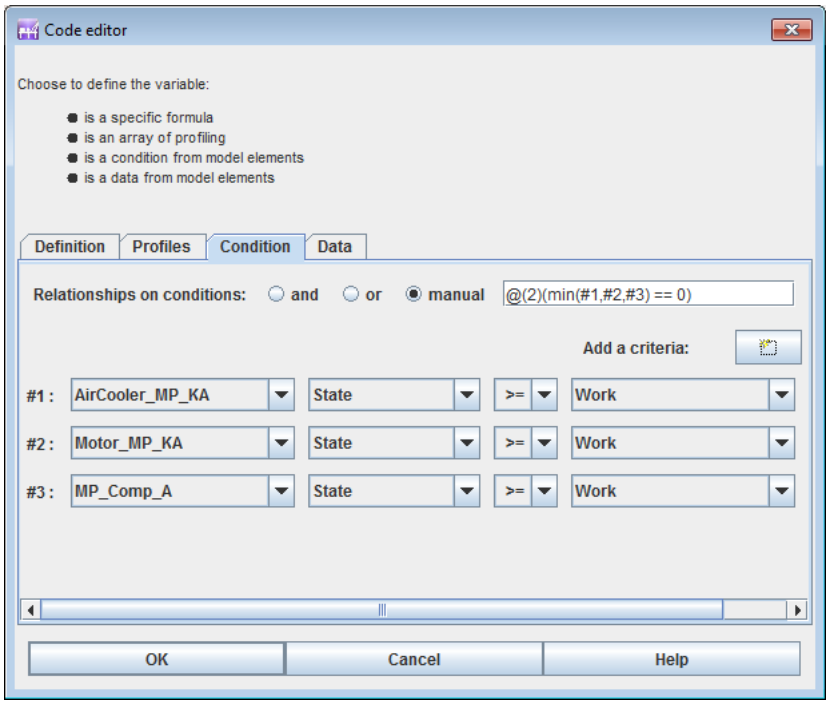
If **Optimize export for serial blocks** option is checked in **Tools - Application (document) options** values displayed for the flow (potential flow, demand, potential demand) are evaluate for all components in series and not only for the concerned equipment.

Relationship on the conditions are:

- The boolean operator **AND** ;
- The boolean operator **OR** between all the conditions ;
- **Manual** expression using:
  - & for the operator **AND** between all the conditions ;
  - | for the operator **OR** ;
  - unitary operator: `!` , `not`
  - Equality / inequality operator: `==` , `!=`
  - Comparison operator: `>` , `>=` , `<` , `<=`

The priorities are to be defined by means of brackets.

8.2.5. Data



This tab enables to connect a variable to 1 or several elements of the model. These variables can based on:

- component state:
  - 1 : Work
  - 2 : By-Pass
  - 3 : Degraded Failure
  - 4 : Restart
  - 5 : Stand-By
  - 6 : Preventive Maintenance
  - 7 : Critical Failure
  - 8 : Repair
  - 9 : SIMOPS
  - 10: Preservation
  - 13: Hidden Failure
- on a **capacity** : real current capacity of production;
- on a **potential capacity** : current potential capacity of production;
- on a **flow** (oil, gas, water, etc...): real flow in the component output;
- on a **potential flow** (huile, gas, eau, etc...): potential flow in the component output;
- on a **demand** (oil, gas, water, etc...): real component demand;
- on a **potential demand** : potential component demand;
- on a **current level of the tank** : current level of the tank.
- on a **PM_init** : preventive maintenance initialization.

If **Optimize export for serial blocks** option is checked in **Tools - Application (document) options** values displayed for the flow (potential flow, demand, potential demand) are evaluate for all components in series and not only for the concerned equipment.

If there are several data, the relationships between them are an expression composed from model data (ex: #1 + #2).The expression can use Moca functions (ex: min(#1,#2), see Section 8.2.2, “Variables definition” for all expressions).All line numbers used in the expression will be replaced by the corresponding data line.

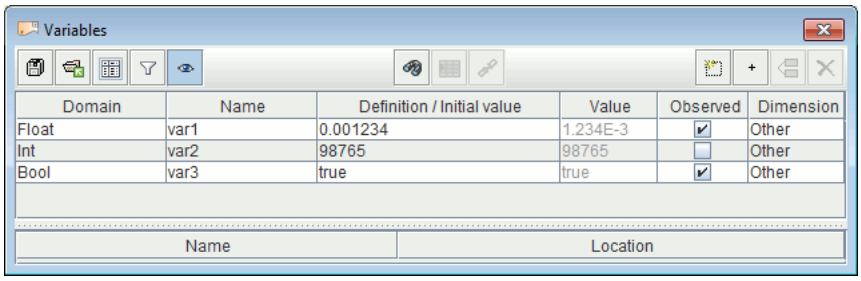
8.3. Statistics and Setup of Variables

In addition to the mean marking of the places and of the mean number of times each transition is fired, the simulation can compute a certain number of additional statistics. Statistical results can be obtained on any of the

model's variables or combination of variables. To do this, a variable must be declared as "observed". When a variable is "observed", a statistic state (Moca meaning) is created for computation.

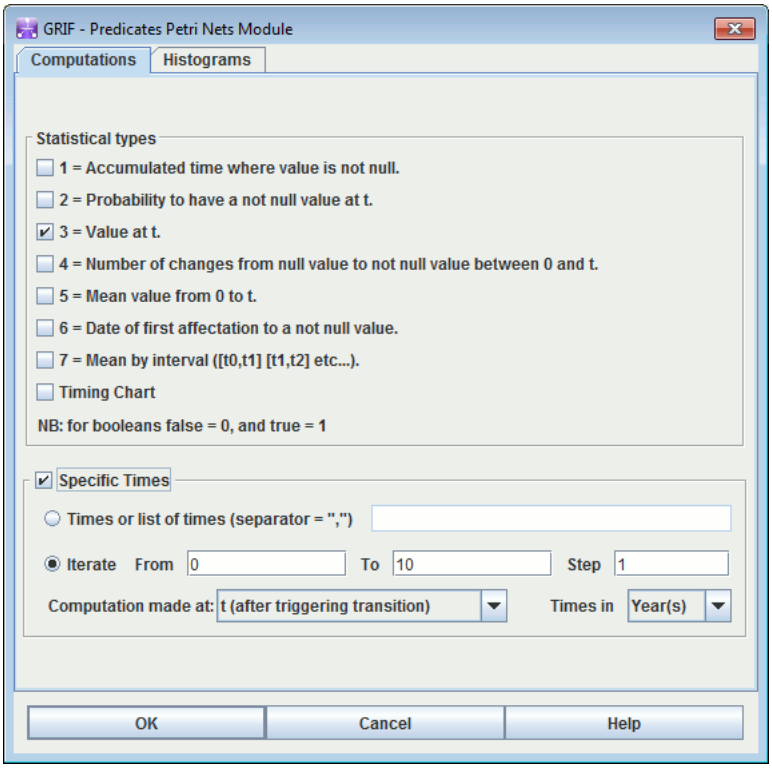
8.3.1. Definition of statistic states

A statistic state is defined like an "Observed" variable. We must initially define the statistic states we wish to observe. To do this, we have to edit variables from the model: either thanks to the **Data and computations - Edit Variables** menu, or thanks to the **Edit Variables** tab. Then, all we have to do is to set **Observed** property of a variable in order to make it a statistic state.



8.3.2. Configuration of statistic states (or variables)

Once variable is observed, we have to configure them by specifying types of computations and computation times to be carried out on them, ... To do this, do a right-click on the variable and select **Configuration of computation**.



The window to edit observed variables (statistic states) is made of two tabs. The first tab is for computation, the second one is for histograms. These two tabs enables configuration of **Types of statistics**, **Computation times** and **Histograms**.

8.3.2.1. Types of statistics

Types of statistical computations are the following:

- **1 - Cumulated time where value is not null:** This is the mean time in which the statistic state is different to 0 on a history.

Purpose: Mainly used to compute mean availabilities during a history.

- **2 - Probability to have a not null value at t:** This is the probability that the statistic state is different to zero at the end of the history.

  Purpose: Among other things, used to compute the mean availability at the end of a history or compute the reliability (to find if the failure state - which is an absorbent state - is present at the end of a history).
- **3 - Value at t:** This is the mean value of the statistic state at the end of the history.

  Purpose: This type of computation can be used to compute the occurrence of specific event during a history.
- **4 - Number of changes from null value to not null value between 0 and t:** This is the mean number of times during a history that the statistic state changes from a zero value to a non-zero value.

  Purpose: This type of computation can be used to compute the occurrence of specific events during a history.
- **5- Mean value from 0 to t:** This is the mean value of the statistic state on the duration of the history.

  Purpose: Among other things, used to compute the production availability.
- **6- Date of first affectation to a not null value:** This is the mean instant from which the value of the statistic state changes from zero to a value different to zero.

  The "uncensored data" field gives the number of histories for which the simulation has been able to retrieve a value. For this mean statistical result to have a meaning, we must verify that a value has been retrieved for each history (uncensored data = number of simulated histories).

  Purpose: Used to obtain information about the mean instant when a system fails for the first time (reliability computations, estimation of the mean trouble-free operating time, etc.).
- **7- Mean by interval ([t0,t1] [t1,t2] etc ...) :** Expression average value of the state for every time interval

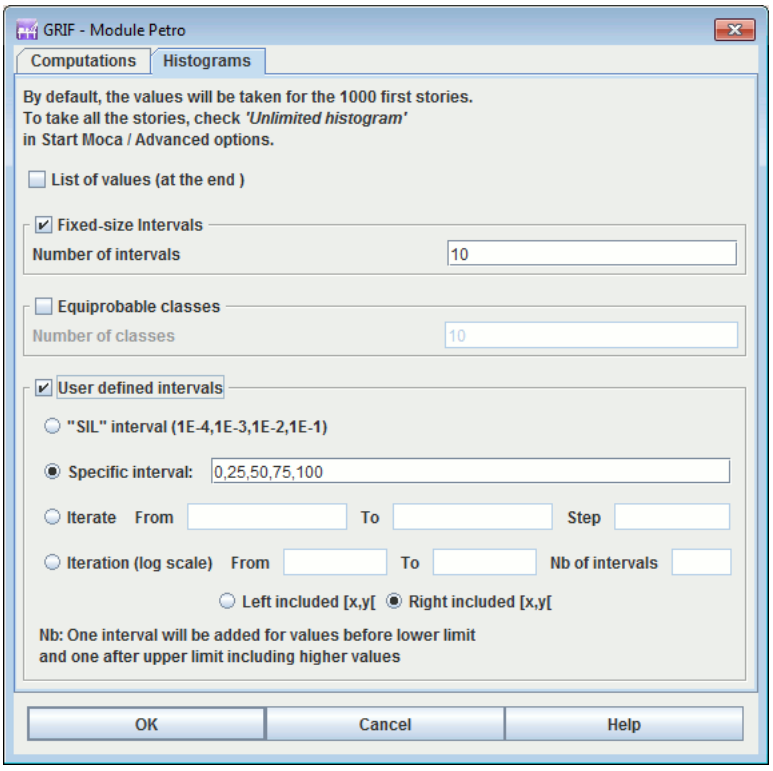
  Was integrated to display the yearly average production of production. Must be used with a list of time (otherwise, the result is identical to a type of statistic 5). The difference between 2 times must be strictly superior in epsilon (a list of including time t and t-epsilon will make impossible the simulation launching).
- **8- Sum value from 0 to t :** represents the sum of the statistic state over history duration.
- **9- Sum by interval ([t0,t1] [t1,t2] etc ...) :** represents the sum of the statistic state between two calculation period.
- **CHRO - Timing chart** Traces the instantaneous value of the variable (as 3) but automatically calculates the "useful" points without taking into account the specified times.

8.3.2.2. Computation times

Two possibilities are available to define computation times:

- **List of times:** the computations will be performed for the values of t given in this list. Separator is comma.
- **Iterate Form A to B step C:** the computations will be performed for values of t ranging from A to B with a step of C. You can also choose is computations are made before or after transitions triggering.

8.3.2.3. Histograms



Computations made previously gives mean value for many histories. **Histograms** enable to know how values are distributed during histories. (Cf. Moca User manual for further information)

- **List of values:** provides value at the end of each history.
- **Fixed size intervals:** provides the way value are distributed by cutting intervals of values in X intervals which size is equal.
- **Equiprobable classes intervals:** provides intervals whose probability to contains a value at the end of a story id the same.
- **User-defined Intervals**

  Bounds of intervals can be defined as follows:
  – Automatically defined limits for SIL
  – Manual definition of limits (separate by commas)
  – "Iteration": user choose lower limit and upper limit, and the size of intervals.
  – "Iteration (log scale)": user choose lower limit and upper limit, and the number of intervals. Size of intervals will be computed in order to have same-size-intervals, on a logarithmic scale.

  Moreover, two limits are added at minus infinity, and plus infinity, in order to have a chart containing every history of the simulation.

  When limits are chosen, user has to choose between "left included" or "right included".

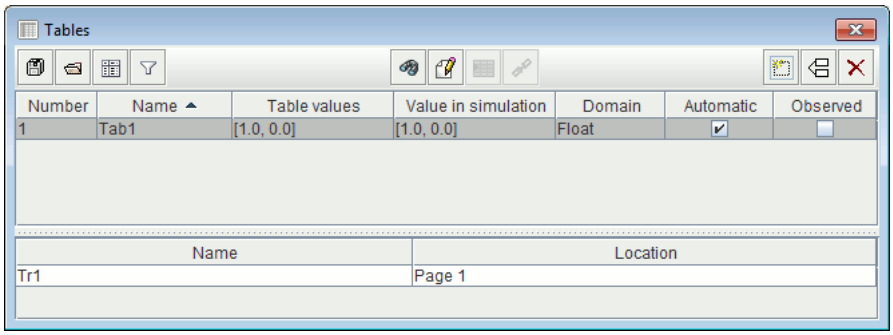
  IEC 61508 specifies "left included" intervals for SIL

9. Table values

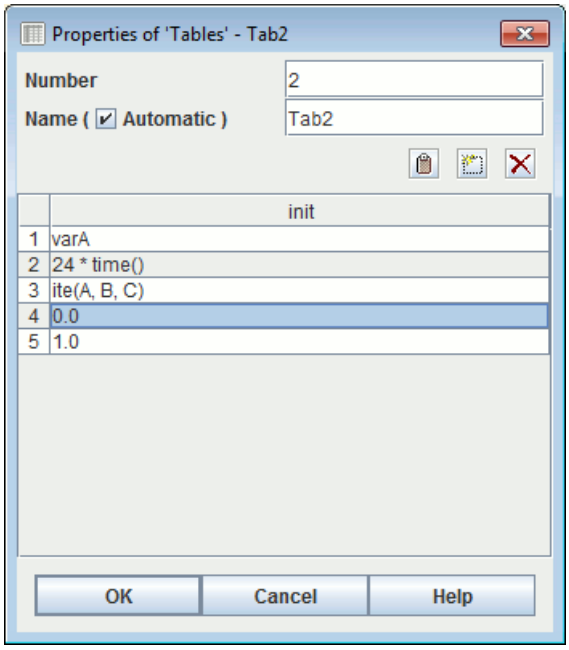
It is possible to create table values. This table have the same behavior as variables, they can be constructed in order to be read in the model, or to store temporary information that will be modified by a transition.

9.1. Creation

Table tab is used to define this table.



Tools bar enables the common data table actions (Section 1, “Description of the Tables”). "Add" button opens a window to create a table:



A table is identified by a number and a name. It contains a list of values that can be numerical or variable expressions.
Others fields are available in the column manager. Some fields are masked by default. To show the others columns see Section 1.2, “Column manager”.

The additional columns accessible are:

Table values	displays value imput by user.
Value in simulation	displays updated values at current simulation time.
Domain	enables to give a domain defintion (Reall, Boolean, Integer) at the table value.
Observed	enables to activate a statitistic computation for each table index. For more information about statistic computation see Section 8.3, “Statistics and Setup of Variables”.

9.2. Use

The value of a table cell can be changed dynamically during the calculations. To do this, the following syntax can be used in the code of a transition assignment:

Tab1[1] = 2; / Assigns first cell of table Tab1 */*

During simulation, the value of the table can be displayed using a **Dynamic field**:

$data.table.name(Tab2).values

An table can also be used as a function argument:

m = min($Tab1);

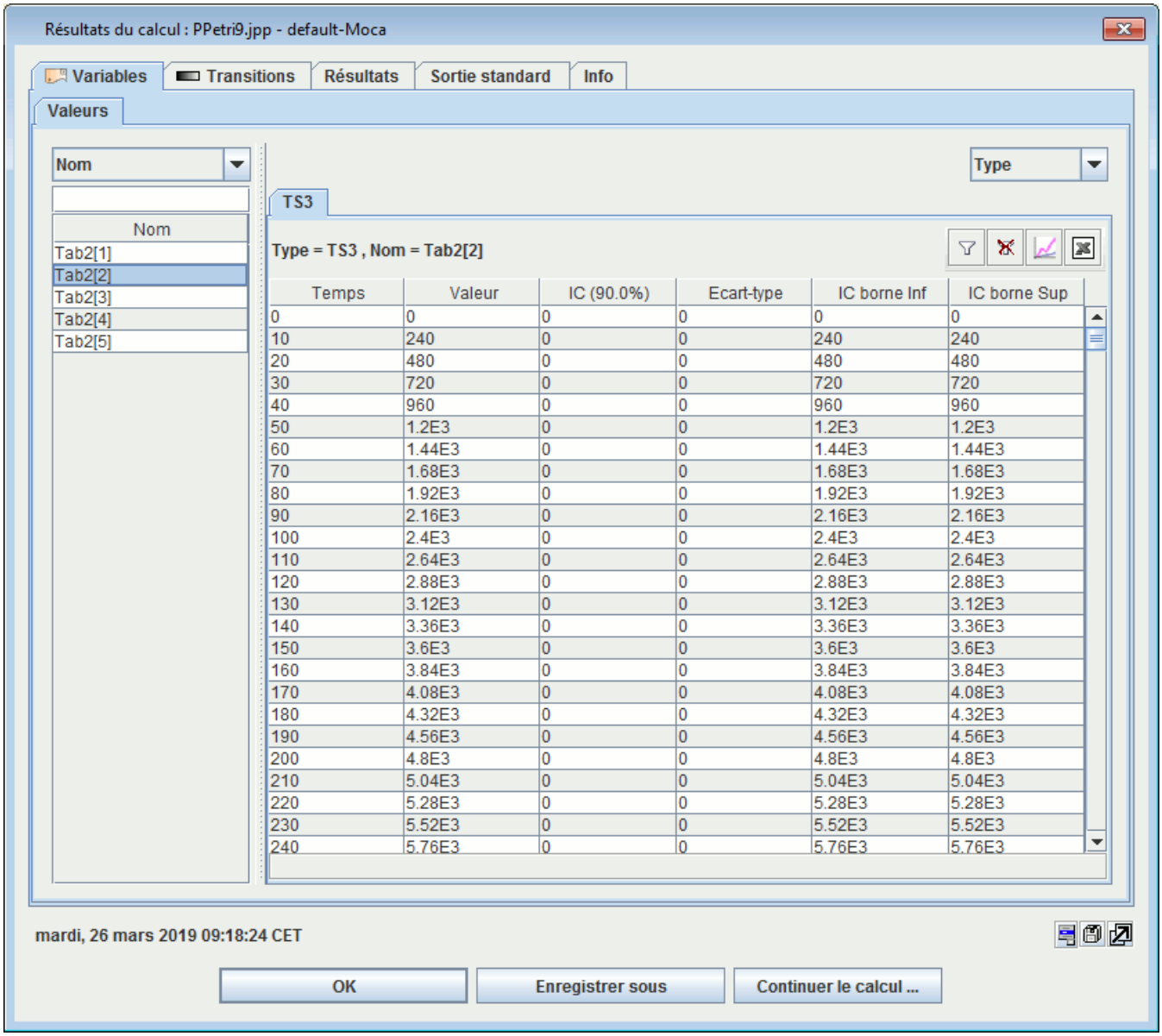
A variable size table can be used. It is the simulation assignments that change the size. This type of table can be used to manage pile or element queues. To use such tables, it is necessary to create them with an empty size (no elements on it).
Assignment example:

```
/* Assigns the 50th cell of the table with the value 1. */
/* The cells not yet initialized [1-49] will take the value 0 */
DynaTab[50] = 1;

/* Increases table size by 1 and affects this cell with value 2 */
DynaTab[DynaTab.size() +1] = 2;
```

9.3. Computations

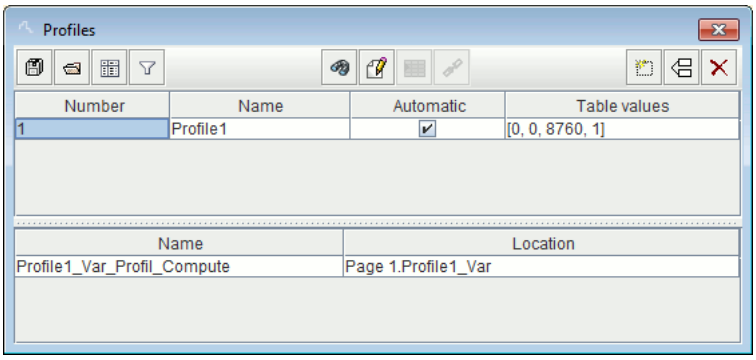
When statistical computations are requested for a table, the statistics are calculated for each index in the table. The results will be displayed in the **Variables** tab of the detailed results window.



10. Profile tables

Profile tables are used to change a variable over time. Variables defined by a profile table are called profiled variables.

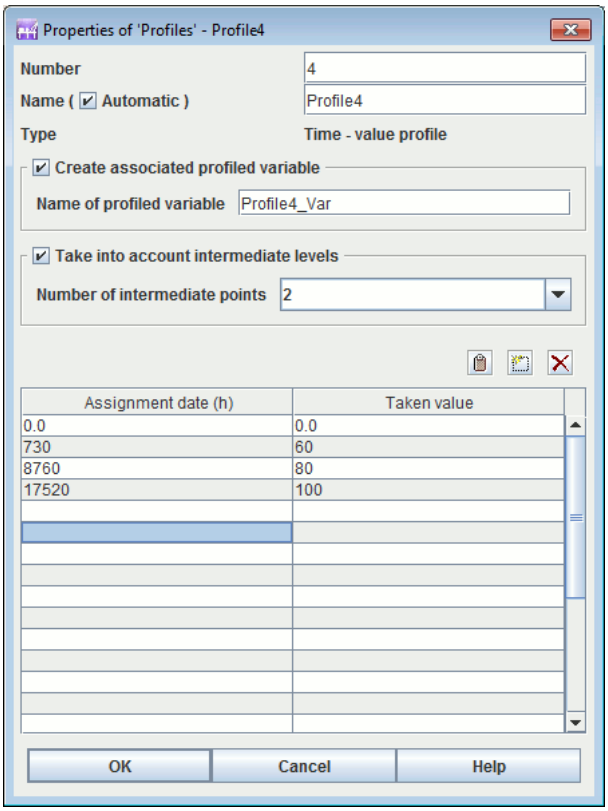
The **Profile** tab enables the user to define his profile tables.



The toolbar enables the classic actions of the data tables (Section 1, “Description of the Tables”). The "New" button opens the profile table creation window.
Several types of definition are possible:

- **Time - value profile** : Table with 2 columns, one for the time and one for the values to be considered at each time.
- **Time - value profile (looped)** : Table with 4 columns: Phase number, Duration, Value and Next phase. Profile entry is done by defining phases. A phase (table row) has a duration in hours and a value that will be taken during this time. Once the time has elapsed, the phase defines as the next phase will be considered.
- **Flow profile** : Table with n + 1 columns, one for the time and n for the number of flows. This table enables the input of flow profiles needed on sources or outputs. The values entered for the flows are throughputs.
- **Flow profile (looped)** : Table with n + 3 columns: Phase number, Duration, Next phase and n for the number of flows. The values entered for the flows are throughputs.
- **Flow profile by quantity produced** : Table with n + 1 columns, one for the quantities consumed and n for the number of flows. This type of profile is different from the previous ones in the sense that no time or duration is provided. Flow variations occur when a certain amount of flow has been consumed.
  To do this, you must first define a reference flow that will be used to calculate the consumption.
  Note: The same flow profile of this type, used on 2 different sources will lead to 2 different profiles (as a function of the consumptions of the respective sources).

Below is the Time-Value profile creation window:



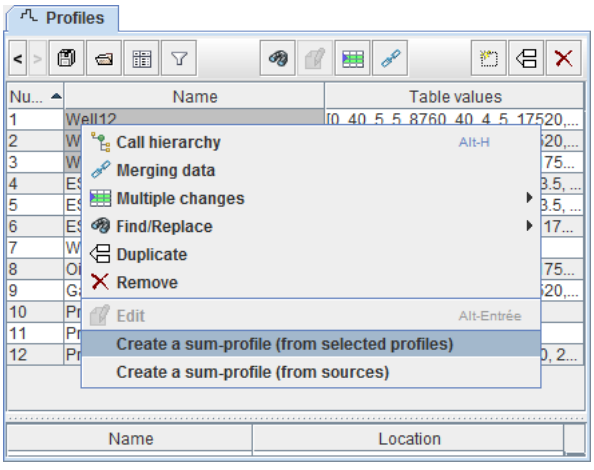
On the Time-Value profiles, it is possible to create the profiled variable at the same time as the creation of the table.

If the variable already exists, it can be attached to its profile in a second step via the definition of the variable (Section 8.2.3, “Profiles”)

It is also possible to linearize the profile table over time. To do this tick **Take into account intermediate levels** and define the number of intermediate points that will be generated between each time interval. Note: This feature is not available for flow profile by quantity produced.

Flow profiles can also be defined through an Excel file. To do this, use the **Import profiles Excel** add-on. In the add-ons information column, double-click on the cell, the window of the import plugin Excel then opens to choose the profiles to use. For more information on using this add-on, refer to Section 16, “Import of profiles from an Excel file”

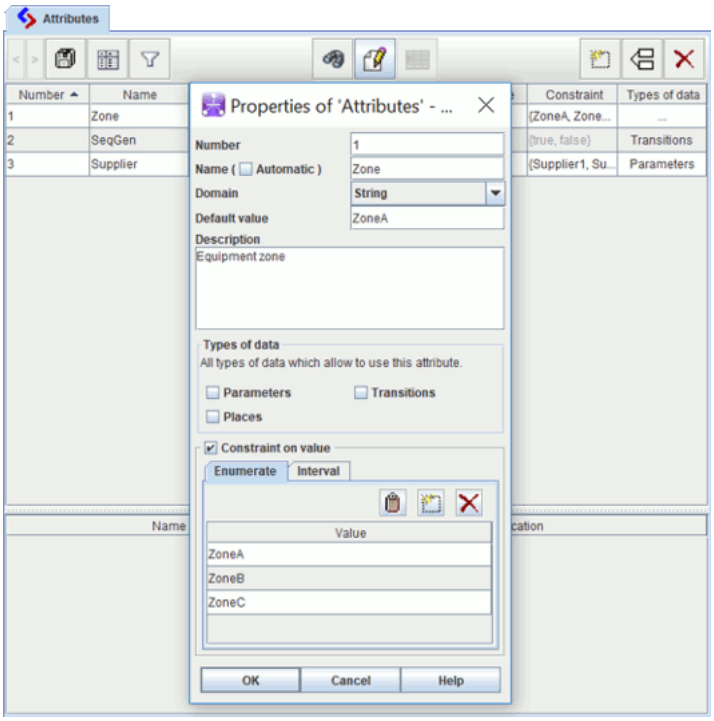
Profiles can be summed together, this function is available in the data table **Profiles** :



11. Attributes

11.1. Creation

The **attribute** tab enables the user to create attributes that are used to qualify elements defined on system.



The attribute properties are the following ones:

- name;
- domain ;
- default value;
- type of data: to choose where apply the attribute;
- constraint.

The domain type can be of the following values:

- **boolean**: This kind of attribute is a boolean;
- **integer**: This kind of attribute is used to affect an integer value;
- **float**: This kind of attribute is used to affect a float value;
- **string**: This kind of attribute is used to affect a free text.

In **Constraint** field, user can enter a constraint on the attribute to ensure the proper use of the attribute in the model.

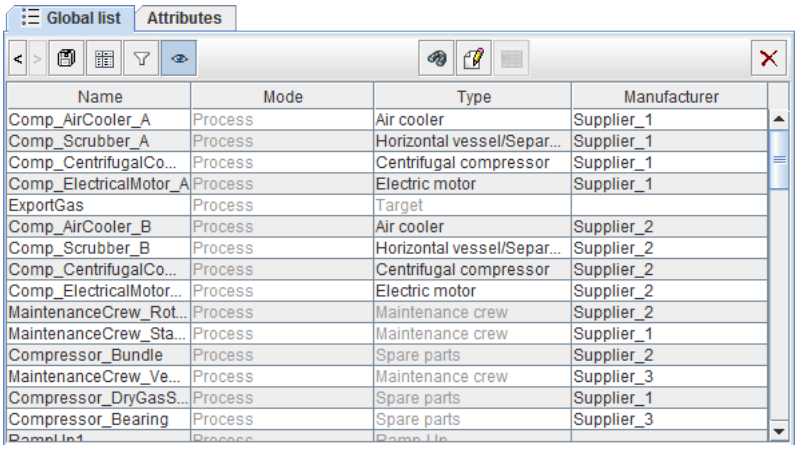
In addition, the attributes of **float** or **integer** type have a **Constraint type Enumerate** or **Interval**.

11.2. Use of the attributes

In a Petro model, it is possible to associate attributes:

- on equipment;
- on spare parts;
- on maintenance teams.

For that, after creation of the attribute, it is enough to go to the **List global** tab to associate the item and the attribute.



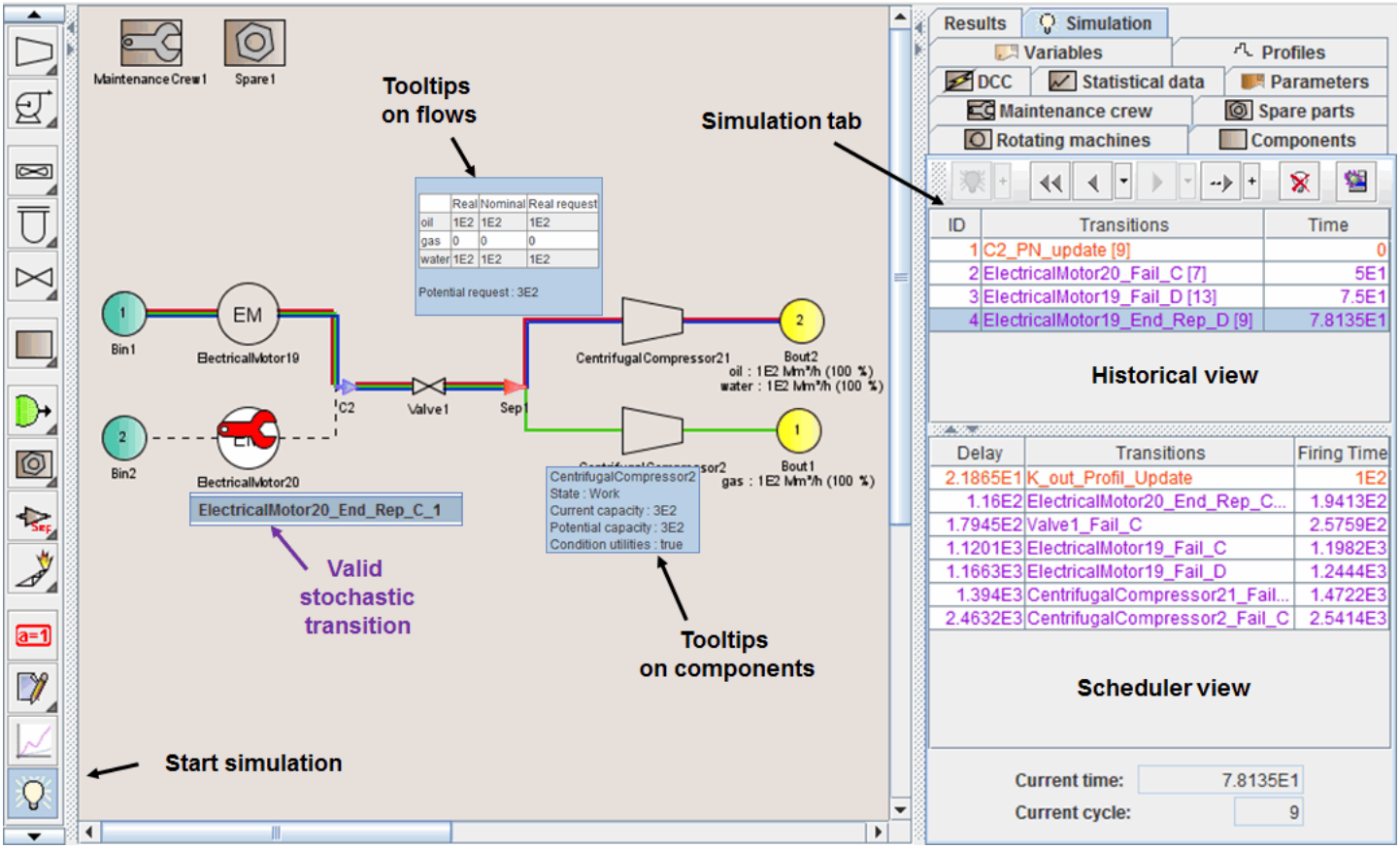
Name	Mode	Type	Manufacturer
Comp_AirCooler_A	Process	Air cooler	Supplier_1
Comp_Scrubber_A	Process	Horizontal vessel/Separ...	Supplier_1
Comp_CentrifugalCo...	Process	Centrifugal compressor	Supplier_1
Comp_ElectricalMotor_A	Process	Electric motor	Supplier_1
ExportGas	Process	Target	
Comp_AirCooler_B	Process	Air cooler	Supplier_2
Comp_Scrubber_B	Process	Horizontal vessel/Separ...	Supplier_2
Comp_CentrifugalCo...	Process	Centrifugal compressor	Supplier_2
Comp_ElectricalMotor...	Process	Electric motor	Supplier_2
MaintenanceCrew_Rot...	Process	Maintenance crew	Supplier_2
MaintenanceCrew_Sta...	Process	Maintenance crew	Supplier_1
Compressor_Bundle	Process	Spare parts	Supplier_2
MaintenanceCrew_Ve...	Process	Maintenance crew	Supplier_3
Compressor_DryGasS...	Process	Spare parts	Supplier_1
Compressor_Bearing	Process	Spare parts	Supplier_3

12. Interactive simulation

12.1. Introduction

One of the most important characteristics of the GRIF interface is that it enables the user to manually simulate the behaviour of the network s/he has just created. It is therefore easier to understand, debug or explain a model.

When the simulation is launched, transitions can be fired to understand how the model reacts, to go back a stage, to replay a sequence of a given event, to view the status of the components, the quantities of flow circulating, the value of each variable, etc. at any time.



There are several ways of starting a simulation:

- Use the menu **Mode - Simulation**,
- Click on the corresponding icon in the vertical toolbar.
- Click on the corresponding icon in the simulation panel toolbar.

Once the simulation mode has started, the "valid" transitions in the initial status of the Petri net appear in the scheduler.

The function that gives the user a major advantage is the possibility of "manually firing" the valid transitions for components from the model **data input area**.

- Select a component.
- Right click on it to display the valid transitions.
- Select a transition to be fired.

This fires the transition and all the subsequent instantaneous transitions (if the option is selected). Note that in the Petro model, only the stochastic transitions can be fired from a component.

In the example, component **ElectricalMotor20** is being repaired. The dotted lines show that no flow is circulating through the connections. A right-click on the component displays the transition that enables the component repair to be completed.

12.2. Simulation panel

The interactive simulation panel has four parts:

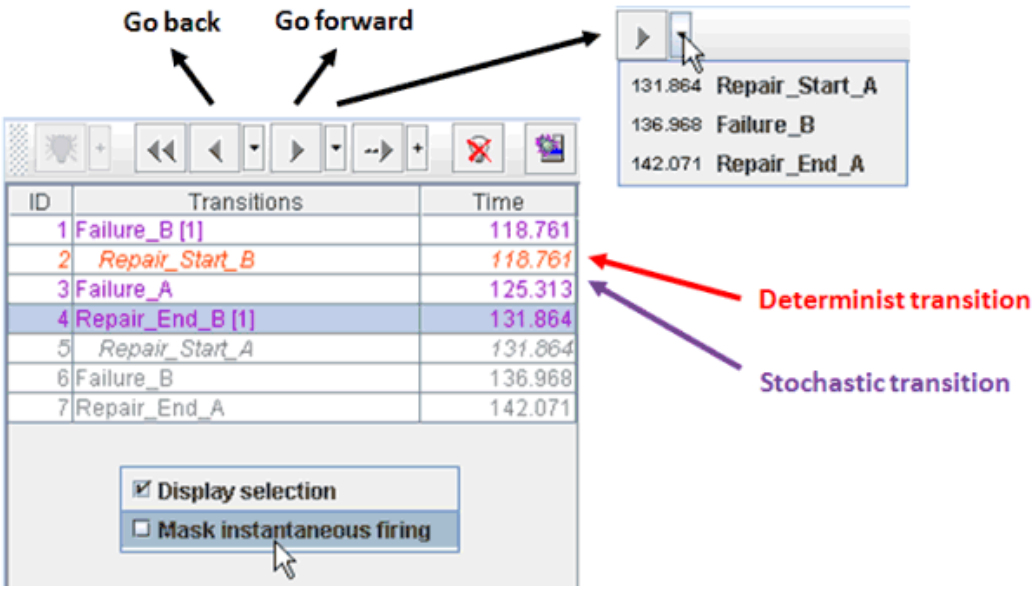
- Right at the top, a **toolbar** groups together the functions that enable a user to start, stop, configure and play the simulation.
- Just below is a **history** of the fired transitions.
- After that is the **scheduler**, which contains the list of the fireable transitions ranked in order of firing date.
- Right at the bottom, a panel displays the **current time** and the **current cycle**.

The toolbar contains the following functions:

	Gives access to the interactive simulation mode.
	Opens a simulation configuration window. This point is covered in the section Section 12.5, “Configuring the simulation”.
	Goes back to initial state.
	Goes back one step.
	Rolls back a fired transition. The simulator will go back to the step before the selected transition.
	Replays a step in the fired transition history.
	Replays the fired transition history up to the selected transition.
	Plays the next transition in the scheduler.
	Goes forward in the simulation to a certain time or to the firing time of a given transition. The simulator will stop one step before the transition or time selected.
	Exits simulation mode
	Modifies simulation options during the simulation. See section Section 12.5, “Configuring the simulation”.

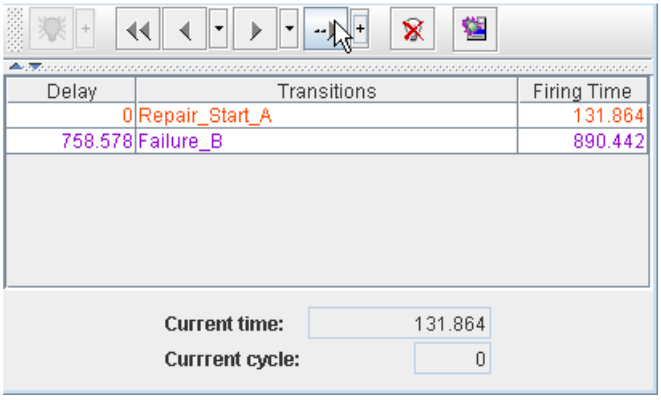
12.3. Simulation history

The simulator contains a panel that displays the history of fired transitions. The user can "explore" the history or take things back a step to modify the running order. A right click on a transition in the history displays a contextual menu that enables the user to display/hide the instantaneous transitions among other operations.



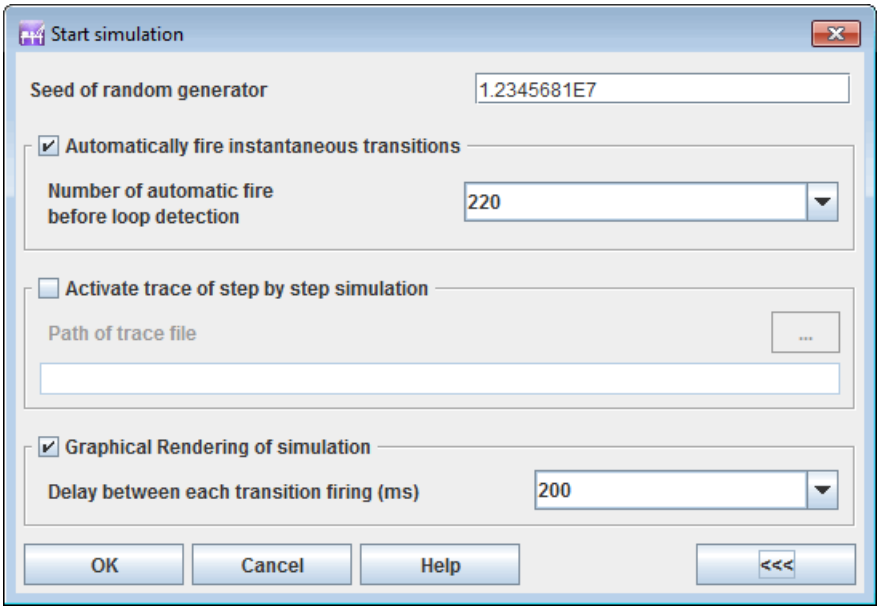
12.4. Simulation scheduler

The simulator includes a panel which displays the scheduler of the transitions to be fired. You can display a transition from the scheduler in the model view by right-clicking on it.



12.5. Configuring the simulation

The simulation options can be accessed using the (+) button located next to the start simulation button, in the simulator toolbar. They can be configured when the simulation is started.



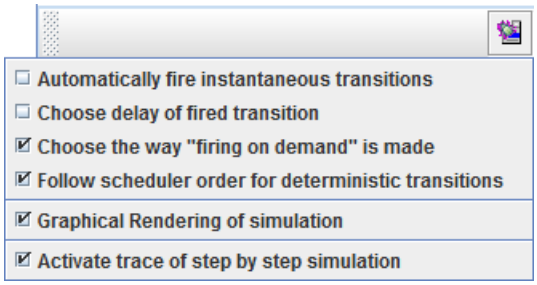
The **seed** to be used by the simulator can be entered here.

The tick box **Automatically fire instantaneous transitions** automatically plays the transitions in Dirac delta function 0. In this case, the user must indicate how many fire operations the simulator can do before considering that there is a loop in the simulation.

The tick box **Activate trace of step-by-step simulation** saves all the transitions fired during the simulation in a file.

The tick box **Graphical rendering of the simulation** enables the user to follow the step-by-step progress of the simulation in the model input area. S/he can then enter the length of time between each transition firing to slow down or speed up the simulation.

Other options can be modified during the simulation



The option **Choose delay of fired transition** enables the user to choose the date on which the transition will be fired (applies only to stochastic transitions).

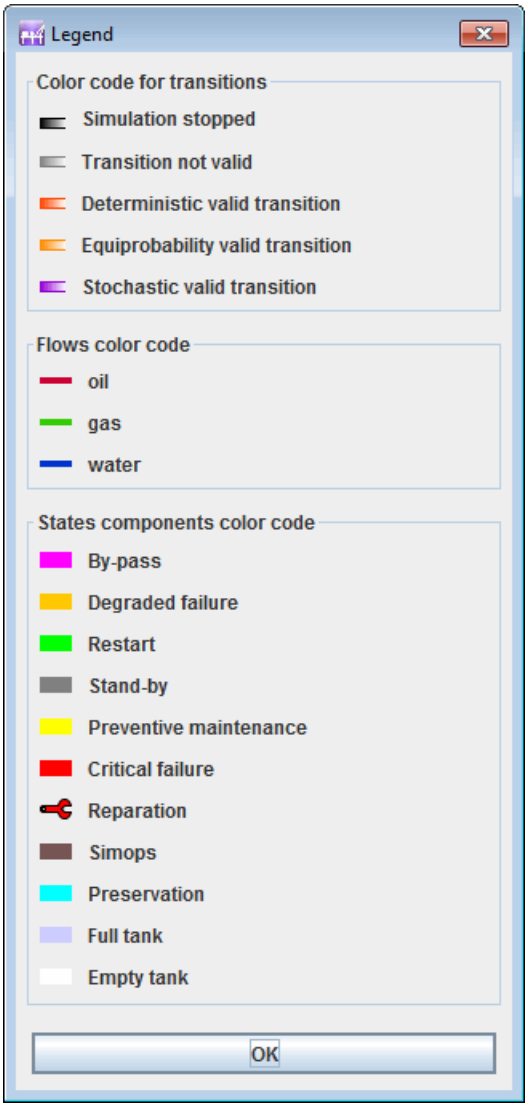
The option **Choose the way "firing on demand" is made** enables the user to automatically fire or not the transitions that use the firing on demand rule. If the option is ticked, a dialogue window will prompt you to select the location that will receive the token if this kind of transition is fired.

If the option **Follow scheduler order for deterministic transitions** is unticked, it is possible to fire deterministic transitions before the date initially planned.

12.6. Colour code / Legend

During a simulation, the transitions active at a given time are displayed in specific colours according to the type of transition.

Below are the colour codes used in the simulator panel for the transitions, and used in the view for the flows and the component states.

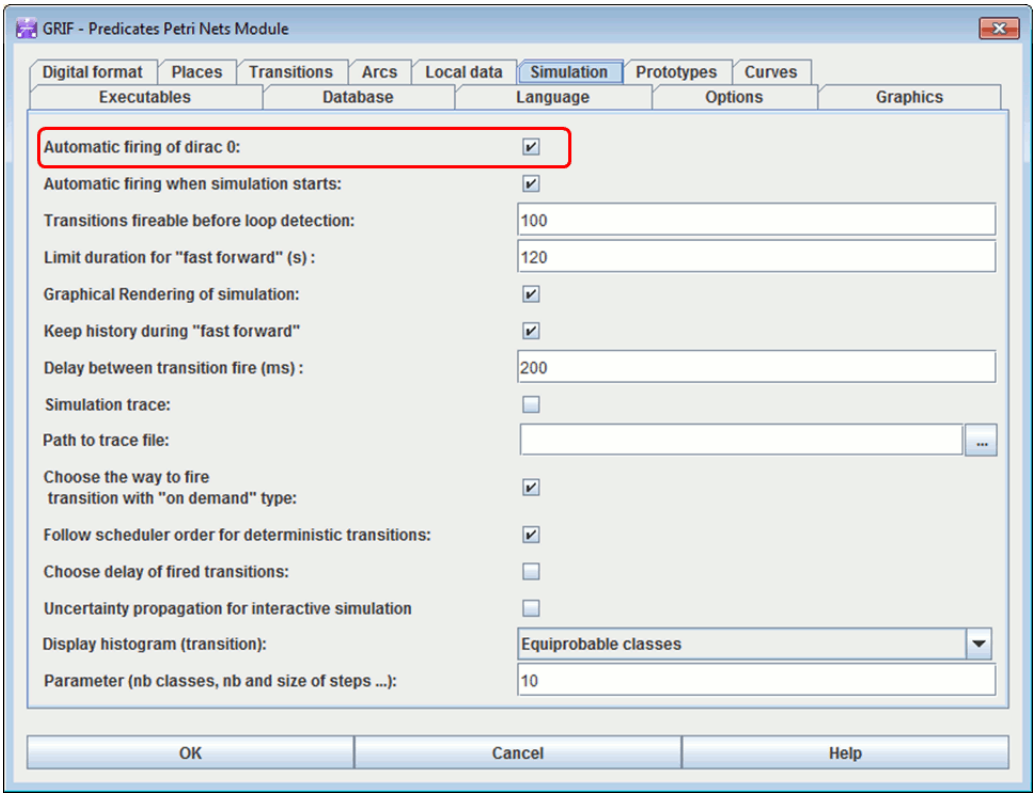


12.7. Automatic firing of transitions with zero delay

The automatic firing of the transitions in simulation mode automatically fires the transitions which have a zero delay (Dirac law with parameter 0) when they are valid. In the case where several Dirac law transitions are in "conflict", the transitions are fired according to their priority, then in chronological order of their creation on a page, then in increasing order of pages. This is how the simulation works when the computations are launched.

Remarque: The transitions with zero delay of a group are fired after those of the page where this same group figures.

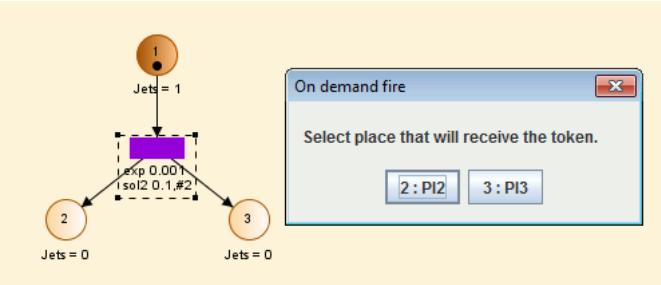
To enable or disable this function go into **Application Option - Simulation**:



12.8. Probability firing for transitions with fire on demand

In simulation mode, when we wish to fire a transition with "fire on demand", the token(s) must only move to a single place downstream. When you click on this type of transition a window is displayed where you have to enter the probability manually (it is normally a probability which is determined by a simulation).

The default value of this probability is 0.5.



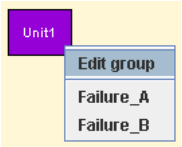
In the above example, if 0.5 is entered, the token will go into place no. 3.

12.9. Simulation with groups

There are two ways of firing one or more valid transitions that belong to a group:

- Enter the group to fire the transitions by clicking on it directly (left click on the group or right click then **Edit group**).

- Or run a fire operation simply using the list of valid transitions in the group and "sub-groups" (right click on the group).



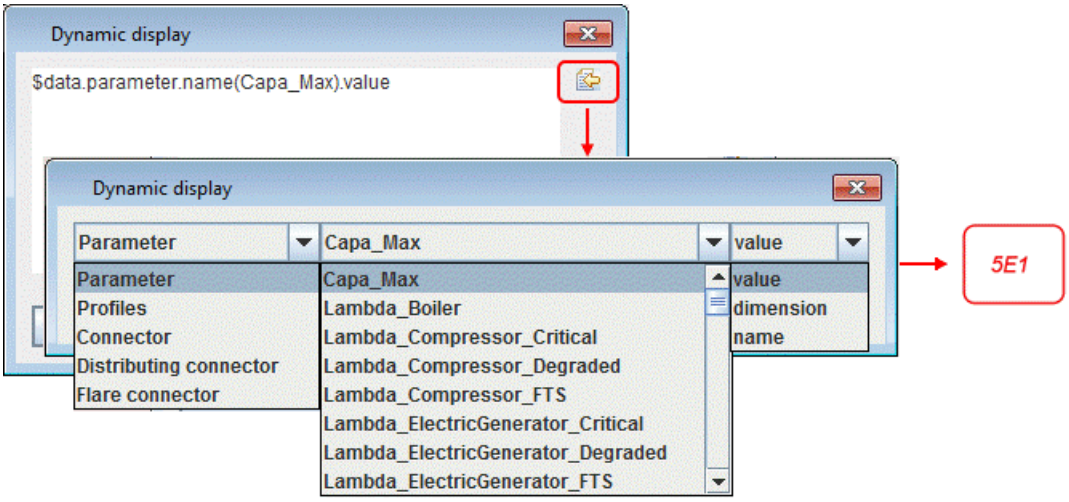
If we go back to the previous example with two components, then the **Failure** transition was effectively the only valid transition.

When transitions are fired within groups, it may prove difficult to determine the modifications that have been made. This is why it is useful to use the dynamic fields.

12.10. Dynamic fields

It may be useful to observe the change in the different parameters of the model. It is also useful to see a result next to its corresponding system. To do this, use dynamic fields by selecting the corresponding icon on the vertical tool bar:

The dynamic fields are a type of "improved comments". They can be used not only to enter words or phrases but also to insert model values or results.



If you want to display information about a data of the model, you must use the following syntax:

$data.'type of data'.'field used o search data'('value that the field must match).'information you want to display for the selected data'

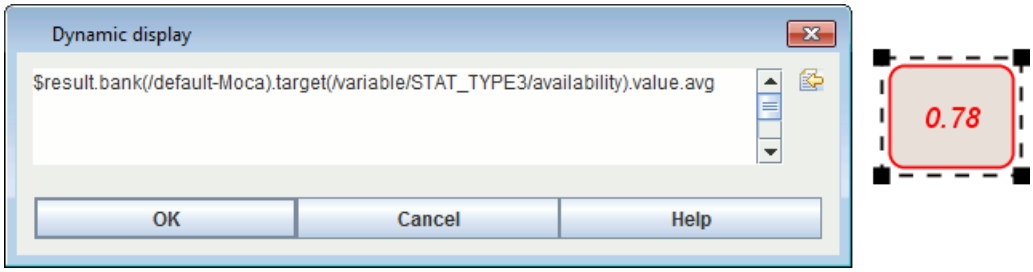
We can analyze the above windows as follows: I am looking for a "parameter" which "name" is 'Capa_Max", and I want to display its "value". When you type the first letters, a completion system helps to type script without error.

Button in right permits to enter complete expression but select what you want to appear.

If you want to display a result of the result-bank, the syntax is the following:

$result.bank('path in the bank').target('target result').'what you want to display'.'at what time'

We can analyze the next picture as follows: I am looking for a result which path in the bank is "default-Moca", I want results for "TS3 for 'available' valriable" and I want its value for the "last" time. If last is replaced by time(10) we obtain value at t=10.

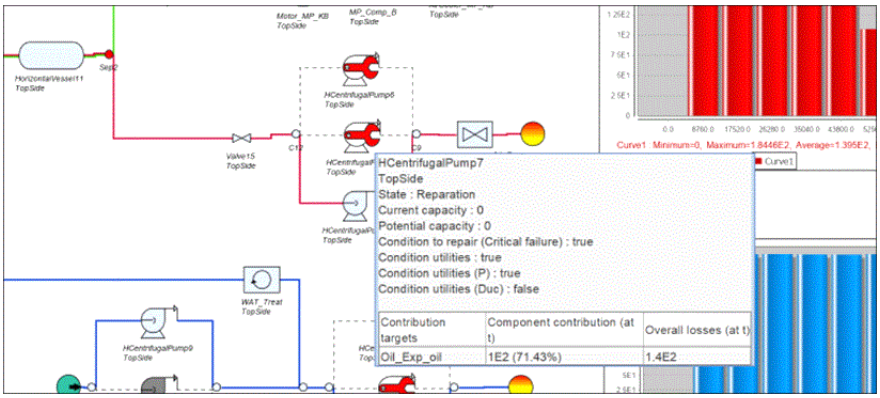


You can also display a summary of result. Replace 'what you want to display' by **summary**. In this case, **summary** is the last word of this script.

12.11. Contribution

During a simulation, it's interesting to know contribution of equipment failure on the loss of flow.

Below, tooltip enables to know the contribution of the element failure on the loss of flow.



13. Computations

13.1. MOCA computations

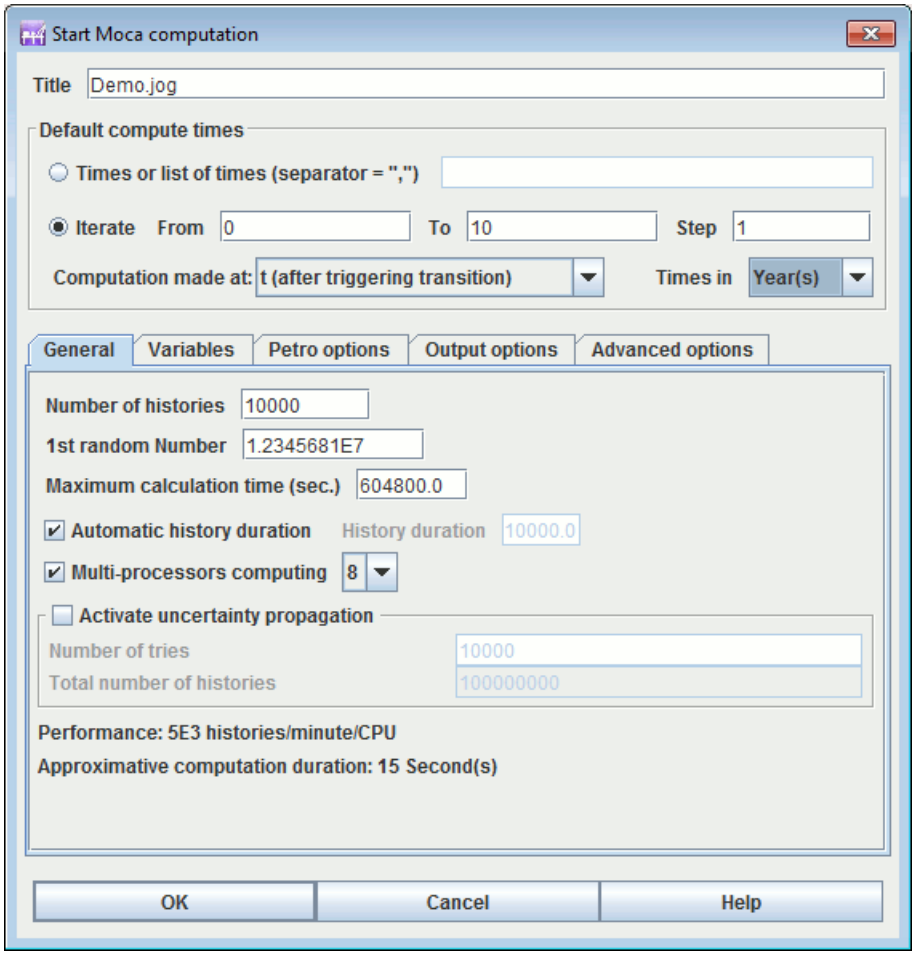
The computations using MOCA-RP V14 are performed in three main steps:

- general configuration of parameters;
- the launch itself;
- reading the results file.

13.1.1. Configuring the computations

The computation configuration window can be accessed in two different ways: either via menu **Data and Computations - Moca Data** or via **Data and Computations - Launch Moca ...** . The difference between the two is that, in the second case, the configuration step is directly followed by the computation launch step.

The configuration window which opens is called **General Information** :



This configuration window is divided into five parts:

1. **Title** : enables you to give a title to the results file.

2. **Default computation times for statistic states** :

   - **Iterate From** A **to** B **step** C: the computations will be performed for values of t ranging from A to B with a step of C.
   - **List of times** : the computations will be performed for the values of t given in this list.
   - **Computation made at** : by default, computations are made immediately after transition triggering, but you can do computation at t-Epsilon (just before triggering), or at both.

- **Unit** : default times unit is "hour". You can choose a unit that will be used for computation times. N.b. results are always in hours.

3. **General** :

- **Number of histories** : Number of histories (NH) to be simulated (each history has a time t indicated below).
- **First random number** : It is the seed of random number generator.
- **Maximum computation time** (MT): The computations are stopped and the results are printed even if the requested number of histories has not been reached.

the unit of time (MT) is the second.

- **Automatic history duration** : If this box is checked, GRIF will compute history duration using computation time of variables and statistical states. If not, user can choose a specific **History duration**
- **Multi-processors computing** Enables (or not) the multi-processor computing (when available).

4. **Advanced options** : used to configure the advanced options.

- **Loop detected when the number of transitions firing at the same time is grater than** : You can choose the limit of transitions fired at the same time before loop detection.
- **Max number of loops while()** : if using in a transition.
- **Continue calculation if errors** : If an error is detected, the current history is stopped and we pass to the following one without stopping the calculations.
- **Display seed history** : Display or not the seed used by the random number generator.
- **Use old seed management (version < 2018)** : From 2019, a new random number generator was developed to increase the number of playable story without seed repetition. It is recommended that you use this option only for backward compatibility for your old documents.
- **Computes every nodes** : Useful option in the boolean modules to have the calculations on all the nodes (by default the calculations are performed only in the top event).
- **Delay re-computation for dynamic transitions** : Selects the method for recalculating the delay of transitions marked as dynamic. For more information on the possible choices, refer to User manual Moca14.10.3 [2.7.10. Dynamic transitions]

5. **Statistics**
   - **Confidence interval** : Allow to choose the confidence interval at 90%, 95%, 99%, 99.9% and 99.99% (by default 90% is checked). .
   - **Disable statistics on places and transitions** : Allow not to display the statistics on places and transitions.
   - **unlimited histogram** : In case of histogram all the stories are taken into account not only the 1000 first ones.
   - **Do not save 0 value in history** : Option to not save the historic if the statistic value is 0.
   - **Activate uncertainty propagation** Enables (or not) the uncertainty propagation computations (two-stage simulation): in this case we must specify the number of sets of parameters "played" (the real number of histories thus simulated will be the "number of sets of parameters x number of histories to be simulated" and will be displayed in the "Total number of histories" field).
   - **Histogram with every history of the tries (with uncertainty propagation)** : Allows to display the histogram with every history of the tries.

6. **Variables** : This tabs reminds computing configuration of variables. If document contains some statistical states, another tab is available.

7. **Petro options** : used to configure specific Petro options:

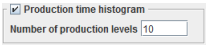
- **Statistic on components and resources** :
  - **Resources management** . For repair teams and spare parts several results are provided such as time without repair crews available, accumulated time of activity, parts used, etc...
  - **Failure modes** . According to failure mode, many details are available such as waiting resources, waiting validation (if repair conditions are used), approach SIMOPS time and repair time.
  - **Component states (using the variables)** . For each component, a table summarizes time spent in each states using the state variables (work, repair, preventive maintenance, etc...). This table is based upon changes of states.
  - **Component states (using time spent on places)** . For each component, a table summarizes time spent in each states based on sojourn time in the petri net places. This table is based upon changes of states.

If instantaneous change (for example if no spare or repair team are expected) there is no value.

- **Contributions to production shortfalls** : Adds a calculation to have the contributions to the losses of each component.

  The checkbox **by failure modes** is used to display the breakdown of contributions to production shortfalls by component failure mode. This option requires the calculation of statistics by failure mode of the components.
- **Contribution to the overtaking threshold** : Displays the components that contributed to over threshold: over limit of the flare or empty tank.
- **Counting of incidents due to components** : Adds a calculation to display for each component the number of times that it goes out from nominal state at the same time that there is an incident on the system.
- **Incidents** : Allows you to sort the incidents according to their durations:

  2 types of incidents (depend on needs):
  – Critical failure of system (production = 0)
  – Loss of production (production != 100%)
- **Production time histogram** : this option enables to split the production level:



- **Compute the histograms of the production results** : Enable to display the histograms of production and the mean and instant availabilities.

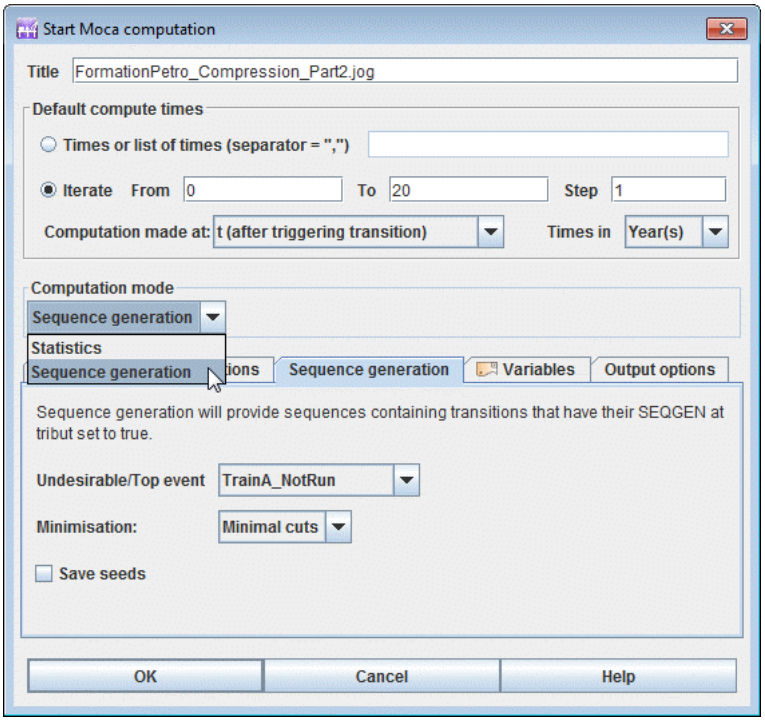
8. **Output options** : used to configure the output.

- **Print censured delays** : Prints the censored delays (or not).
- **Verbose** : Give internal information of Moca RP (launching, compilation, ...).

13.1.1.1. Sequence generation computations

The MocaRP computation engine enables to perform sequence generation. In this computation mode, it is possible to list all the sequences of transition fire which leads to undesirable event. However, statistical computation will not be made.

To activate this computation mode, in the option in the computation launching, select **Sequence Generation** in **Computation mode**. **Statistics** tab disappears in favour of **Sequence Generation** tab.



In the **Sequence Generation** tab, it is necessary to indicate the name of the **Undesirable event**. This event is a variable defined by a boolean expression. A `true` value indicates that the undesirable event has occurred during the current story.

Sequences that result in the undesirable event are grouped into an equivalence class. It is possible to choose the algorithm used to define this equivalence class:

- **None** : All transition sequences leading to the undesirable event will be returned. (no minimization)
- **Sub-words** : The minimality consists to search a common prefix, factor or suffix.
- **Minimal cut sets** : only the transitions are interested, without duplicates and without order, which led to the undesirable event. A minimal cut set includes all the sequences that include it in the same equivalence class.

See Manuel utilisateur Moca14.10.3 [3.4.5. Lancement d'une génération de séquences] for more information.

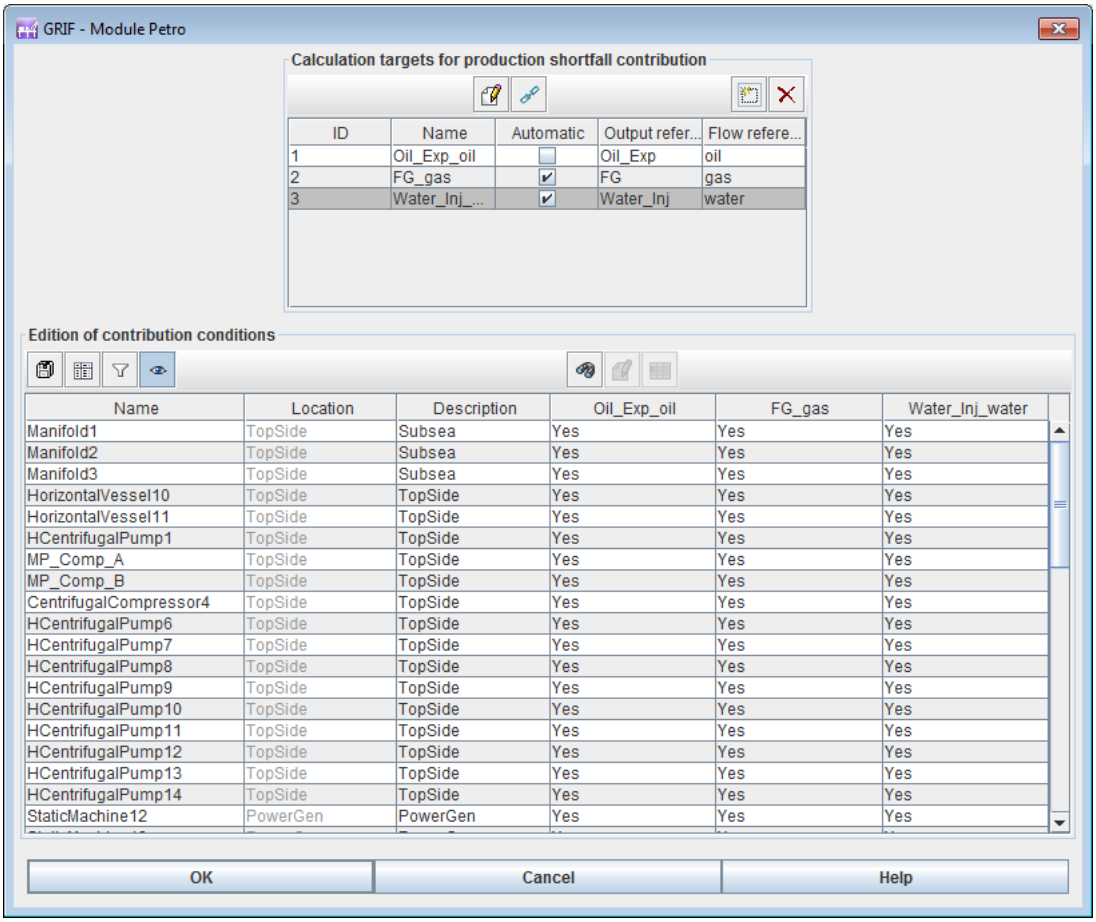
The computations options **Save seeds** enables to save the history seed and the occurrence time when undesirable event occurs.

Only the transitions with the flag `SEQGEN` with a true value are considered in the sequence. All others transitions are not taken into account. They generally correspond to transitions that don't contribute to the sequence generation (such as instantaneous reconfiguration transitions).

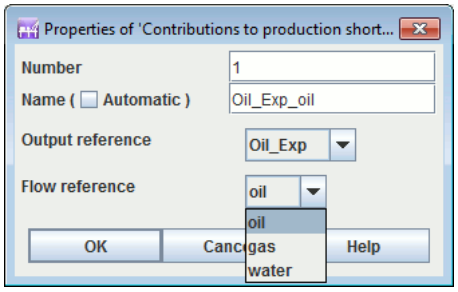
By default, this flag is generated in the prototype for *interresting* transitons. The choice to consider additional failures is left to the user's discretion. To take account additional failure in the sequences, check the box in the **SEQGEN** column (Section 4.2.5, “Additional failures”).

13.1.1.2. Contributions to production shortfalls

It is possible to study the contribution to production shortfalls of several outputs at once. To do that, go to menu **Data and Computations - Configuration of contributions to production shortfall**. The following window appears:



The upper part enables to select the different output and the associated flow reference. By clicking on it is possible to add a new output using the following window:



This window is also accessible by clicking on with previously selected data.

allow to merge 2 data.

The lower part enables to choose the impact of each element of the model:

- **Yes** : element contributes to the production shortfalls;
- **No** : element does not contribute to the production shortfalls;
- **User condition** : user fills out his own condition of production shutdown.

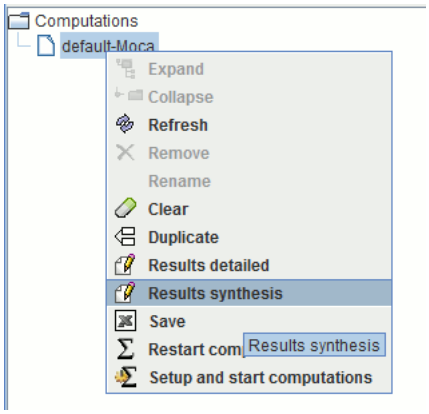
13.1.2. Reading the results (New GUI)

Following a Moca computation, the results can be displayed in one of two ways. A first window (Section 13.1.2.1, “Synthesis of results”) highlights the average productions and the contributions to loss. A second window (Section 13.1.2.2, “Moca Results”) presents all the results of the Moca computation.

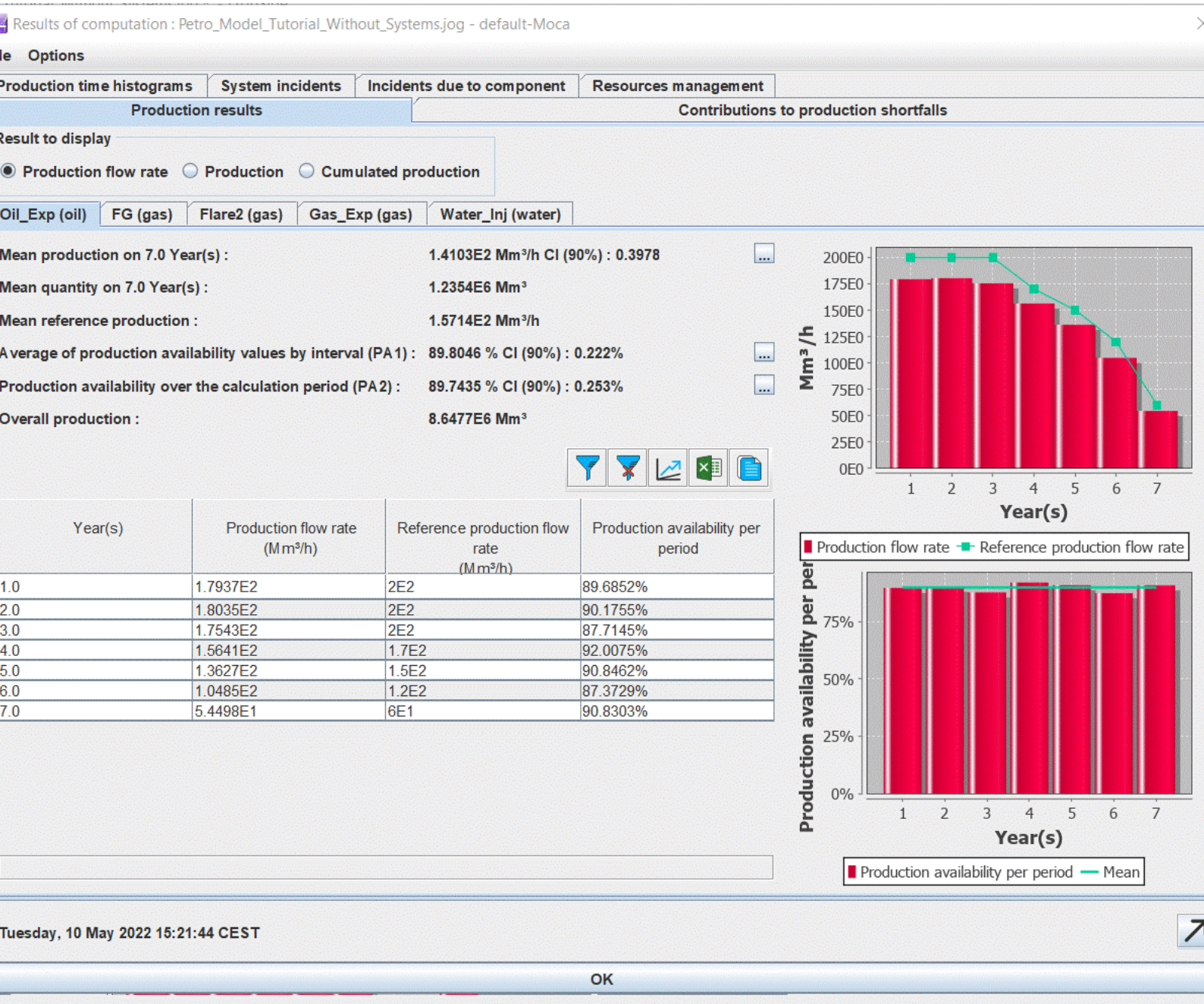
13.1.2.1. Synthesis of results

13.1.2.1.1. Presentation

This window is displayed after a Moca computation but can also be accessed from the results bank by right clicking on it:



The following results window is displayed:



As a rule, it includes a tab **Production results** which per output and per flow observed on the outputs give the main results. A **Contributions to loss** tab will be added if the contributions to loss calculations are requested.

In the example above, the mean productions are displayed on the left with the request or reference profile entered on the output, from which the availability over time is deduced.

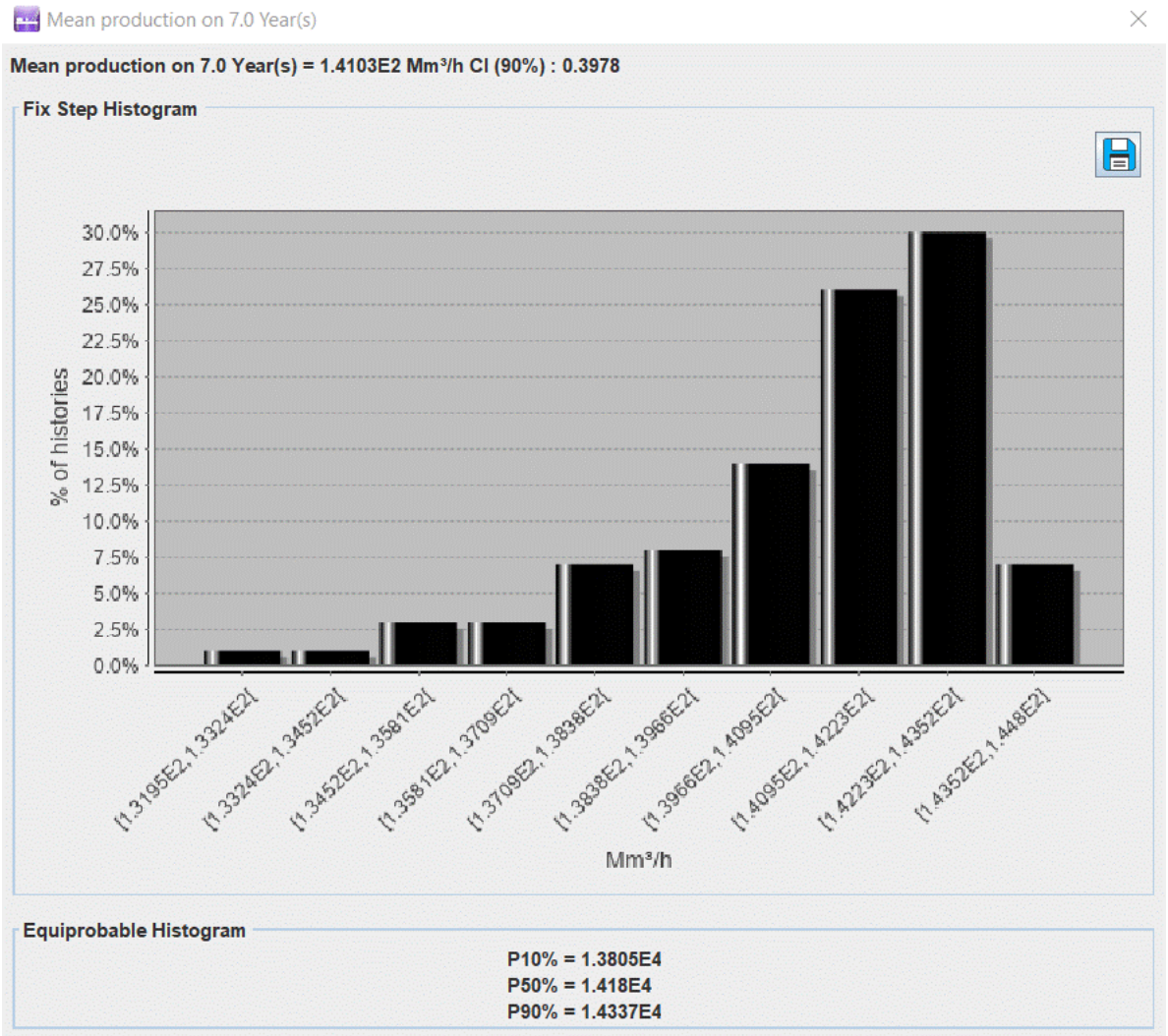
The first calculation of production availability is calculated using mean annual value and the second one using definition ISO 20815 standard.

To the right, a histogram shows the mean production and the reference profile over time. The histogram below shows the average availability over time.

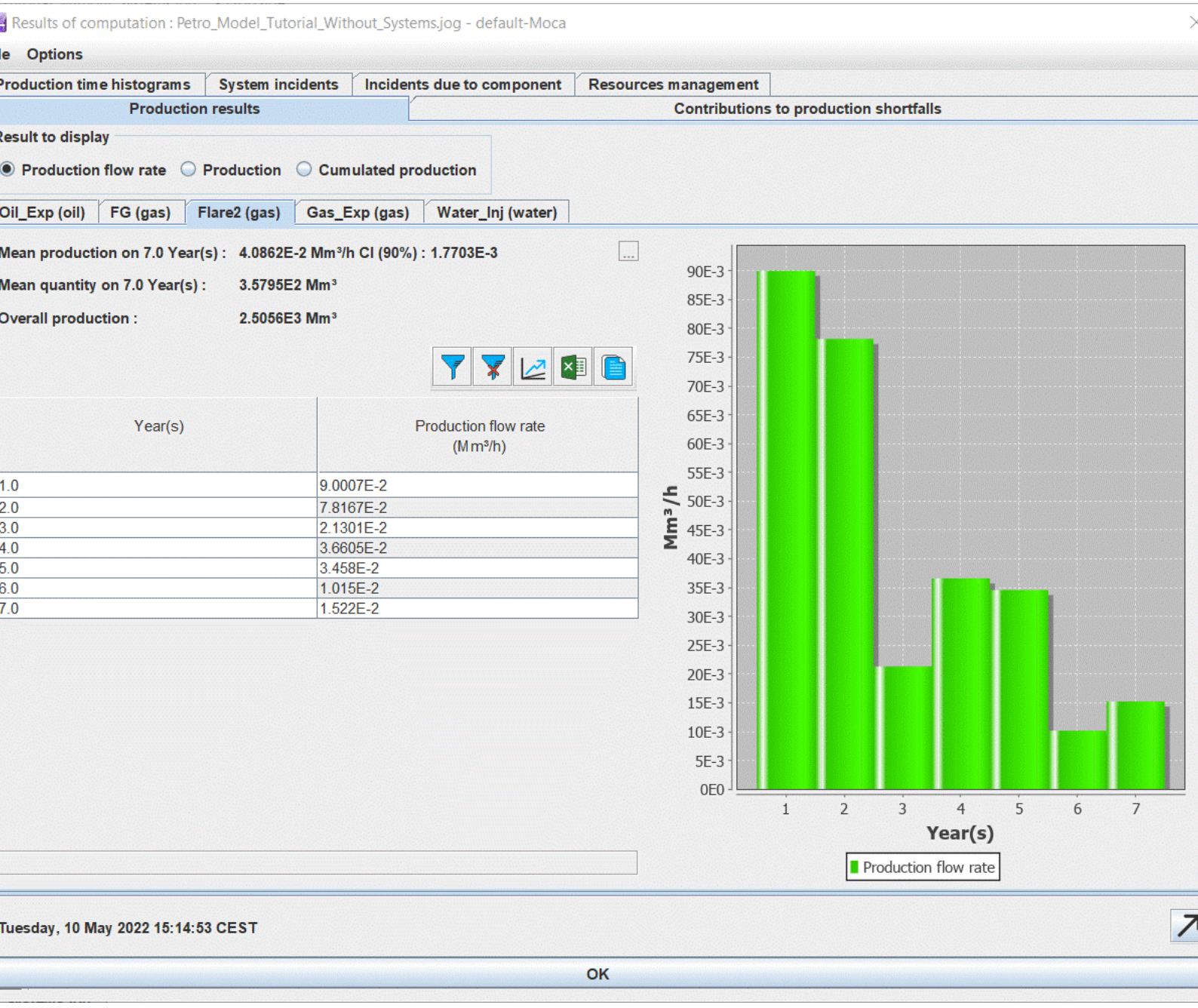
Two others buttons in the "Result to display" panel are available, the first one allowing to display the values of the production of our model with its histograms, the second one allowing to visualize the cumulated production.

If in configuration of Moca coputation, "Compute the histograms of the production results" is selected, it's possible to display with below button the desired histograms.

The following windows opens and enables to see the histograms of production disponibility on selected period and the dispersion due the system design.



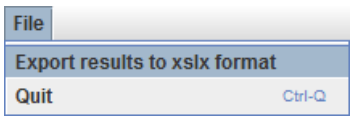
If results are requested on an output that does not have any request or reference profiles, only the mean production values will be displayed.



The tables can be exported in .csv format or to Excel using the toolbar for each table.

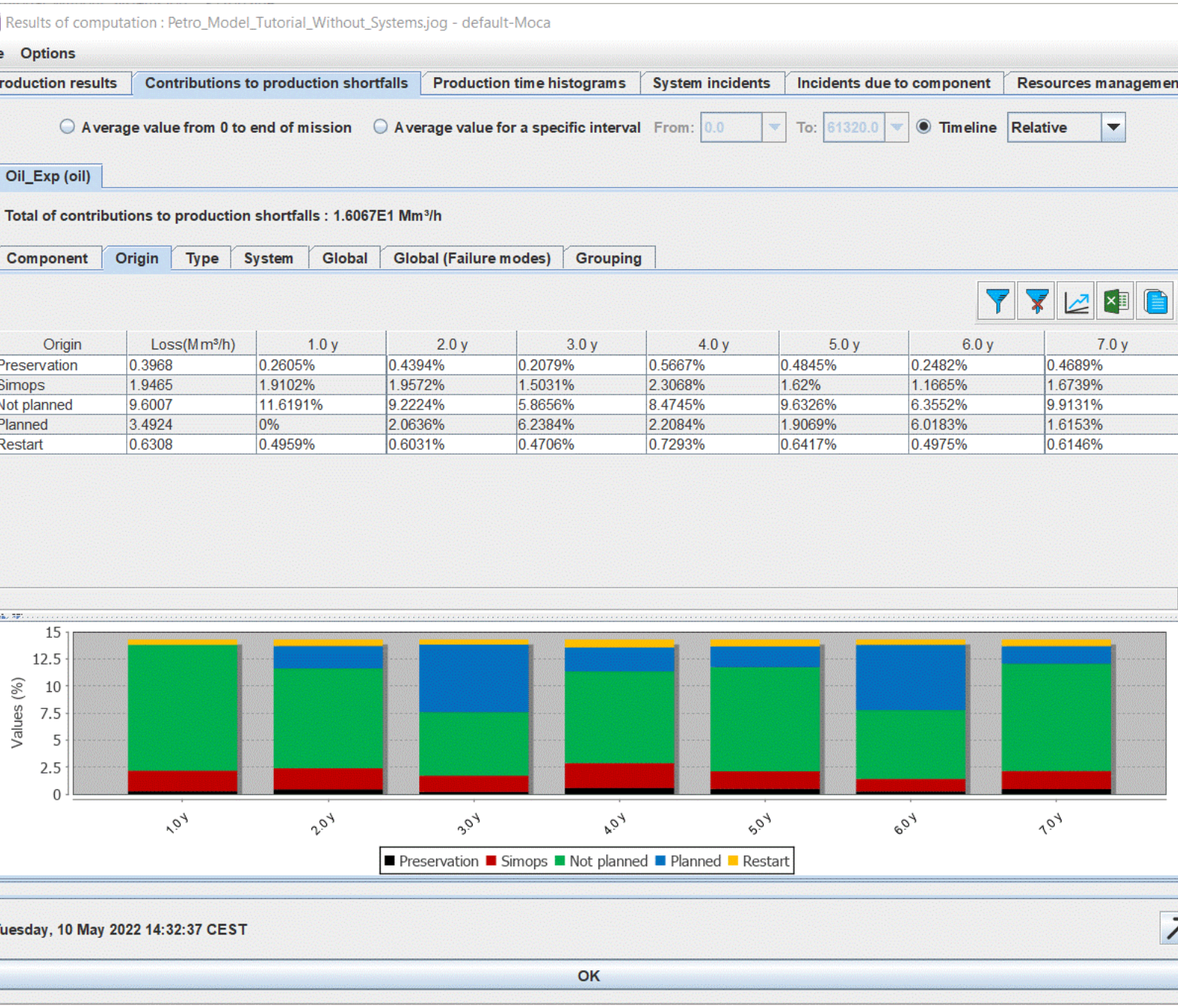
The graphs can be copied individually as images by right clicking on them.

The entire synthesis of results can be exported to Excel using the menu **File/Export results to xslx format**.



13.1.2.1.2. Contributions to loss

If contributions to loss calculations are activated, the following tab becomes available:

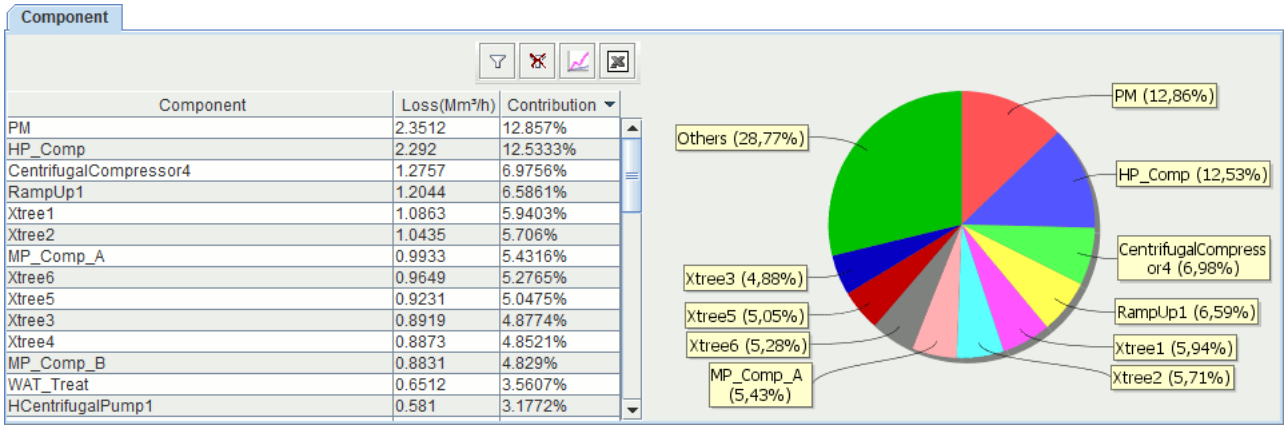


The results can be given over the mission time, over a specific period or according to the course of time.

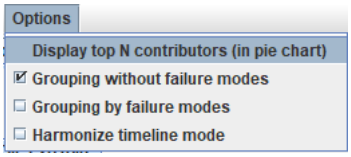
13.1.2.1.2.1. Average values

It contains a number of "sub-tabs":

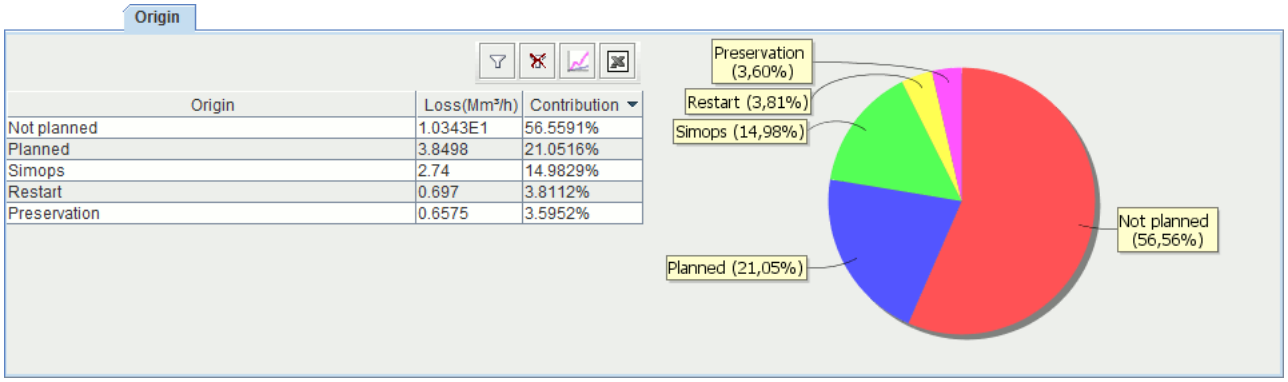
- The **Component** tab lists the set of components that contribute to losses. On the graph, it represents only the 10 biggest contributors. The rest is allocated to the category Others.



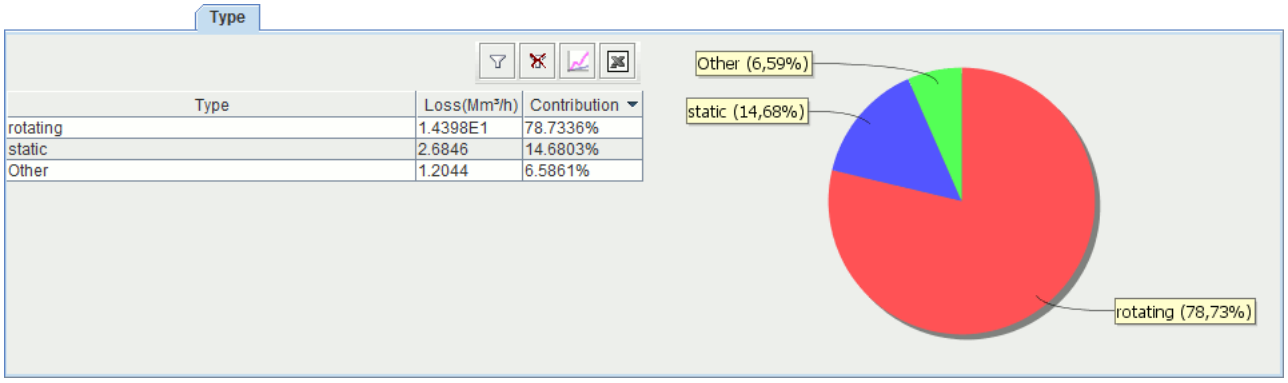
In the options, it is possible to indicate the number of contributors represented in the pie chart.



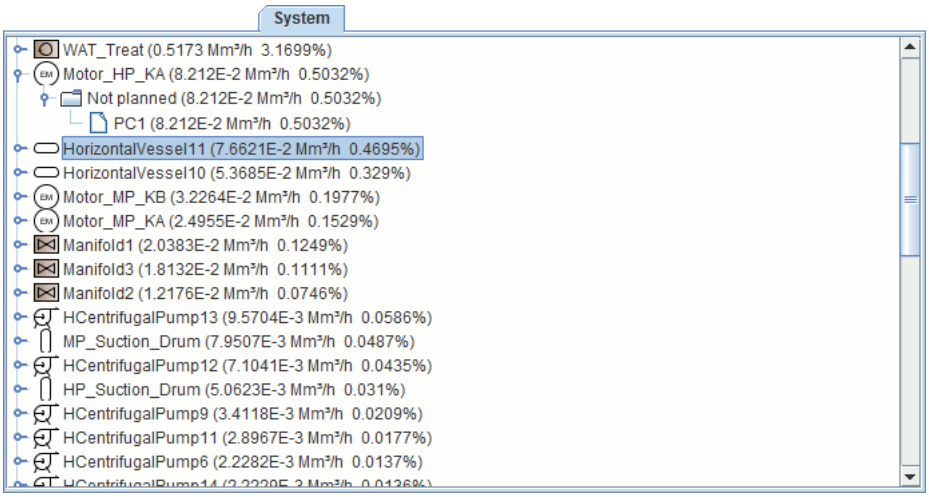
- A component can contribute to production loss in different ways. The **Origin** tab is used to distinguish the different sources of loss.



- The **Type** tab differentiates between rotating machines, static machines, etc.



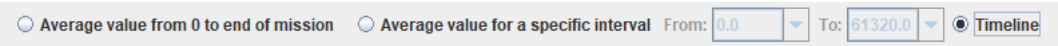
- The **System** tab enables to detail the contributions in the form of tree by failure modes.



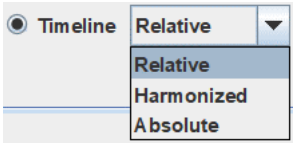
- The **Global** tab lists all the contributors to production loss. The **Loss** column displays the total losses attributable to this contributor. The **Contribution** column displays the percentage of the contribution relative to the total loss.
- If it is asked the tab **Global (Failure mode)** details the contribution in function of the failure modes.
- The last tab **Grouping** enables to create these own groups of contributors.

The left part is all the contributors to production shortfalls. The right part indicates the different groups in the first table **Groups of contributors** et the details are given in the table **Contributors of the group...**.

13.1.2.1.2.2. Values over time



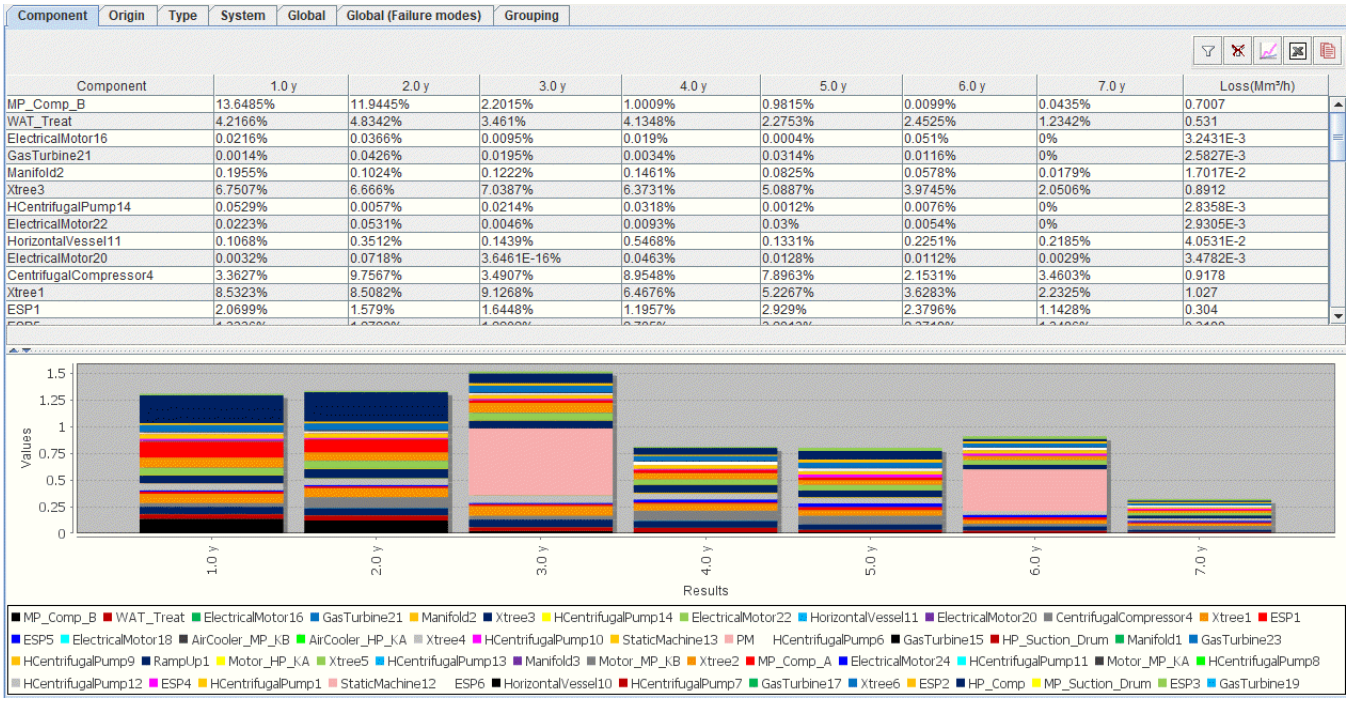
By selecting **Timeline**, you have the choice between 3 timeline options :



- Relative : Each cells corresponds to "loss due to component in year N" / "Total loss"
- Harmonised : Each cells corresponds to "loss due to component in year N" / "Total loss in year N"
- Absolute : Each cells corresponds to "loss due to component in year N" / "Reference production"

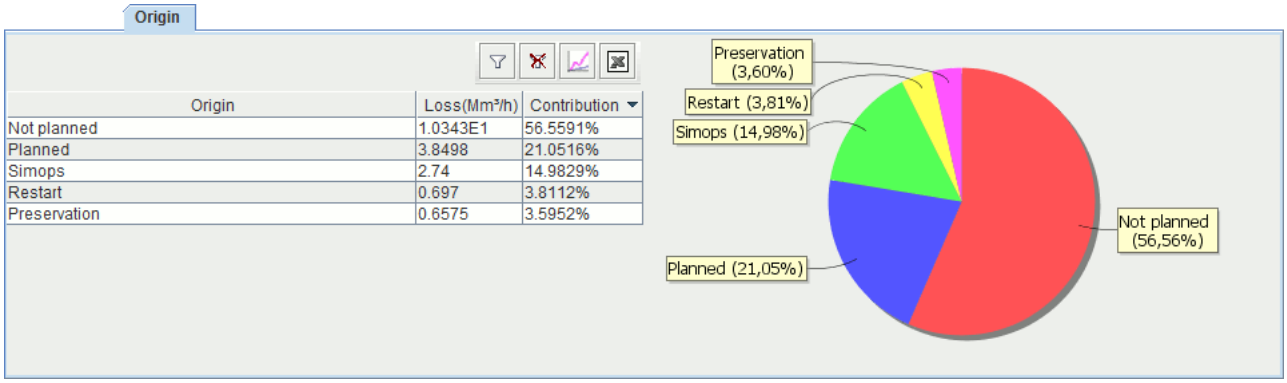
After selecting the desired timeline, the content of the tabs are modified :

- The **Component** tab lists the set of components that contribute to losses. By default, a bar chart displays the cumulative contributions to production shortfall for each year. For a component in year X, the contribution value is calculated and compared to the total contributions to derive a percentage.



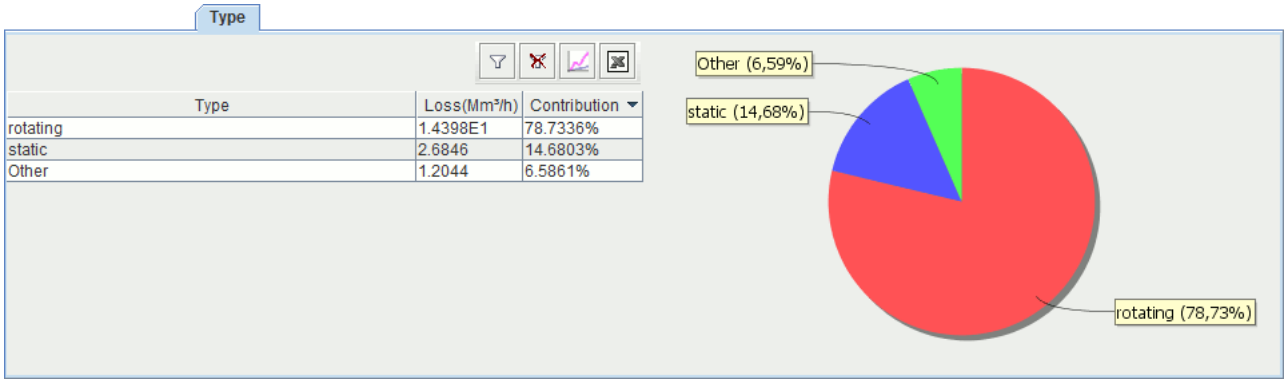
Component	1.0 y	2.0 y	3.0 y	4.0 y	5.0 y	6.0 y	7.0 y	Loss(Mm³/h)
MP_Comp_B	13.6485%	11.9445%	2.2015%	1.0009%	0.9815%	0.0099%	0.0435%	0.7007
WAT_Treat	4.2166%	4.8342%	3.461%	4.1348%	2.2753%	2.4525%	1.2342%	0.531
ElectricalMotor16	0.0216%	0.0366%	0.0095%	0.019%	0.0004%	0.051%	0%	3.2431E-3
GasTurbine21	0.0014%	0.0426%	0.0195%	0.0034%	0.0314%	0.0116%	0%	2.5827E-3
Manifold2	0.1955%	0.1024%	0.1222%	0.1461%	0.0825%	0.0578%	0.0179%	1.7017E-2
Xtree3	6.7507%	6.666%	7.0387%	6.3731%	5.0887%	3.9745%	2.0506%	0.8912
HCentrifugalPump14	0.0529%	0.0057%	0.0214%	0.0318%	0.0012%	0.0076%	0%	2.8358E-3
ElectricalMotor22	0.0223%	0.0531%	0.0046%	0.0093%	0.03%	0.0054%	0%	2.9305E-3
HorizontalVessel11	0.1068%	0.3512%	0.1439%	0.5468%	0.1331%	0.2251%	0.2185%	4.0531E-2
ElectricalMotor20	0.0032%	0.0718%	3.6461E-16%	0.0463%	0.0128%	0.0112%	0.0029%	3.4782E-3
CentrifugalCompressor4	3.3627%	9.7567%	3.4907%	8.9548%	7.8963%	2.1531%	3.4603%	0.9178
Xtree1	8.5323%	8.5082%	9.1268%	6.4676%	5.2267%	3.6283%	2.2325%	1.027
ESP1	2.0699%	1.579%	1.6448%	1.1957%	2.929%	2.3796%	1.1428%	0.304

- A component can contribute to production loss in different ways. The **Origin** tab is used to distinguish the different sources of loss.



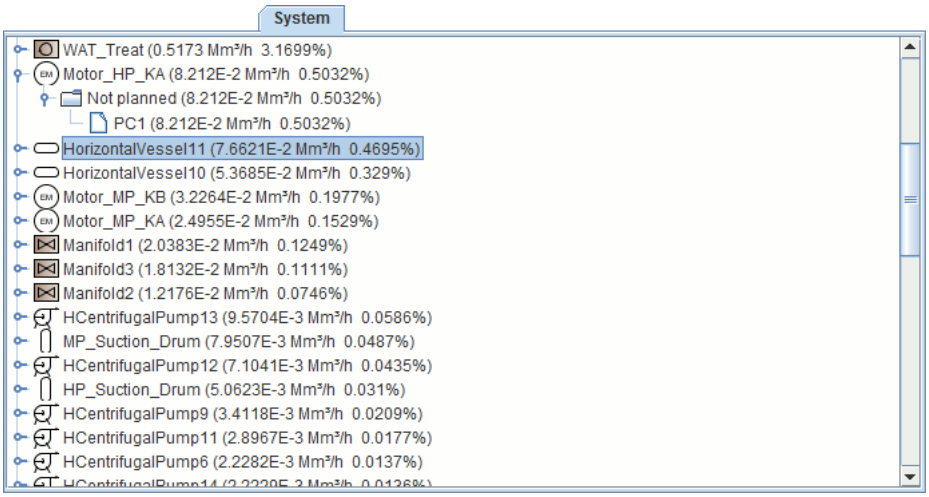
Origin	Loss(Mm³/h)	Contribution
Not planned	1.0343E1	56.5591%
Planned	3.8498	21.0516%
Simops	2.74	14.9829%
Restart	0.697	3.8112%
Preservation	0.6575	3.5952%

- The **Type** tab differentiates between rotating machines, static machines, etc.



Type	Loss(Mm³/h)	Contribution
rotating	1.4398E1	78.7336%
static	2.6846	14.6803%
Other	1.2044	6.5861%

- The **System** tab enables to detail the contributions in the form of tree by failure modes.

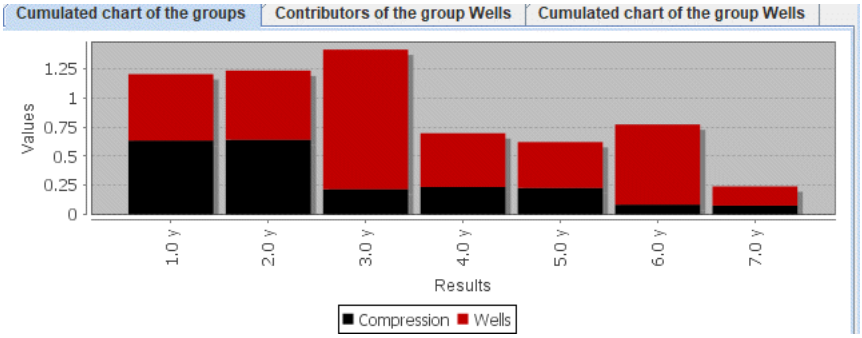


- The **Global** tab lists all the contributors to production loss. The **Loss** column displays the total losses attributable to this contributor. The **Contribution** column displays the percentage of the contribution relative to the total loss.
- If it is asked the tab **Global (Failure mode)** details the contribution in function of the failure modes.
- The **Global** tab lists all the contributors to production loss. The **Loss** column displays the total losses attributable to this contributor. The columns show the percentage of losses contribution of this component at year x (default).
- If it is asked the tab **Global (Failure mode)** details the contribution in function of the failure modes.
- The last tab **Grouping** enables to create these own groups of contributors.

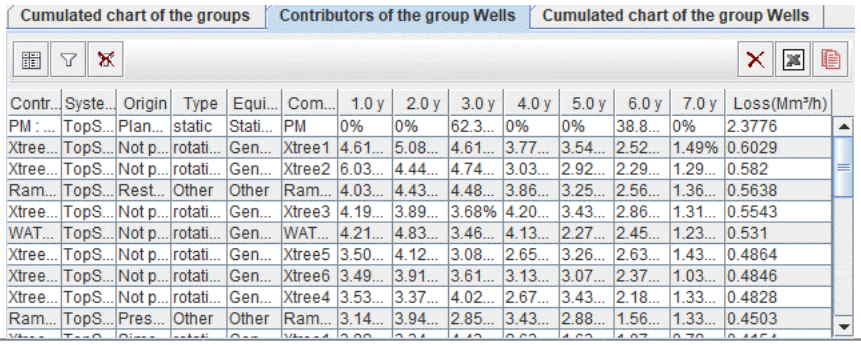
The left part is all the contributors to production shortfalls. The right part indicates the different groups in the first table **Groups of contributors** et the details are given in the table **Contributors of the group...**.

Below this table are three tabs:

– **Cumulated graph of the groups** summarizes the contributions to groups losses for each year

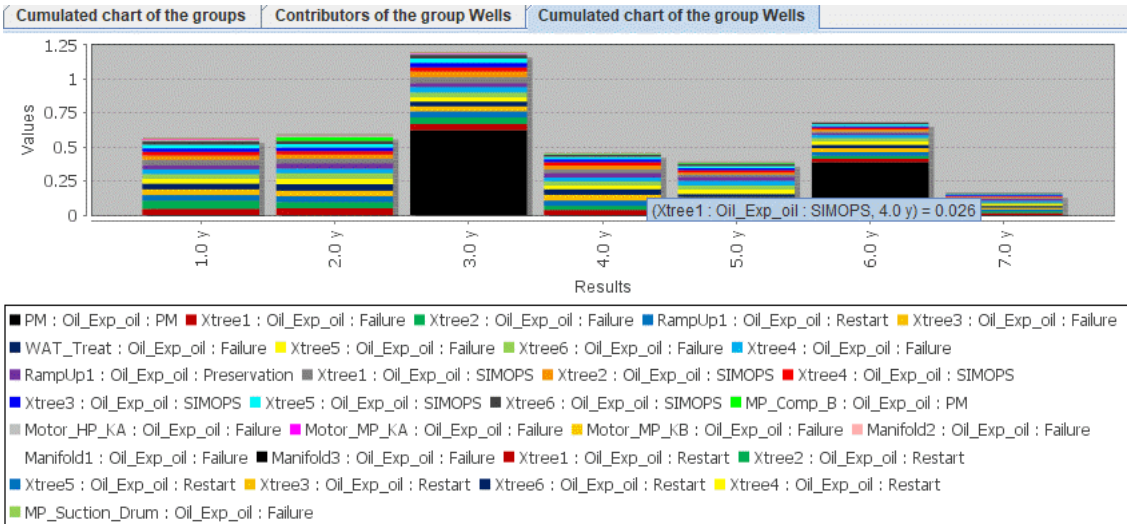


– **Contributors of the group ...** details the components that make up the selected group



Contr...	Syste...	Origin	Type	Equi...	Com...	1.0 y	2.0 y	3.0 y	4.0 y	5.0 y	6.0 y	7.0 y	Loss(Mm³/h)
PM : ...	TopS...	Plan...	static	Stati...	PM	0%	0%	62.3...	0%	0%	38.8...	0%	2.3776
Xtree...	TopS...	Not p...	rotati...	Gen...	Xtree1	4.61...	5.08...	4.61...	3.77...	3.54...	2.52...	1.49%	0.6029
Xtree...	TopS...	Not p...	rotati...	Gen...	Xtree2	6.03...	4.44...	4.74...	3.03...	2.92...	2.29...	1.29...	0.582
Ram...	TopS...	Rest...	Other	Other	Ram...	4.03...	4.43...	4.48...	3.86...	3.25...	2.56...	1.36...	0.5638
Xtree...	TopS...	Not p...	rotati...	Gen...	Xtree3	4.19...	3.89...	3.68%	4.20...	3.43...	2.86...	1.31...	0.5543
WAT...	TopS...	Not p...	rotati...	Gen...	WAT...	4.21...	4.83...	3.46...	4.13...	2.27...	2.45...	1.23...	0.531
Xtree...	TopS...	Not p...	rotati...	Gen...	Xtree5	3.50...	4.12...	3.08...	2.65...	3.26...	2.63...	1.43...	0.4864
Xtree...	TopS...	Not p...	rotati...	Gen...	Xtree6	3.49...	3.91...	3.61...	3.13...	3.07...	2.37...	1.03...	0.4846
Xtree...	TopS...	Not p...	rotati...	Gen...	Xtree4	3.53...	3.37...	4.02...	2.67...	3.43...	2.18...	1.33...	0.4828
Ram...	TopS...	Pres...	Other	Other	Ram...	3.14...	3.94...	2.85...	3.43...	2.88...	1.56...	1.33...	0.4503

– **Cumulated graph of the group ...** graphically details the contributions to losses of the selected group



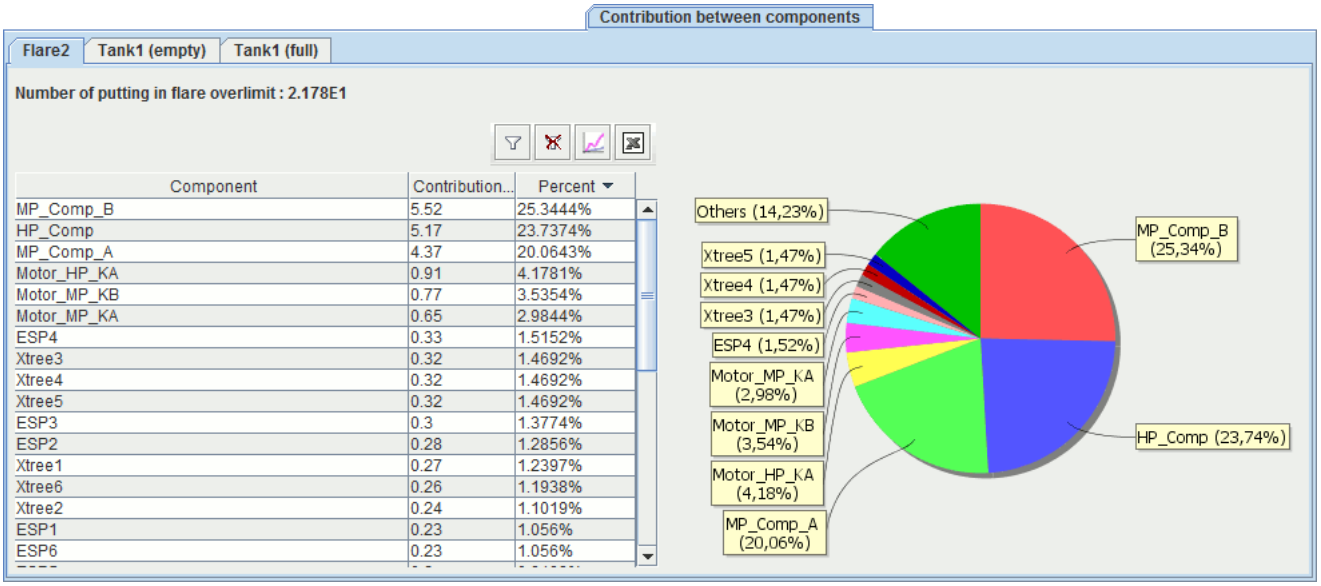
 : enables to add the contributors selected in the left part in a new group.

 : enables to add the element selected in existing group. It is beforehand necessary to have selected ungroup in the right part.

 : enables to delete elements selected in the right part of the group of contributors.

13.1.2.1.3. Contributions between components

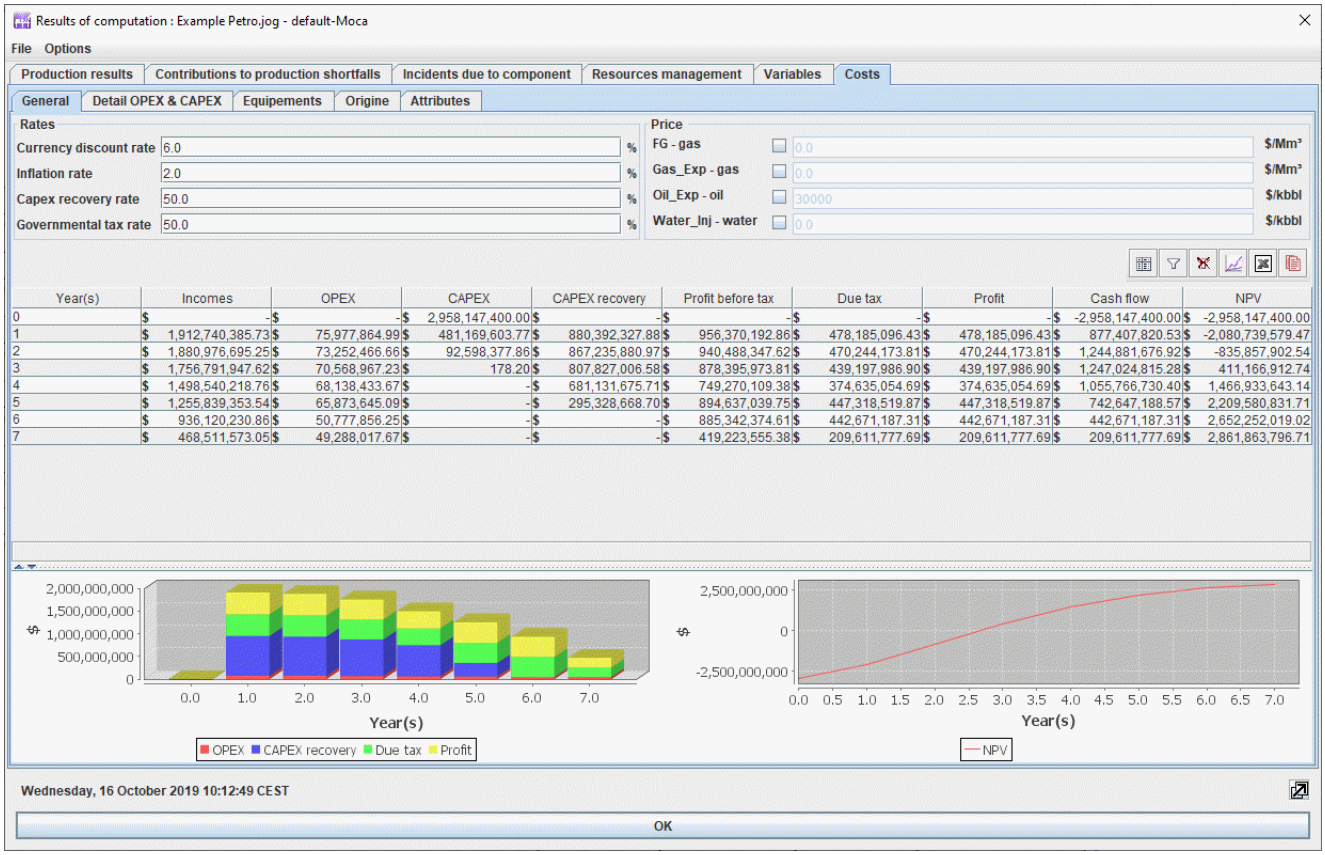
This result tab enables to know, in case of very limited flaring or tank, the last component that failed just before the flare “overlimit”. In this case the flare or tank doesn't appear as a principal contributor.



For the tank, the results indicate which emptied or fills lastly the tank.

13.1.2.1.4. Costs

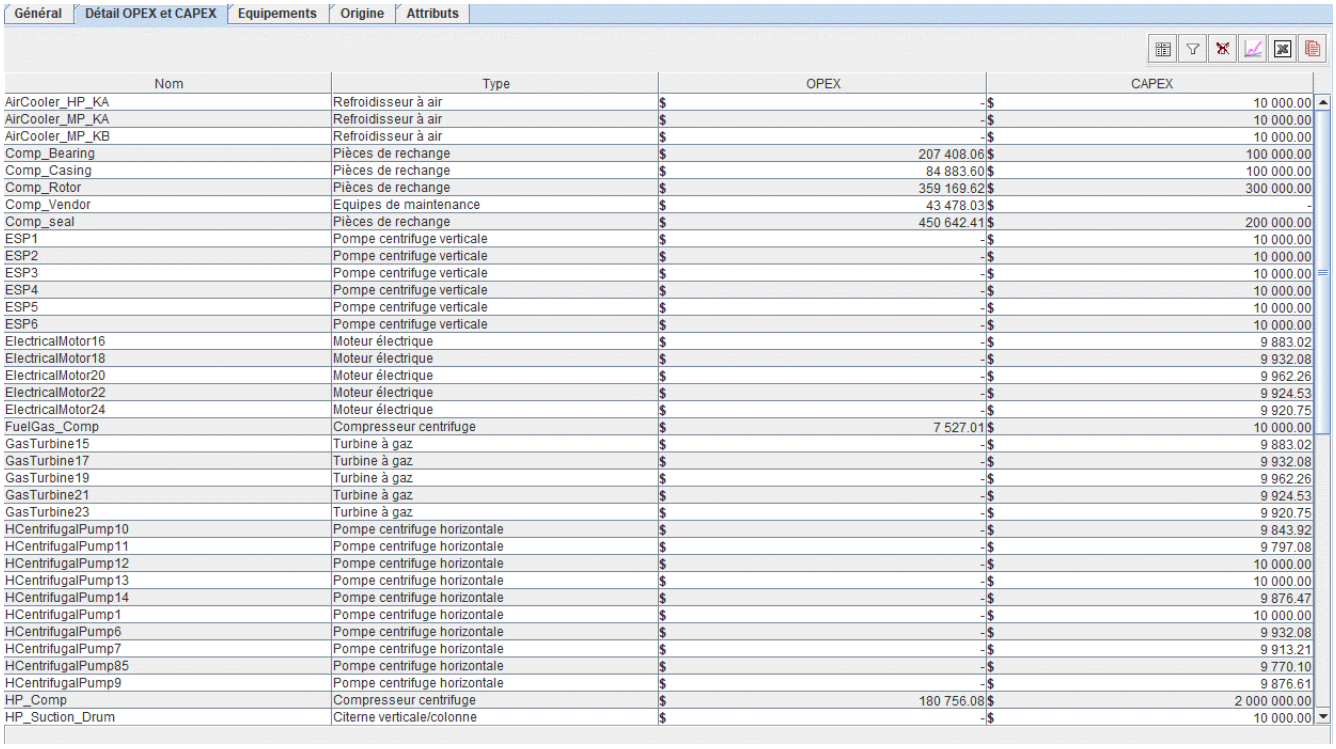
This tab lets you know the overall **Costs** and the **Costs** associated with each component.



When the detailed cost calculations option is enabled, five tabs are displayed:

- **General** describes the revenues, expenses, refunds and taxes for each year of operation. The different **Rates** and sales **Prices** can be modified directly from this tab. The costs are updated automatically.

- **Detail OPEX and CAPEX** describes operating (**OPEX**) or investment-type expenses (**CAPEX**) for each equipment.

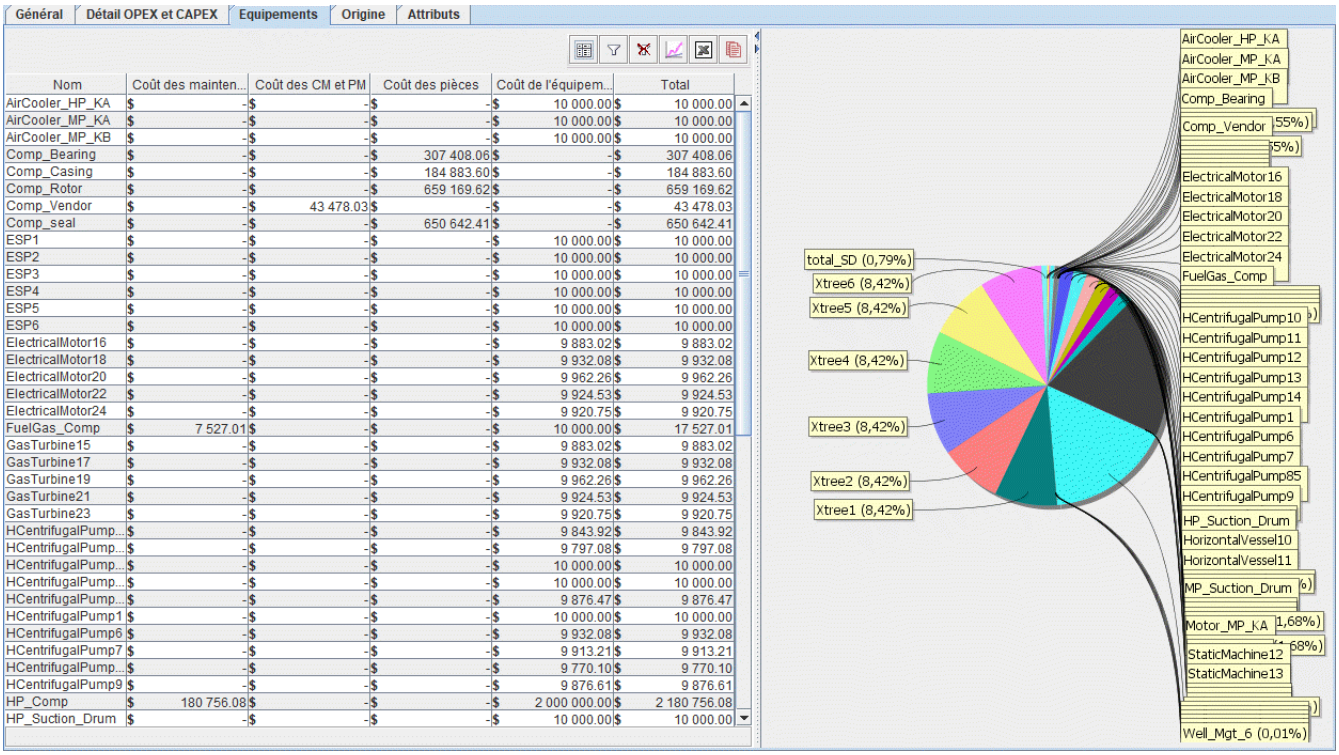


Général | Détail OPEX et CAPEX | Equipements | Origine | Attributs

Nom	Type	OPEX	CAPEX
AirCooler_HP_KA	Refroidisseur à air	$ -	$ 10 000.00
AirCooler_MP_KA	Refroidisseur à air	$ -	$ 10 000.00
AirCooler_MP_KB	Refroidisseur à air	$ -	$ 10 000.00
Comp_Bearing	Pièces de rechange	$ 207 408.06	$ 100 000.00
Comp_Casing	Pièces de rechange	$ 84 883.60	$ 100 000.00
Comp_Rotor	Pièces de rechange	$ 359 169.62	$ 300 000.00
Comp_Vendor	Equipes de maintenance	$ 43 478.03	$ -
Comp_seal	Pièces de rechange	$ 450 642.41	$ 200 000.00
ESP1	Pompe centrifuge verticale	$ -	$ 10 000.00
ESP2	Pompe centrifuge verticale	$ -	$ 10 000.00
ESP3	Pompe centrifuge verticale	$ -	$ 10 000.00
ESP4	Pompe centrifuge verticale	$ -	$ 10 000.00
ESP5	Pompe centrifuge verticale	$ -	$ 10 000.00
ESP6	Pompe centrifuge verticale	$ -	$ 10 000.00
ElectricalMotor16	Moteur électrique	$ -	$ 9 883.02
ElectricalMotor18	Moteur électrique	$ -	$ 9 932.08
ElectricalMotor20	Moteur électrique	$ -	$ 9 962.26
ElectricalMotor22	Moteur électrique	$ -	$ 9 924.53
ElectricalMotor24	Moteur électrique	$ -	$ 9 920.75
FuelGas_Comp	Compresseur centrifuge	$ 7 527.01	$ 10 000.00
GasTurbine15	Turbine à gaz	$ -	$ 9 883.02
GasTurbine17	Turbine à gaz	$ -	$ 9 932.08
GasTurbine19	Turbine à gaz	$ -	$ 9 962.26
GasTurbine21	Turbine à gaz	$ -	$ 9 924.53
GasTurbine23	Turbine à gaz	$ -	$ 9 920.75
HCentrifugalPump10	Pompe centrifuge horizontale	$ -	$ 9 843.92
HCentrifugalPump11	Pompe centrifuge horizontale	$ -	$ 9 797.08
HCentrifugalPump12	Pompe centrifuge horizontale	$ -	$ 10 000.00
HCentrifugalPump13	Pompe centrifuge horizontale	$ -	$ 10 000.00
HCentrifugalPump14	Pompe centrifuge horizontale	$ -	$ 9 876.47
HCentrifugalPump1	Pompe centrifuge horizontale	$ -	$ 10 000.00
HCentrifugalPump6	Pompe centrifuge horizontale	$ -	$ 9 932.08
HCentrifugalPump7	Pompe centrifuge horizontale	$ -	$ 9 913.21
HCentrifugalPump85	Pompe centrifuge horizontale	$ -	$ 9 770.10
HCentrifugalPump9	Pompe centrifuge horizontale	$ -	$ 9 876.61
HP_Comp	Compresseur centrifuge	$ 180 756.08	$ 2 000 000.00
HP_Suction_Drum	Citerne verticale/colonne	$ -	$ 10 000.00

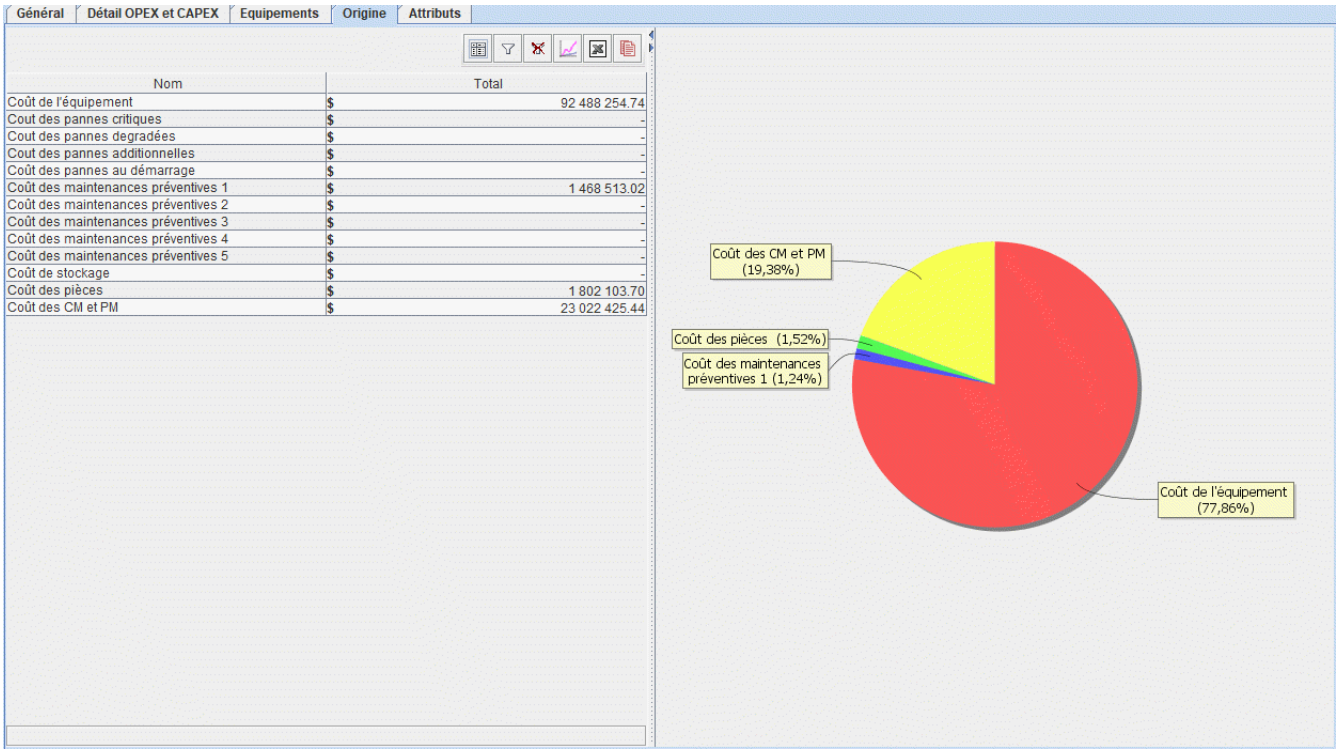
- **Equipments** details the different costs for each equipment:
  - **Equipment cost**
  - **Critical failure cost**
  - **Degraded failure cost**
  - **Additional failure cost**
  - **Fail to start cost**
  - **Storage cost**
  - **Spare parts cost**
  - **Cost of Preventive maintenance 1**
  - **Cost of Preventive maintenance 2**
  - **Cost of Preventive maintenance 3**
  - **Cost of Preventive maintenance 4**
  - **Cost of Preventive maintenance 5**

– **CM and PM cost**: curative and preventive maintenance



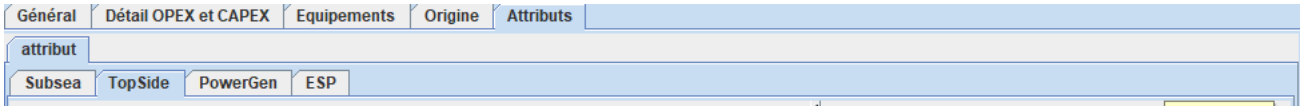
Nom	Coût des mainten...	Coût des CM et PM	Coût des pièces	Coût de l'équipem...	Total
AirCooler_HP_KA	$ -	$ -	$ -	$ 10 000.00	$ 10 000.00
AirCooler_MP_KA	$ -	$ -	$ -	$ 10 000.00	$ 10 000.00
AirCooler_MP_KB	$ -	$ -	$ -	$ 10 000.00	$ 10 000.00
Comp_Bearing	$ -	$ -	$ 307 408.06	$ -	$ 307 408.06
Comp_Casing	$ -	$ -	$ 184 883.60	$ -	$ 184 883.60
Comp_Rotor	$ -	$ -	$ 659 169.62	$ -	$ 659 169.62
Comp_Vendor	$ -	$ 43 478.03	$ -	$ -	$ 43 478.03
Comp_seal	$ -	$ -	$ 650 642.41	$ -	$ 650 642.41
ESP1	$ -	$ -	$ -	$ 10 000.00	$ 10 000.00
ESP2	$ -	$ -	$ -	$ 10 000.00	$ 10 000.00
ESP3	$ -	$ -	$ -	$ 10 000.00	$ 10 000.00
ESP4	$ -	$ -	$ -	$ 10 000.00	$ 10 000.00
ESP5	$ -	$ -	$ -	$ 10 000.00	$ 10 000.00
ESP6	$ -	$ -	$ -	$ 10 000.00	$ 10 000.00
ElectricalMotor16	$ -	$ -	$ -	$ 9 883.02	$ 9 883.02
ElectricalMotor18	$ -	$ -	$ -	$ 9 932.08	$ 9 932.08
ElectricalMotor20	$ -	$ -	$ -	$ 9 962.26	$ 9 962.26
ElectricalMotor22	$ -	$ -	$ -	$ 9 924.53	$ 9 924.53
ElectricalMotor24	$ -	$ -	$ -	$ 9 920.75	$ 9 920.75
FuelGas_Comp	$ 7 527.01	$ -	$ -	$ 10 000.00	$ 17 527.01
GasTurbine15	$ -	$ -	$ -	$ 9 883.02	$ 9 883.02
GasTurbine17	$ -	$ -	$ -	$ 9 932.08	$ 9 932.08
GasTurbine19	$ -	$ -	$ -	$ 9 962.26	$ 9 962.26
GasTurbine21	$ -	$ -	$ -	$ 9 924.53	$ 9 924.53
GasTurbine23	$ -	$ -	$ -	$ 9 920.75	$ 9 920.75
HCentrifugalPump...	$ -	$ -	$ -	$ 9 843.92	$ 9 843.92
HCentrifugalPump...	$ -	$ -	$ -	$ 9 797.08	$ 9 797.08
HCentrifugalPump...	$ -	$ -	$ -	$ 10 000.00	$ 10 000.00
HCentrifugalPump...	$ -	$ -	$ -	$ 10 000.00	$ 10 000.00
HCentrifugalPump...	$ -	$ -	$ -	$ 9 876.47	$ 9 876.47
HCentrifugalPump1	$ -	$ -	$ -	$ 10 000.00	$ 10 000.00
HCentrifugalPump6	$ -	$ -	$ -	$ 9 932.08	$ 9 932.08
HCentrifugalPump7	$ -	$ -	$ -	$ 9 913.21	$ 9 913.21
HCentrifugalPump...	$ -	$ -	$ -	$ 9 770.10	$ 9 770.10
HCentrifugalPump9	$ -	$ -	$ -	$ 9 876.61	$ 9 876.61
HP_Comp	$ 180 756.08	$ -	$ -	$ 2 000 000.00	$ 2 180 756.08
HP_Suction_Drum	$ -	$ -	$ -	$ 10 000.00	$ 10 000.00

- **Origin** details for each type of the aggregate costs related to it.

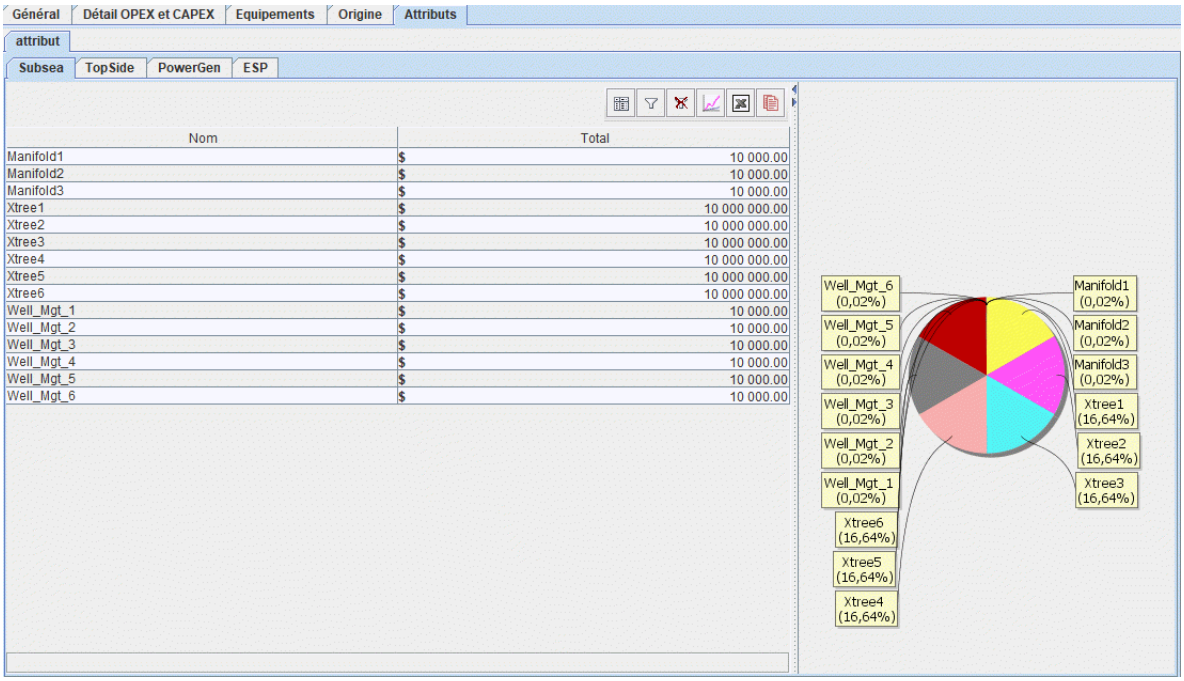


Nom	Total
Coût de l'équipement	$ 92 488 254.74
Cout des pannes critiques	$ -
Cout des pannes degradées	$ -
Cout des pannes additionnelles	$ -
Coût des pannes au démarrage	$ -
Coût des maintenances préventives 1	$ 1 468 513.02
Coût des maintenances préventives 2	$ -
Coût des maintenances préventives 3	$ -
Coût des maintenances préventives 4	$ -
Coût des maintenances préventives 5	$ -
Coût de stockage	$ -
Coût des pièces	$ 1 802 103.70
Coût des CM et PM	$ 23 022 425.44

- **Attributes** is composed of sub tabs listing the attributes and each attribute are composed of sub-tabs if each value:

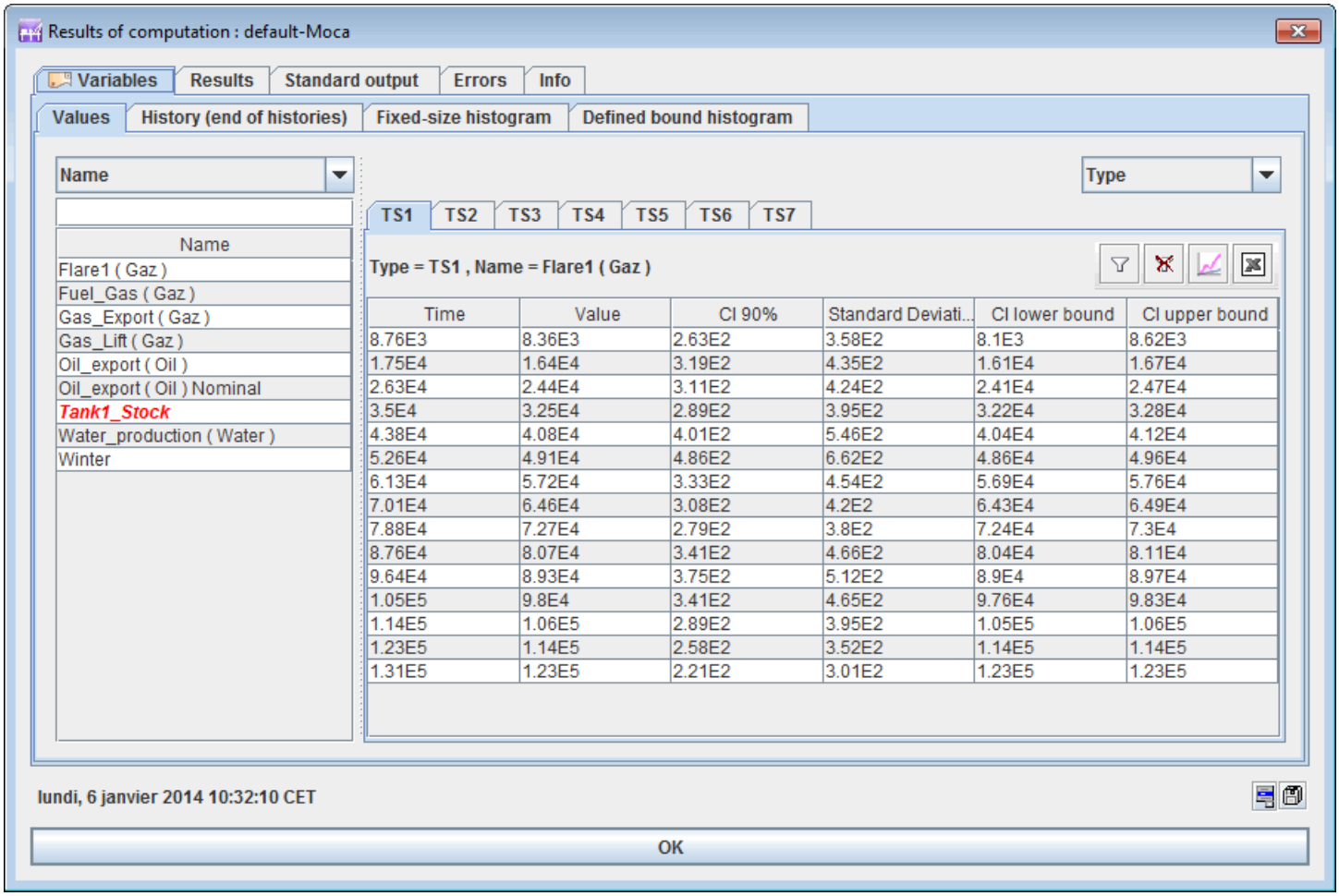


A table displays the sum of costs for each equipment with the same attribute:



13.1.2.2. Moca Results

This window can be accessed from the results bank, by right clicking on the **default-Moca** computation.



The Moca data are presented in a window containing tabs: variables, results, standard output, info.

13.1.2.2.1. Tab of Variables

The **Variables** tab contains all the information computed for each variable (or statistical state).

- **Value**: Contains every value of a variable for every type of statistic. (cf. Section 8.3.2.1, “Types of statistics”)
- **History (at the end of histories)**: contains historical values for each computed statistic.
- **Fixed size Histogram**: Contains histograms computed by Moca (cf.)
- **Equiprobable classes Histogram**: Contains histograms computed by Moca (cf.)
- **User defined Histogram**: Contains histograms computed by Moca (cf.)
- **Timeline**: Contains a timeline for each variable. Times are automatically computed by Moca.

13.1.2.2.2. Other tabs

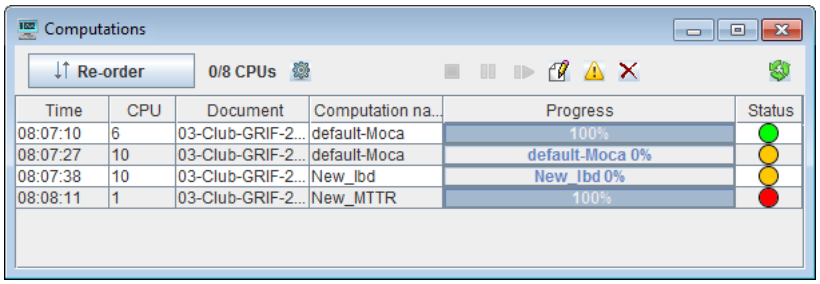
Other tabs display "raw" results. XML tab contains XML output of Moca, it is the file used to retrieve data. This file can be used for further post-treatments.

Standard output displays the standard output of Moca (available only after computing).

Info tab contains useful information about computation (simulation time, number of histories that have been done ...)

13.2. Compute manager

Compute manager shows the calculations. That are currently running or already performed.

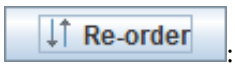


Compute manager is automatically displayed when calculations are performed. User can display the window using the following icon  .

This tab is made of 6 columns:

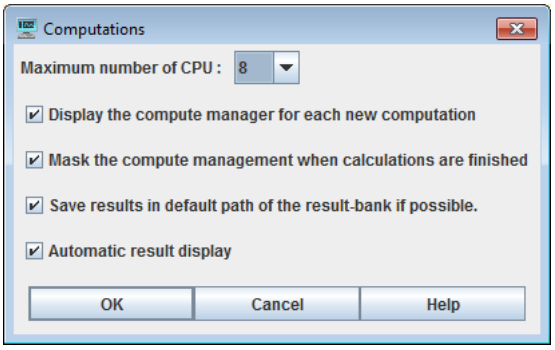
- **Time**: The hour of calculation launch;
- **CPU**: number of CPU used;
- **Document**: document name;
- **Computation name**: name of results file;
- **Progress**: progress bar;
- **Status**: finished in green, in progress in yellow, error in red;

In **Compute Manager** some actions are available:

- 

: allow to reorganize the calculations order;

- : display the following windows for computation settings:

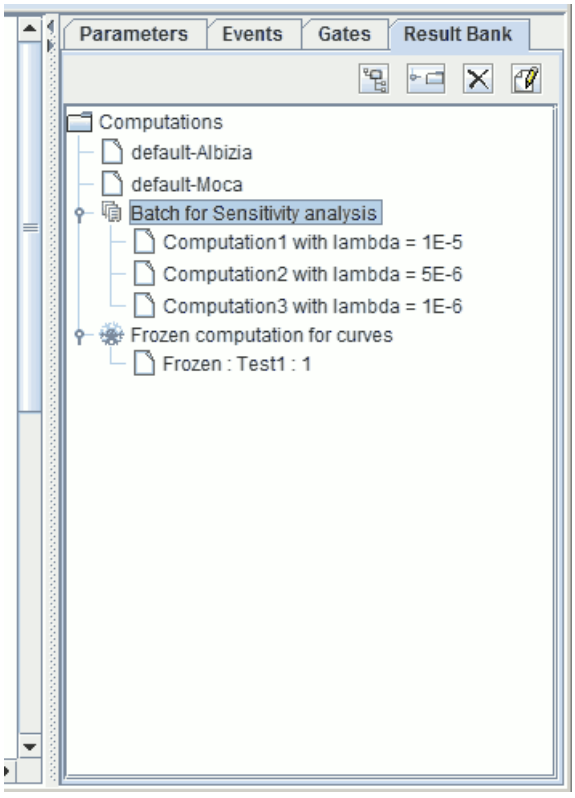


- : stop selected compute;
- : suspend selected compute;
- : resume compute in suspend;
- : display results of selected compute;
- : details errors;
- : remove selected compute;
- : clear all compute;

When a task is added to Compute manager, user is not blocked until the task is ended. He can continue to work on his model. He can even relaunch a calculation. The various tasks accumulate and are treated sequentially.

13.3. Result Bank

Every GRIF computation is stored in result bank which is available on the right of the module.

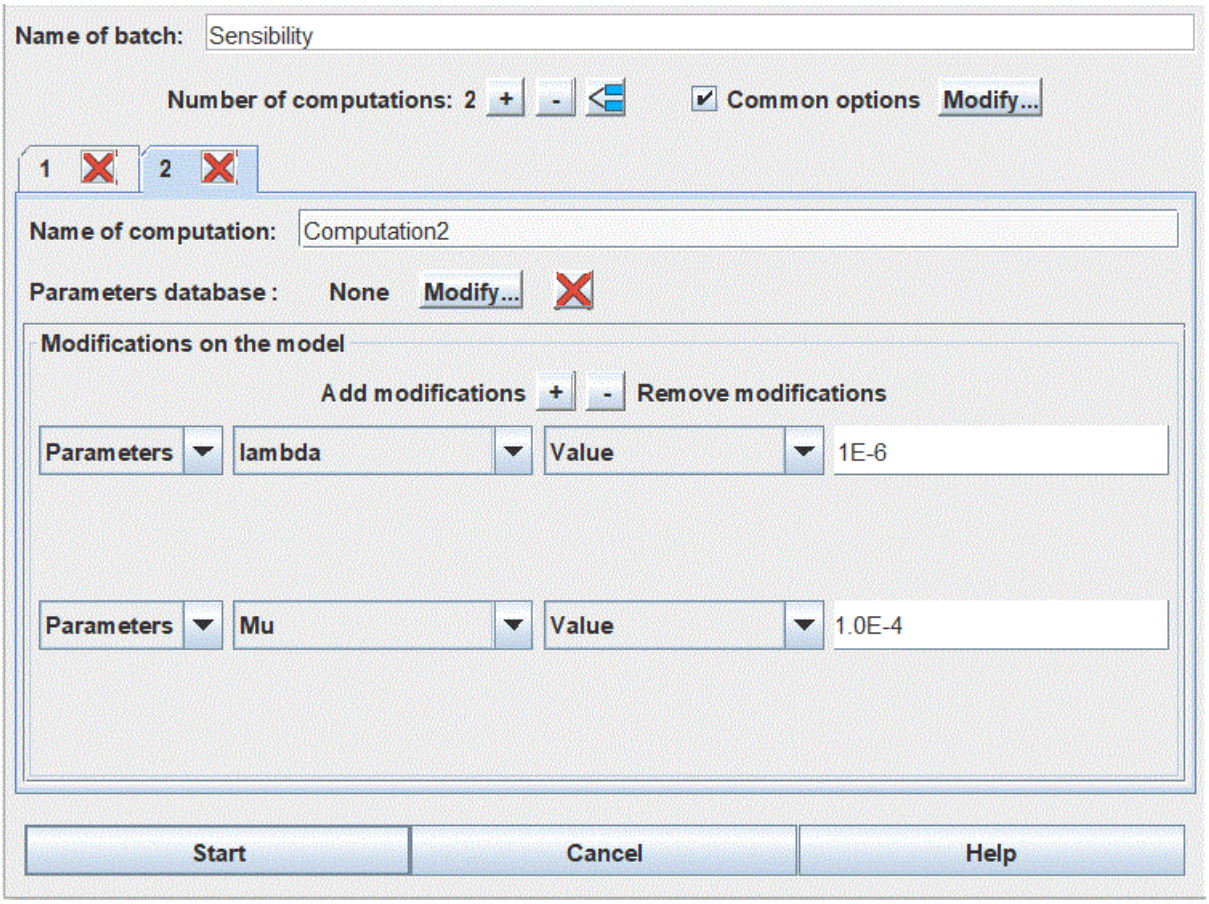


You can display a result with a double-click on it. There is a default result for each computation engine, it is the place where "standard" computations are stored. Then, each is a batch computation directory, it contains as

many results as computations asked for the batch. Finally, the directory contains results for curves that have been frozen.

13.4. Batch computation

In order to do fast sensibility analysis or to compare some results with different parameters, it can be very useful to do calculation one after another automatically. To do this, use the **Data and Computation / Batch computation ...** menu.



The batch launching window is made of two part, the first is for the name of the batch and the number of computations in the batch. Then each computation can be set up:

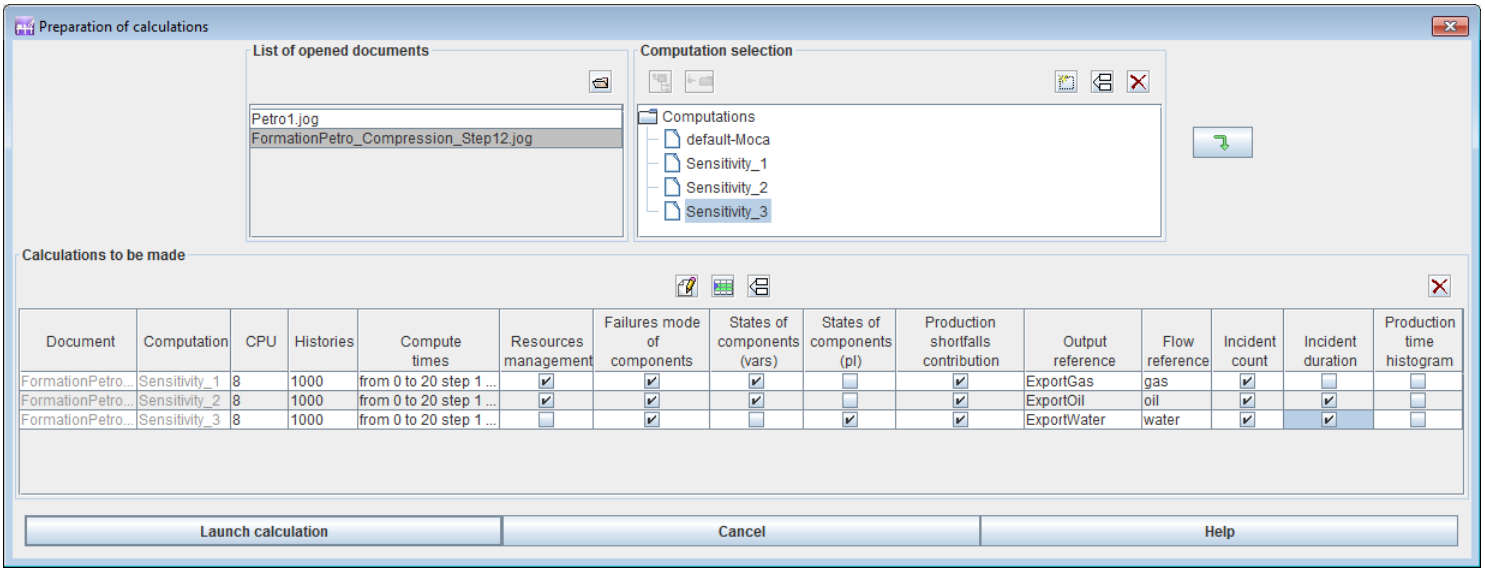
- **Name of computation:** for identification in results
- **Computation options:** contains every option related to this computation (times, types ...)
- **Modifications on the model:** specifies modifications that will be made on the model before computation launching. You can add as many modifications as you want with the + button. Each modification is made of 4 parts:
  1. 1 drop-down menu for object type
  2. 1 drop-down menu for the object that will be modified
  3. 1 drop-down menu to specify what will be modified on this object (value for a parameter, law for other object ...)
  4. Then you must enter the new value in the cell

The above example shows a batch with 2 computations, the first is made with a lambda parameter with 1.0E-5 value and a mu parameter with 0.1 value.

After a computation, the model is always reset up to an initial state without modification.

13.5. Preparation of calculations

by clicking on the icon  user has access to **Preparation of calculations** .



Document	Computation	CPU	Histories	Compute times	Resources management	Failures mode of components	States of components (vars)	States of components (pl)	Production shortfalls contribution	Output reference	Flow reference	Incident count	Incident duration	Production time histogram
FormationPetro...	Sensitivity_1	8	1000	from 0 to 20 step 1 ...	☑	☑	☑	☐	☑	ExportGas	gas	☑	☐	☐
FormationPetro...	Sensitivity_2	8	1000	from 0 to 20 step 1 ...	☑	☑	☑	☐	☑	ExportOil	oil	☑	☑	☐
FormationPetro...	Sensitivity_3	8	1000	from 0 to 20 step 1 ...	☐	☑	☐	☑	☑	ExportWater	water	☑	☑	☐

This window is made up with 3 parts:

- the left top part indicates the **list of opened documents** on which the calculations will be performed. It is possible to load a new document using the icon  ;
- The right top part enables (once the document is chosen) to select the bank of result where the calculations will be record. The following options can be used:
  -  enables to add a bank of results;
  -  enables to duplicate a bank of results;
  -  enables to delete a bank of results;
  -  enables to expand the different banks of results;
  -  enables to collapse the different banks of results;

  It is then enough to press on the button  to configure the calculations to be made.
- The bottom part enables to specify the options of wanted calculations.
  -  edits a computation;
  -  gives the access to the multiple changes;
  -  duplicates a computation;
  -  deletes a computation;

14. Curves

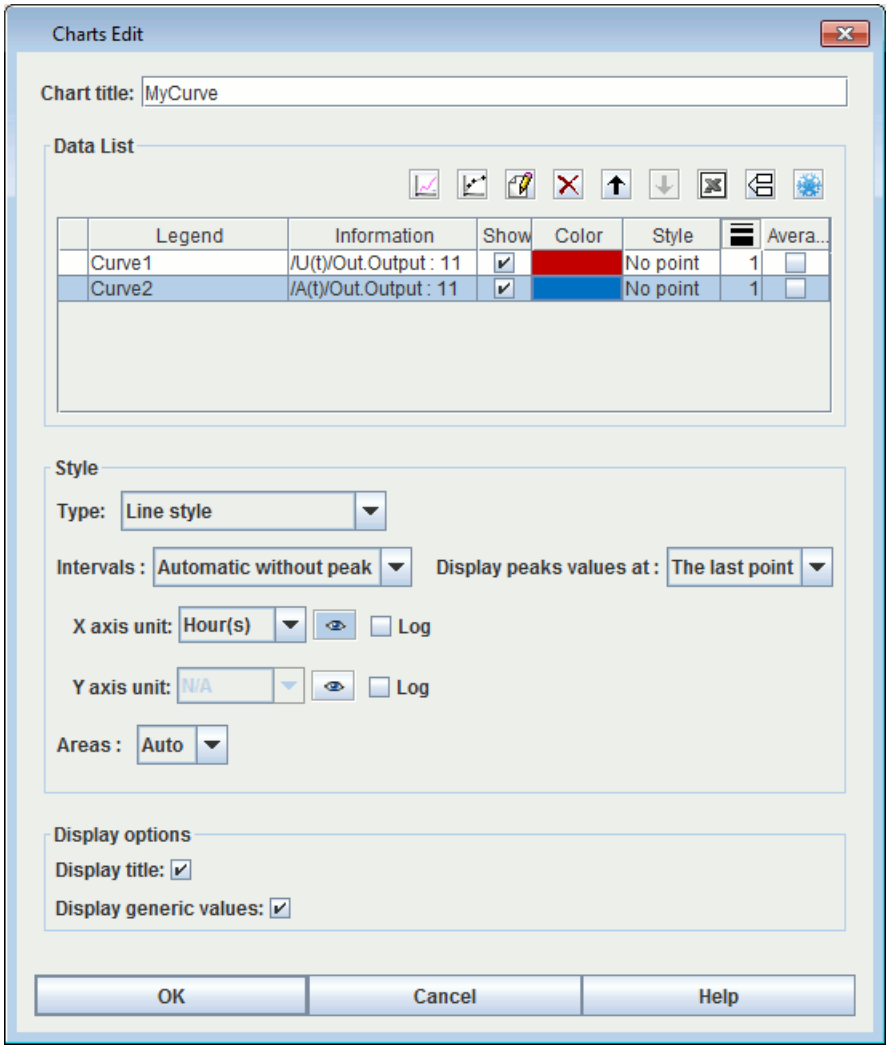
The curves can be drawn to study the model and the results better. To do this, click left on the corresponding icon on the vertical task bar then draw a box. This box will be the space assigned to displaying the curve(s). Initially it is only a white box with two axes without graduation.

Charts icon:

We must now define the curves to be drawn. To do this, click right on the box to display the **Charts Edit** window.

14.1. Edit curves window

The edit curves window is the same for all the GRIF modules.



The window is divided into several sections:

1. **Charts title**: enters a title for the graph.

2. **Data list**: this part includes a table with several columns in which the different curves on the graph are listed (name, description, display, curve colour, curve style, curve thickness, display average). A number of different buttons are available above this table.

-  : Selects a result of computations to display. It sends the user back to the Select results window to add a curve plot to the graph (see. Section 14.2.1, “Curves from data in result-bank”).
-  : Compares several results from different calculations for the same data. It sends the user to the Compare results window to add a curve plot to the graph (see Section 14.2.2, “Comparative curves from data in results bank ”).
- **Edit**: edits the plot of the selected curve.
- **Delete**  : deletes the plot of the curve selected on the graph.
- **Up**  : moves up the plot of the curve selected in the list.
- **Down**  : moves down the plot of the curve selected in the list.
- **Save**: saves the list of points calculated to plot the selected curves in .csv format. This export does not contain the generic values. To obtain an export with the generic values, right click on the curve and select **Individual export**.
- **Duplicate**: creates a new curve identical to the curve selected.
- **Freeze**  : freezes the display of the curve, which will no longer be updated automatically according computation results.

For each curve, the user can specify the colour, point style, line thickness and display.

3. **Computation options**: enables the user to enter settings for the computation (optional depending on the module).

4. **Style**: this section concerns the curve display.
   - **Type of style**: specifies the type of all the curves on the graph (line, histogram, etc.). N.B. In the case of a histogram, the bars that exceed the display area will be displayed in shading to show the user that s/he must change the display intervals so that the entire bar can be displayed.
   - **Intervals** defines the display limits for the curve. **Automatic without peak**: the graph will not display the "peaks" of specific cases of exceptional values which would make the graph illegible. Even if the peaks are not displayed on the graph, the user can display their values using the choices proposed in the option **Display peak values**.
   - **X and Y intervals**: specification of the display interval on the X and Y axes (default intervals or intervals defined by the user). The last function enables users to "zoom in" on the most interesting parts of the graph.
   - The axis unit can be selected according to the type of computation result. For example for units of time, the user can enter hours, days, months or years.
   - The  button determines whether or not the unit is displayed on the graph.
   - The **log** boxes are used to activate the logarithmic scale on the axis in question. N.B. 0 cannot be represented on a log scale, remember to enter a start value that is strictly positive (e.g. E-10). If 0 is entered, the log scale will begin at an arbitrary value E-15. Where the computation engine enables, the **trust interval** can be displayed by ticking the corresponding box.
   - **Areas**: distinguishes a range of values on a coloured background.
   - In the histogram style, a box can be ticked to create a cumulative histogram.

5. **Display options**: activates the **Display title** function (display graph title) and the **Display generic values** function (display min, max and average for each curve).

When a curve is edited, the edit curve window often includes 3 parts: the times at which the computations are performed, what has been calculated and the extra information (generic values) that should or should not be displayed below the curve.

Comment: it is sometimes necessary to refresh all the graphs in a document. This can be done using the **Tools / Refresh** command, or the keyboard shortcut F5 or the  icon.

14.2. Selection of results window

14.2.1. Curves from data in result-bank

When you click the **Add** button in the **Data list** part you reach a window for curves setup. Each curve displays data stored in the result-bank. The following window helps users to specify how to retrieve data.

- **Legend**: legend of the curve.
- **Computation selection**: select the computation in the result-bank.
- **Result to be displayed**: each computation contains many results. Select the one you want to be drawn.
- You can display a list a point whose X and Y will be taken from the data selected in the combo-boxes; or you can display the time spent in each zone.
- **Axes**: When a result is selected, select what must be in X-Axe and what must be in Y-Axe.
- **Value to be displayed**: Then you can display additional information about the result (min, max, average ....)

14.2.2. Comparative curves from data in results bank

When the user clicks on **Compare** in the **Data list** section, a window opens enabling the user to specify the curve to be plotted. S/he can then choose whether or not to display a result from several different computations. The following window is used to indicate the way in which the information is to be recovered.

- **Legend**: curve legend.
- **Computation selection**: the user can select the different computations to be used from the results bank. Hold down the Ctrl button to select several different computations.
- **Result to be displayed**: a computation often contains several results and this file tree structure is used to specify the result that the user wants to display.
- **Information to show**: indicates which data are to be displayed on the y-axis.
- **For which point?**: indicates which point of the computation is to be compared to the others.
- **Display x-axis values according to**: indicates which data must be displayed on the x-axis.
- **Value to be displayed**: finally, certain extra data can be displayed (min, max, average, etc.)

15. Database of parameters

In every GRIF module, a connection can be established with a database of parameters, to import parameters in GRIF. There are three ways to connect to a different database:

- connection to a .csv file
- connection to a .xls file
- other connection (via JDBC).

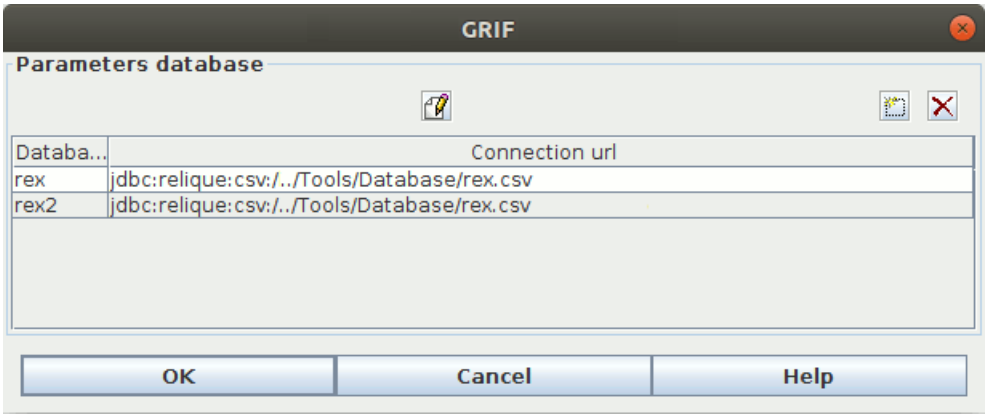
15.1. Format of the databases

The database must contain the identifier, the name and the value of the parameter. It is possible to add to the parameters more information, as the unit, the dimension and the description of the parameter. So we can have three to six columns, inquiring:

Data's type:	Possible values:
Parameter's identifier	Numbre, Text
Parameter's name	Text
Parameter's value	Number
Parameter's description	Text
Parameter's unit	HOUR : hours
	DAY : days
	MONTH : months
	YEAR : years
	HOUR_1 : $hours^{-1}$
	DAY_1 : $days^{-1}$
	MONTH_1 : $month^{-1}$
	YEAR_1 : $years^{-1}$
	FIT : Failure In Time (= 10^{-9} $hours^{-1}$)
Parameter's dimension	BOOLEAN, FACTOR, PROBABILITY, RATE, TIME, OTHER

15.2. Connect to a database

To access to the window to create the connections to databases, go to the menu **Data and Computations -> Parameters database -> Connections ...**. A window appears then:



From this window, it is possible to :

	Add a connection to a database.
	Modify a connection to an existing database. It opens the same window when adding a connection, but the fields are already filled by the data previously entered.
	Delete the selected connections of the databases.

15.2.1. Connection to a CSV file

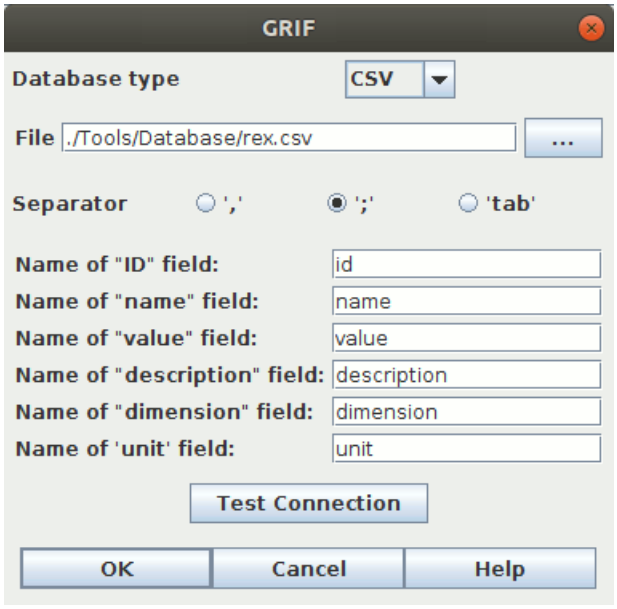
15.2.1.1. Form of the database

This type of connection is the simplest. The CSV file has for extension ".csv". It is a simple text file where the different fields are separated by commas, tabulations or semicolons.

```
ID,NOM,VALEUR,DESCRIPTION,DIMENSION
000001,Lambda,0.001,Exemple de Lamda,RATE
000002,Mu,0.01,Exemple de Mu,RATE
000003,Gamma,0.5,Exemple de Gamma,PROBABILITY
000004,ProdMax,1000.0,Exemple de Production maximum,OTHER
```

15.2.1.2. Connection

Once clicked on the button "Add a connection to a database", a window opens up:



This window has as a common base, the selection of the database, the fields for "ID", "name", "value", "description", "dimension" and "unit", and a button **Test Connection**. By clicking on this button, GRIF tries to connect to the database and so verifies the configuration provided by the user.

When adding a CSV database, the type **CSV** must be selected. A new field appears: the separators between the data. To sum up, there are three steps to add a connection to a CSV database:

- First, fill the path of the CSV file in. A file explorer is at your disposal (button **...**).
- Then, specify the type of the separators used in the CSV file.
- Finally, enter the six fields names of the CSV file. (Or only the ID, name and value fields) (Uppercase letters are taken into account as lowercase)

Warning : It's important to note that when creating a connection to a CSV database, you must have all of the data on a single sheet.

15.2.2. Connection to a XLS file

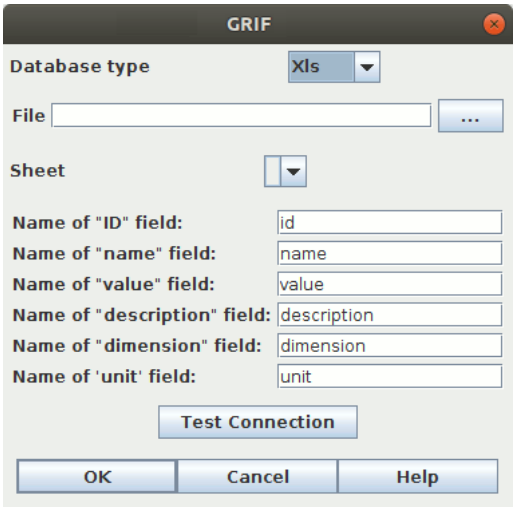
15.2.2.1. Form of the database

The databases of the .xls or .xlsx extensions correspond to EXCEL files. Here is an example of an EXCEL Database :

	A	B	C	D	E	F
1	ID	NOM	VALEUR	DESCRIPTION	DIMENSION	
2	1	Lambda	0.001	Exemple de Lamda	RATE	
3	2	Mu	0.01	Exemple de Mu	RATE	
4	3	Gamma	0.5	Exemple de Gamma	PROBABILITY	
5	4	ProdMax	1000.0	Exemple de Production maximum	OTHER	
6						
7						

15.2.2.2. Connection

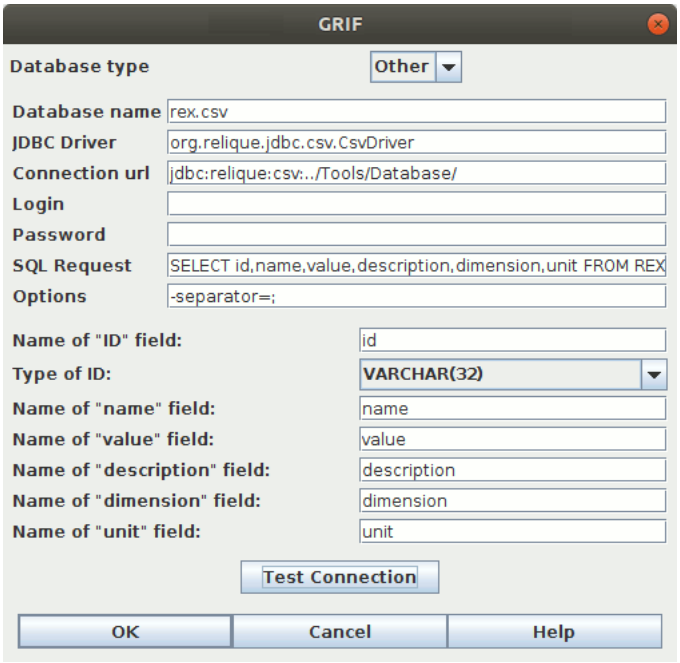
To connect GRIF to this database, select the **XLS** type in the connection window. The window is now as followed:



Sheet is the sheet's name where the data are located, and will be filled once a valid path to an EXCEL file has been entered.

15.2.3. Connection to a database (with a JDBC connection)

GRIF can connect to any database with JDBC, as long as the database follows the same rules of the databases seen earlier. The window for that kind of connection has multiples fields to fill:



1. **Driver JDBC** is the name of the JDBC driver (ex : sun.jdbc.odbc.JdbcOdbcDriver)

2. **Connection URL** is the URL of the database.

3. The fields **Login** and **Password** can be left empty.

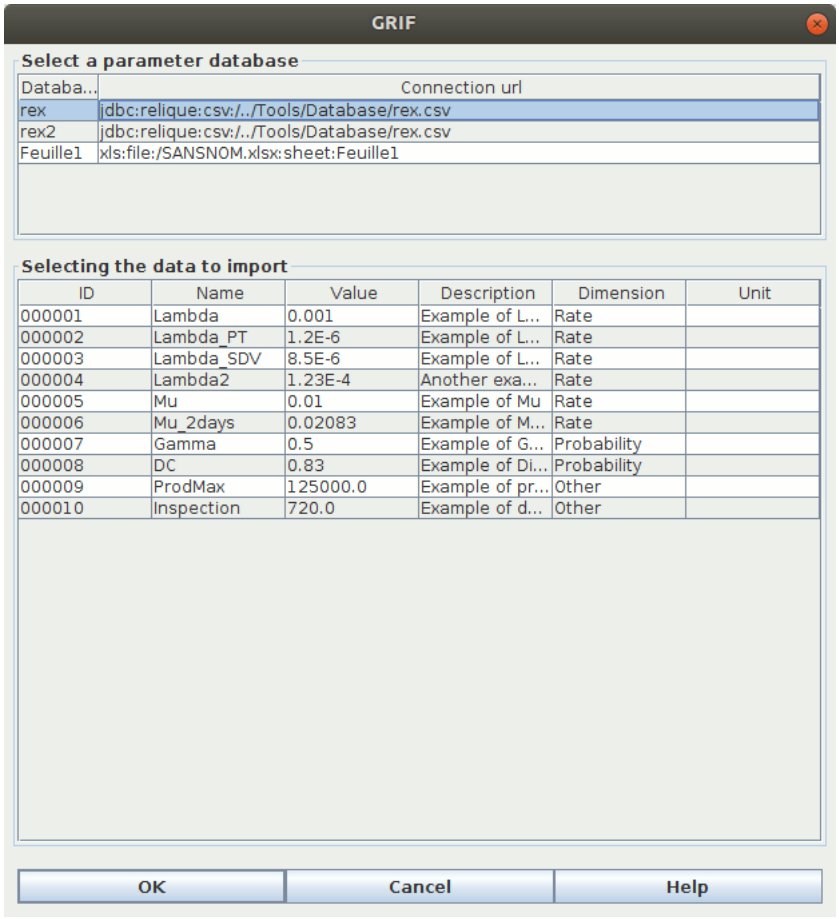
4. The SQL request **SELECT id,name,value,description,dimension,unit FROM REX** is used to gather the dates.

5. **Option** field inform of all of the database's options: separator, ...

Once a connection with a database is ready, GRIF can now import a set of parameters from the database, but also updates these parameters when modifications has been made in the databases, or recreate the links of these parameters so they can now take the values of another database.

15.3. Import parameters from a connected database

Once a database is connected, GRIF can import a set of parameters from the database, via the window reachable by the **Data and computations -> Parameters database -> Copy parameters from database ...** menu.

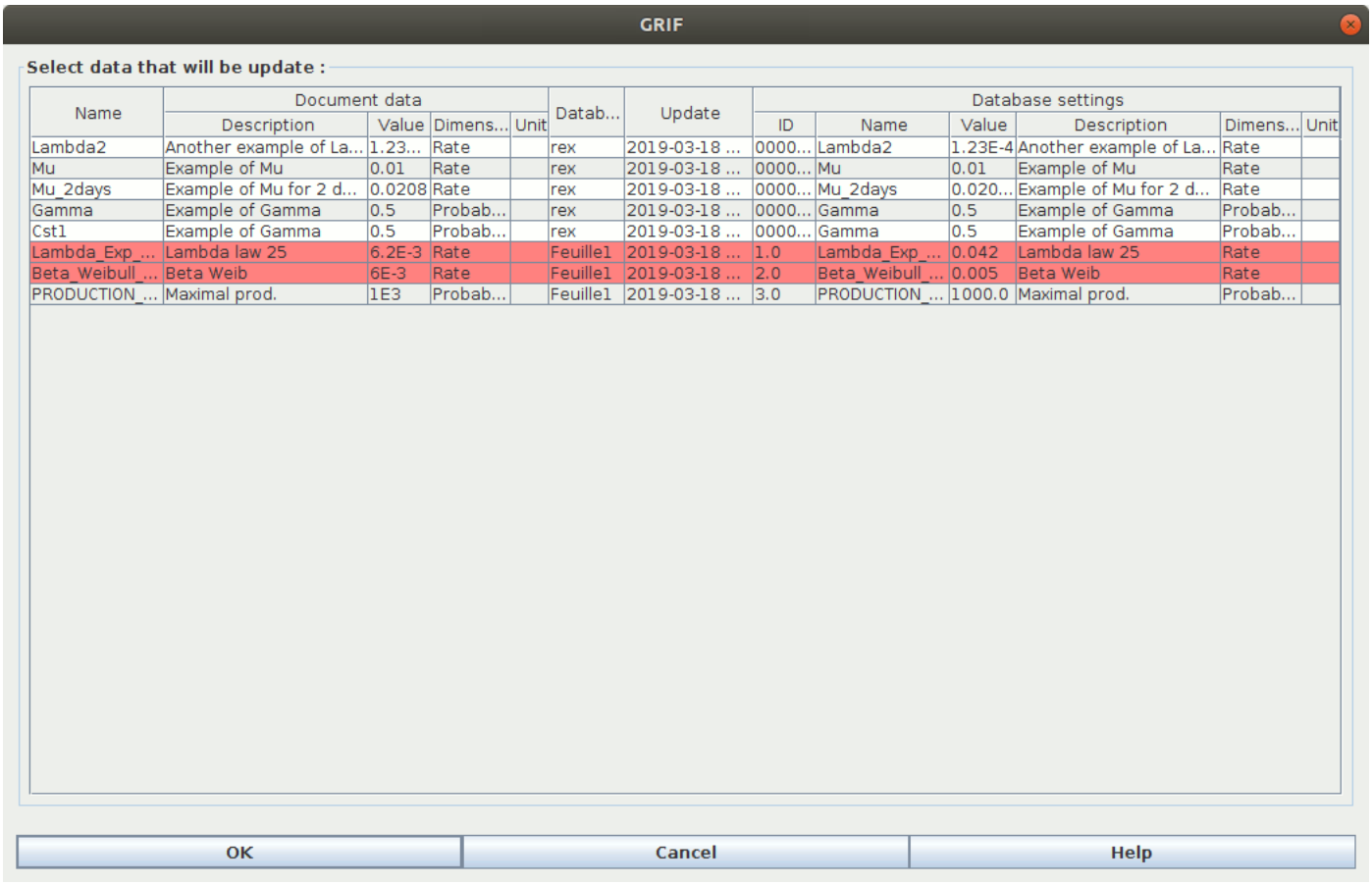


Select the parameters you want to import, and click on **OK**. The parameters are now created and imported in GRIF. The created parameters have the same names than the database's parameters, and the fields "Description" or "Dimension" are identical of those found in the database.

It is important to underline that it is possible to manually create a parameter in GRIF, and then with its **Add-On** menu, assign the parameter's value of the connected database. This operation is detailed in this link.

15.4. Update of the parameters from the database

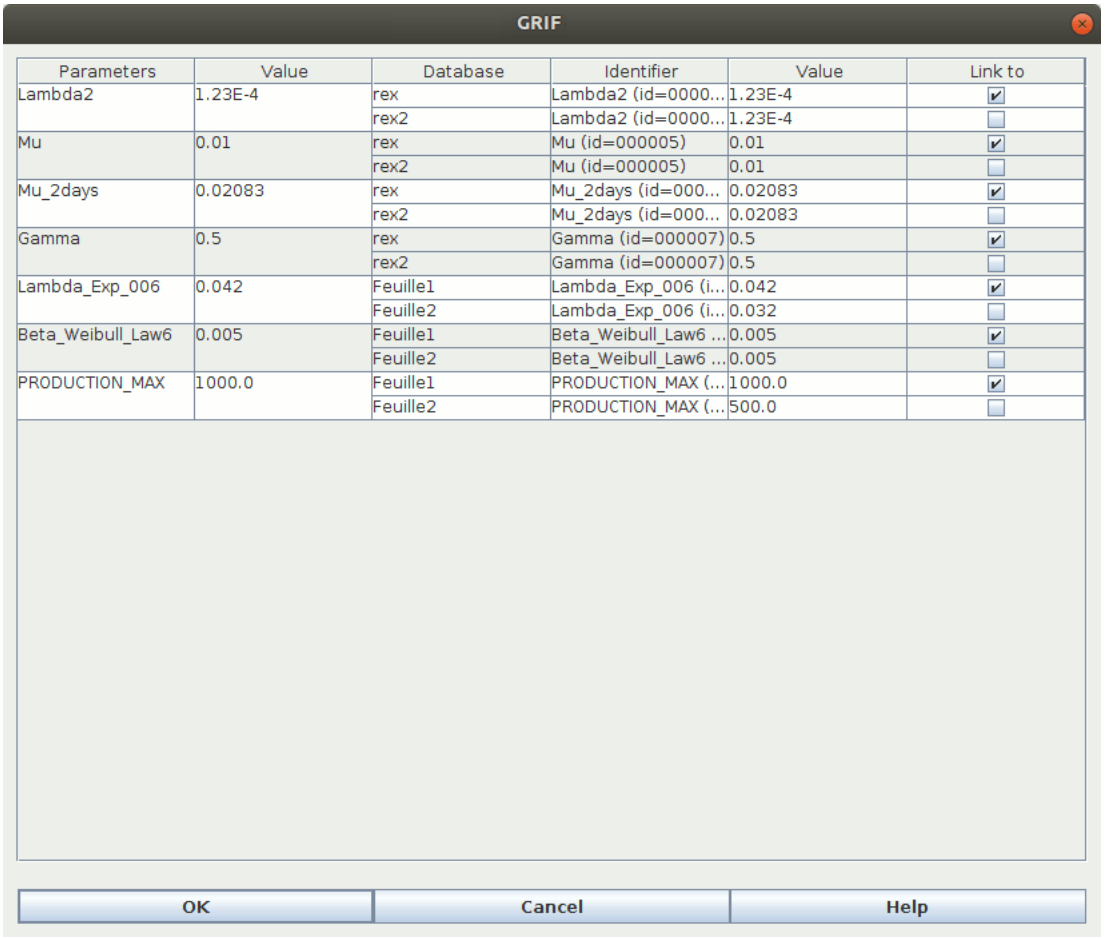
When an user, who has updated some of his data in his database, wants to have these modifications done on his parameters in GRIF too, he can then use the update action, from the **Data and Computations -> Parameters database -> Update from database ...** menu:



This window shows the parameters in GRIF which are connected to parameters from the databases. The red lines correspond to data which have been modified in the database. If the user wants to update some of his parameters in GRIF, he must select the lines of the wanted parameters, and then press the **OK** button. The parameters are now updated.

15.5. Rebuild of the links to the database

It is possible to modify an existing parameter's connection in GRIF, by changing the database of its associated parameter. However the parameter can only connect to the parameters with the same name. This action is available by the **Data and Computations -> Parameters database -> Rebuild links to the database** menu.



Here we can see the different parameters of the databases, which are imported in GRIF, and which are on multiples databases. So on the line of the parameters you want to rebuild the links, select the right database, and then validate your modifications by clicking on **OK**. GRIF then update the values of the parameters by rebuilding the links.

16. Import of profiles from an Excel file

GRIF provides a new tool to import profiles from an Excel file. The user, instead of creating profiles one by one, can now generate them thanks to this tool.

16.1. Format of the Excel file

The Excel file containing the profiles must respect some structural requirements. The file must have at least one profile name, one row or column of times and, for every profile, at least one row or column of flow values at time t given by the times row/column. A sample Excel file must look like this:

Times	**Profile 1**	Oil	Gas	**Profile 2**	Oil	Gas
01/01/19		0,5	1		0,5	1
01/02/19		1,5	3		1,5	3
01/03/19		2,5	5		2,5	5
01/04/19		3,5	7		3,5	7
01/05/19		4,5	9		4,5	9
01/06/19		5,5	11		5,5	11

Here we have the profiles names in bold font, the times in yellow and the flows in orange(Oil) and blue(Gas). The first column of Oil flow is assigned to the first profile, the second column is assigned to the second profile, etc... There must be an offset of rows or columns between every profile, which must be respected in order to have consistent profiles. In this example below, the arrows indicate an offset of three columns between each profiles.

Times	**Profile 1**	Oil	Gas	**Profile 2**	Oil	Gas
01/01/19		0,5	1		0,5	1
01/02/19		1,5	3		1,5	3
01/03/19		2,5	5		2,5	5
01/04/19		3,5	7		3,5	7
01/05/19		4,5	9		4,5	9
01/06/19		5,5	11		5,5	11

The time range is common for the profiles in the above example, but it is possible to assign a time range for each profile, and so have different times for each flows, as long as the offsets between the profiles are respected.

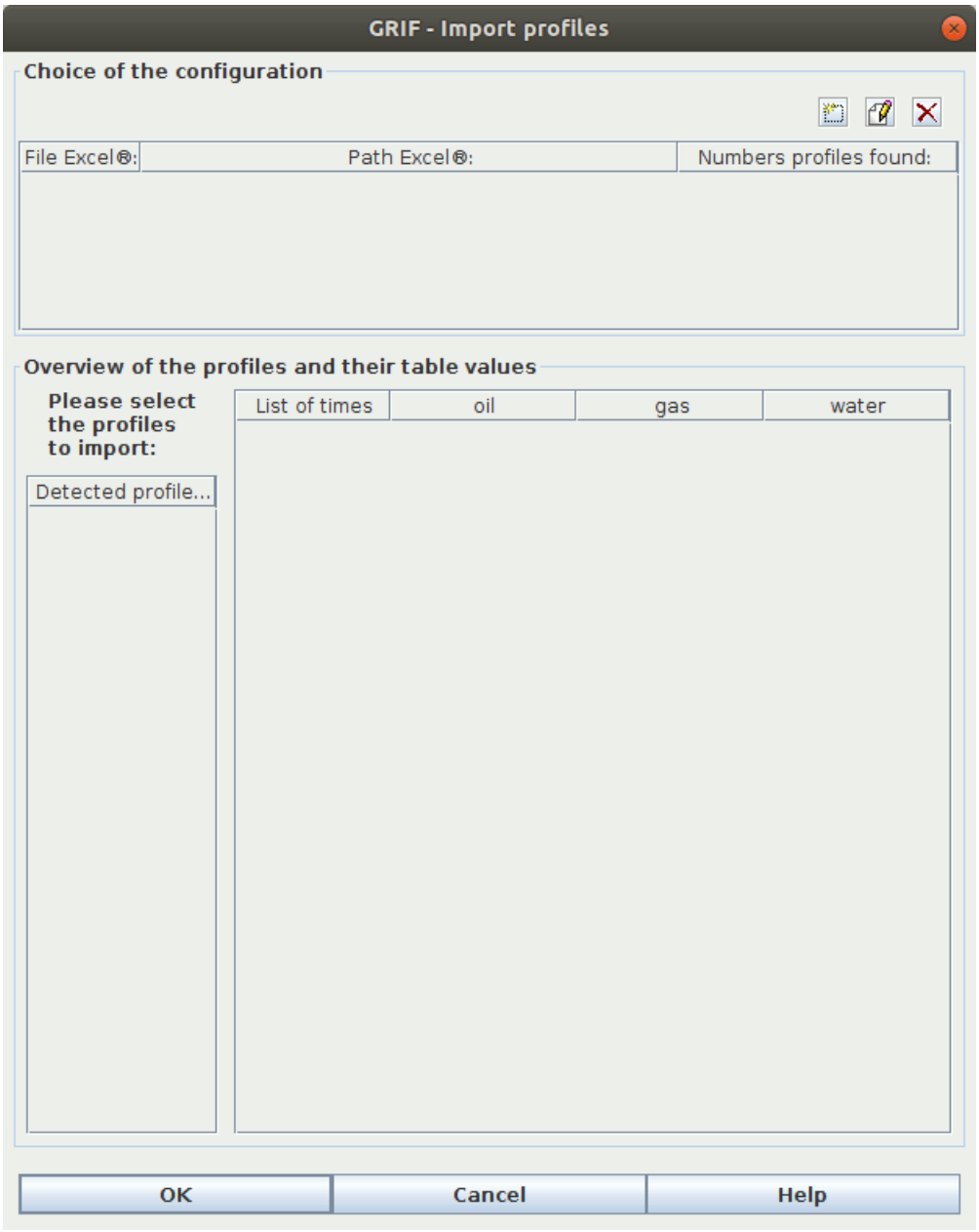
Profile 1	Time	Oil	Gas	**Profile2**	Time	Oil	Gas
	01/01/19	0,5	1		01/01/19	0,5	1
	01/02/19	1,5	3		01/02/19	1,5	3
	01/03/19	2,5	5		01/03/19	2,5	5
					01/04/19	3,5	7
					01/05/19	4,5	9
					01/06/19	5,5	11

However, it is not possible to treat in one action profiles with different number of flows, because it would directly affect the offset between the profiles.

		Offset of 2		Offset of 2			
Times	**Profile 1**	Oil	**Profile 2**	Oil	Gas	**Profile 3**	Oil
01/01/19		0,5		0,5	1		0,5
01/02/19		1,5		1,5	3		1,5
01/03/19		2,5		2,5	5		2,5
01/04/19		3,5		3,5	7		3,5
01/05/19		4,5		4,5	9		4,5
01/06/19		5,5		5,5	11		5,5

16.2. Import some profiles

To import profiles, access to the menu **Data and Computations -> Imports profiles from an Excel file-> Imports ...** A window opens:



This window is split in two parts. The first part, called **Choice of the configuration**, is in charge of the management of the configurations of Excel files, while the second part **Overview of the profiles and their table values** list the profiles of a selected configuration, and display the flows table of a selected profile.

The first part **Choice of the configuration** is composed of the configurations table, informing the Excel file name of a configuration, the path of this Excel file and the number of profiles found in this Excel file. To interact with these configurations, there are three buttons:

	Add a configuration (open a Wizard)
	Modify an existing configuration, which will reopen the Wizard with the parameters previously given.
	Suppress the selected configuration(s).

16.2.1. Wizard of the configuration creation

To assist the user in creating a profile recovery configuration, GRIF includes a wizard, which will create this configuration step by step. This wizard is a window divided in two main parts:

- The left part display information related to the current step (Profiles' names, Times or Flows)
- The right part is an overview of the Excel file, with the profiles' names, times and flows highlighted according to the data entered by the user in the left part.
- In the right part, a button **Overview** is available to know the selected profiles' names, times or flows with the current configuration.
- In the case of an error made in a previous step, the wizard has a button **Go back**, which make the wizard go to the previous step, with the data previously entered.

To illustrate the use of this Wizard, an Excel File (see below), containing profiles, will be used as an example to get its configuration step by step.

	A	B	C	D
1	Profile 1	Times	Oil	Gas
2		01/01/18	1	11
3		01/02/18	2	12
4		01/03/18	3	13
5		01/04/18	4	14
6		01/05/18	5	15
7		01/06/18	6	16
8				
9	Profile 2	Times	Oil	Gas
10		01/01/18	1	11
11		01/02/18	2	12
12		01/03/18	3	13
13		01/04/18	4	14
14		01/05/18	5	15
15		01/06/18	6	16
16				
17	Profile 3	Times	Oil	Gas
18		01/01/18	1	11
19		01/02/18	2	12
20		01/03/18	3	13
21		01/04/18	4	14
22		01/05/18	5	15
23		01/06/18	6	16
24				
25	Profile 4	Times	Oil	Gas
26		01/01/18	1	11
27		01/02/18	2	12
28		01/03/18	3	13
29		01/04/18	4	14
30		01/05/18	5	15
31		01/06/18	6	16

16.2.1.1. Step 1: The profiles' names

For the first step, the user has to load the Excel file, and retrieve the profiles' names. Multiples fields must be filled in to retrieve the profiles' names:

- The field **Excel file** has to be filled by the Excel file, containing the profiles to retrieve. An Explorer is available with the button  . Once the file path is entered, click on the button **OK** to display the Excel file in the right overview.
- The field **Number of profiles to import** needs to be filled by the number of profiles the user wants to retrieve (not to be confused with the number of profiles found in the file)

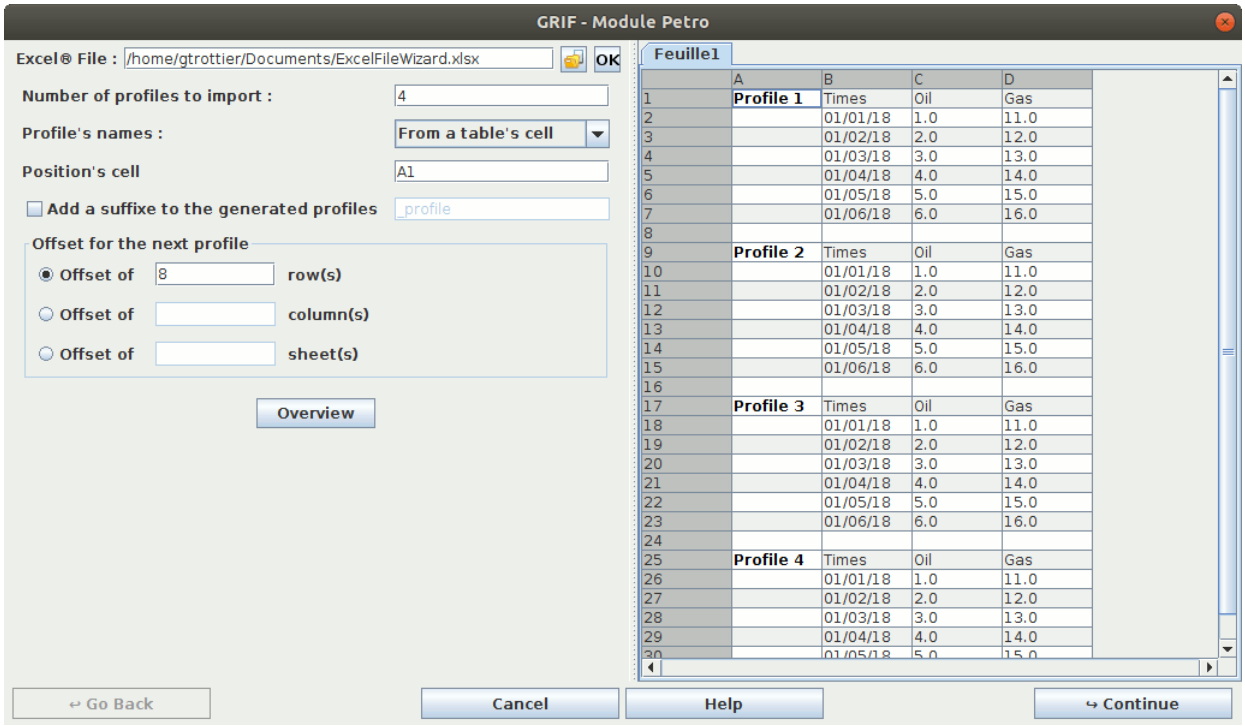
- The field **Profile's names** inform about the method of acquisition of the profile's names. There is two specific cases: **From a table's cell**, which get directly the profile's names from the value of the selected cells, or **The names of the sheets** which get the profile's names by the tiles of the selected sheets. Note that this last method only enables an offset by sheet.

- The field **Position's cell** needs to be filled by the coordinate of the cell of the first profile's names (where the offset begins). The coordinate can be manually entered with the Excel coordinates (Example: A6), or can be filled by a click on a cell in the Excel overview, which will automatically enter the coordinate of the cell in this field.

- If ticked, the field **Add a suffix to the generated profiles** allow the user to add a suffix at every generated profile.

- The panel **Offset for the next profile** gives the number of rows, columns or sheets are between two profiles. This offset is then used to detect all the profiles' name.

In the example, we want to import four profiles from given cells, beginning at the cell A1, with an offset of eight rows. We don't want any suffix to the generated profiles so we let the checkbox empty. This is now the overview:



	A	B	C	D
1	**Profile 1**	Times	Oil	Gas
2		01/01/18	1.0	11.0
3		01/02/18	2.0	12.0
4		01/03/18	3.0	13.0
5		01/04/18	4.0	14.0
6		01/05/18	5.0	15.0
7		01/06/18	6.0	16.0
8				
9	**Profile 2**	Times	Oil	Gas
10		01/01/18	1.0	11.0
11		01/02/18	2.0	12.0
12		01/03/18	3.0	13.0
13		01/04/18	4.0	14.0
14		01/05/18	5.0	15.0
15		01/06/18	6.0	16.0
16				
17	**Profile 3**	Times	Oil	Gas
18		01/01/18	1.0	11.0
19		01/02/18	2.0	12.0
20		01/03/18	3.0	13.0
21		01/04/18	4.0	14.0
22		01/05/18	5.0	15.0
23		01/06/18	6.0	16.0
24				
25	**Profile 4**	Times	Oil	Gas
26		01/01/18	1.0	11.0
27		01/02/18	2.0	12.0
28		01/03/18	3.0	13.0
29		01/04/18	4.0	14.0
30		01/05/18	5.0	15.0

16.2.1.2. Step 2: The times

The second step is to retrieve the range of times for every profile.

- The field **Position of the cell of the times** works in the same way as the field of the profiles' name cell: the coordinate of the cell can be entered manually or click on a cell in the Excel overview to have its coordinate written in the field.

- The field **Search of the times** inform the direction of the time range from the first cell. These iterations stop at the first empty cell encountered.

- The **Times type** field make the user able to specify the time format, which can be a duration or a date. In the case of a **Time in duration**, the user has the choice between a duration in Hour(s), Day(s) or Year(s). In the case of a **Time in date**, the user needs to enter the format of the date, respecting the times units (d for days, M for months and y for years) as well as the units separators (/, -, .). Here are some examples:

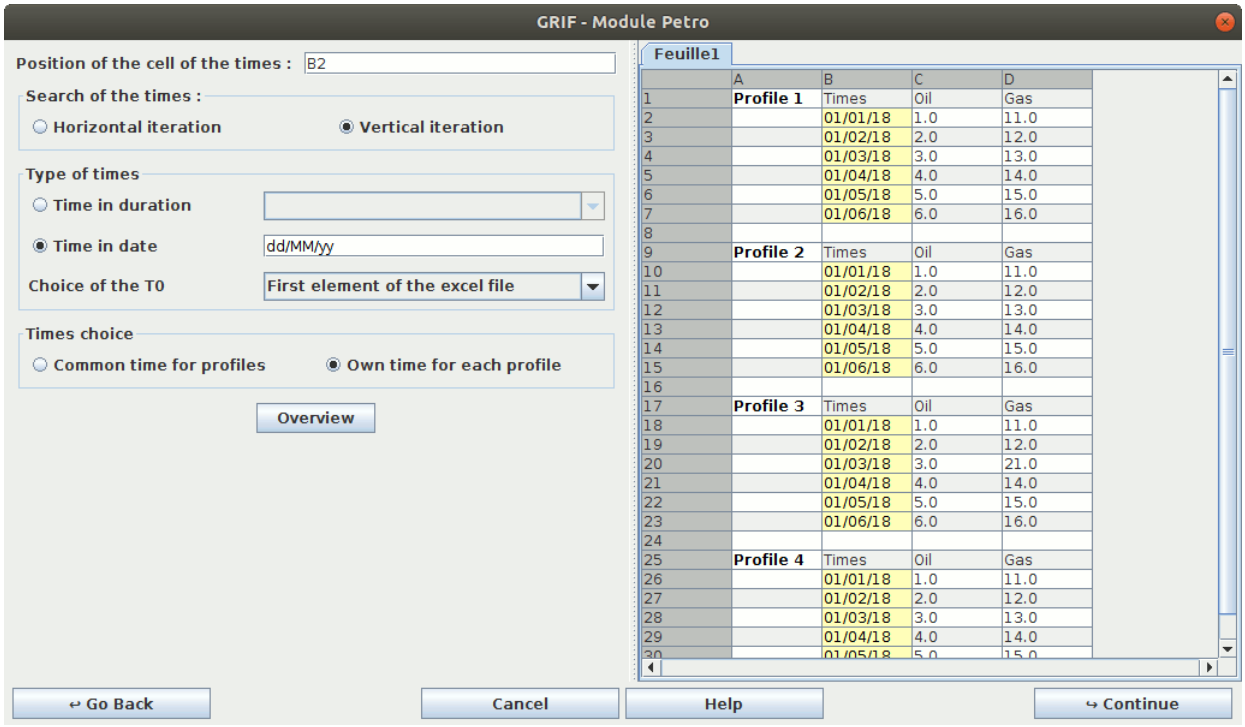
yyyy.MM.dd	2019.01.10
dd-MMM-yyyy	10-Jan.-2018

MMMMM.dd.yyyy	January.10.2018
yyyy	2018

If the type of times is in date, then the user has to inform his **Choice of the T0**. It can be the **First element of the Excel file**, or the **Document's start date**

- Finally, the field **Times choice** suggest to the user to have either a common time range for all profiles, or a time range specific to each profile.

For the example, the time range start at the cell B2, goes vertically down, and uses dd/MM/yy as date's format. T0 in this example is the first element of the time range. In this example, the user can choose either Common Times or Own times for each profiles, since profile's time ranges are all the same.



16.2.1.3. Step 3: Intermediate step for the flows

An intermediate step ask the user to fill in the number of flow in the field **Number of flows to import**, as well as the direction of the flows (Vertically or Horizontally).

This window being simplistic and bringing no modifications on the Excel overview, the choices made for the example are detailed without screenshot: The profiles have two flows: Oil and Gas. We decide to import the two flows, and the ranges will be selected vertically.

The next step will then have two substeps (so two windows): the first window will treat the Oil flow, while the second one will treat the Gas flow.

16.2.1.4. Step 4: The flows

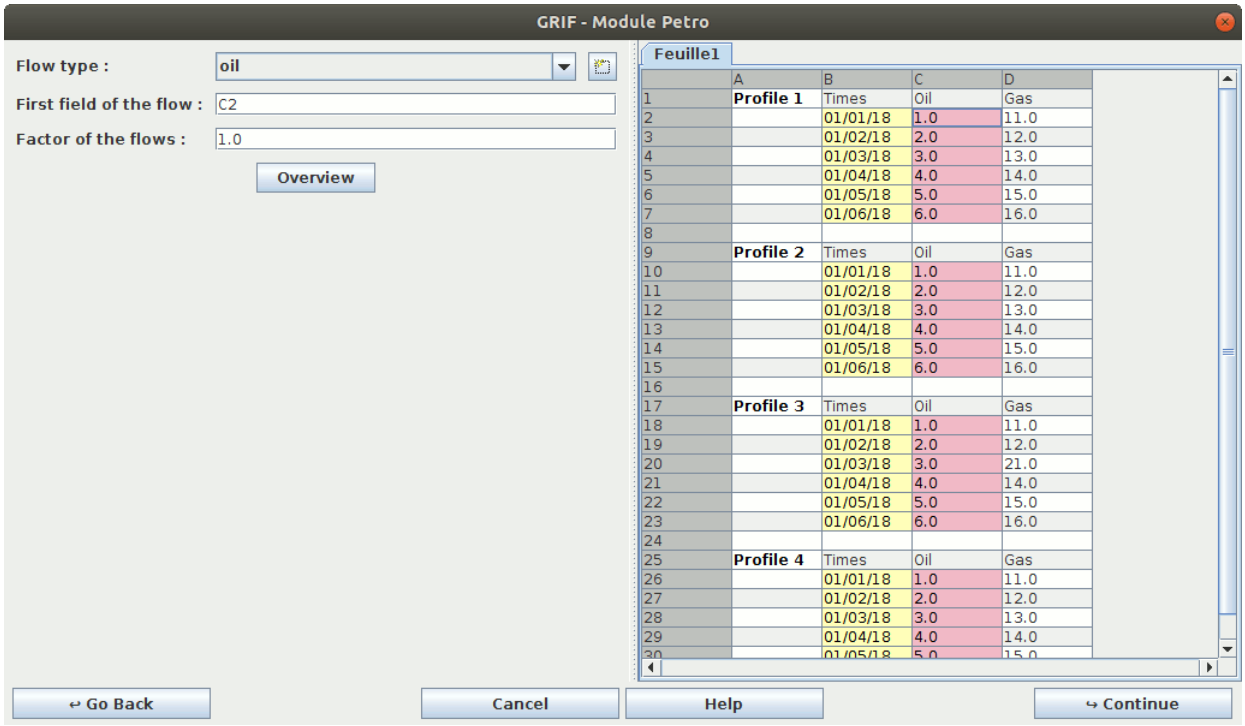
The last step is about how to retrieve the flows.

- The **Flow types** contained in a dropdown list, are the default GRIF's flows. The user needs to select the wanted flow, and in the case where his flow does not exist, he can create a new one by clicking on the button . This button opens the flow creation window, detailed in this part. Section 3.2, “Flows configuration”

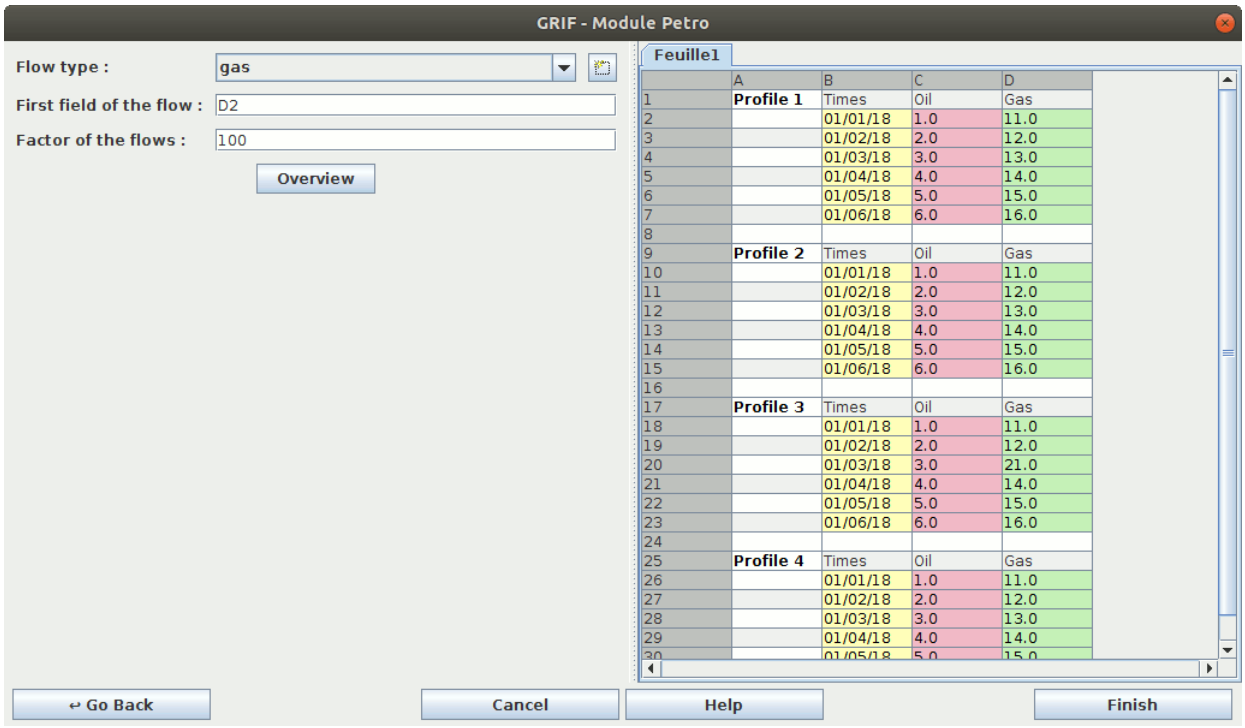
- The **First field of the flow** is the coordinate of the first cell of the flow's range. As the others cells fields encountered before, it can be modified by a click on the Excel overview.

- The **Factor of the flow** is a factor applied on the values of the flows at the end of the wizard. Its main purpose is when the flow has a unit, like tons or kilograms.

The first flow to treat is the flow Oil. We select the 'Oil' type in the **Flow type**, the coordinate of the first cell is C2, and the direction of the flow range, entered in the previous step, is vertical. In this example, the values of the flow correspond to the number of barrels per day. So we let factor's default value, set to one.



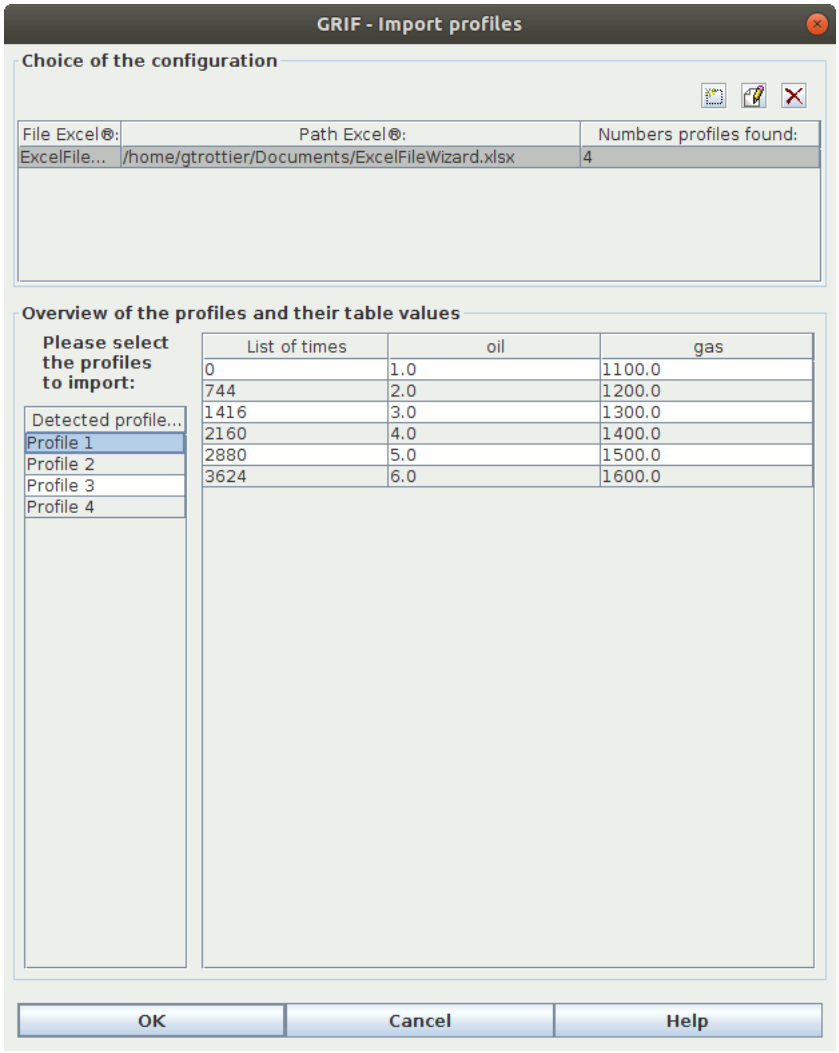
For the Gas flow, we select the Gas type, which is one of the GRIF default type too, and the first cell of this flow is D2. The unit of the values of the flow is in hectometers cube, so we have to enter one hundred as the factor of this flow.



The configuration is now ready, click on the **Finish** to validate the configuration.

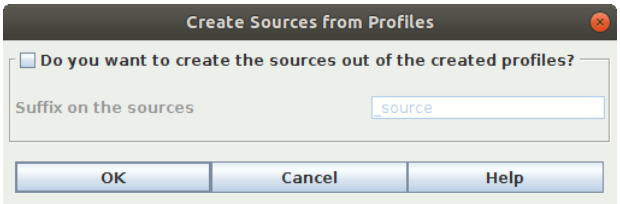
16.2.2. Import the created profiles

Once a configuration has been made and is displayed in the configuration table, the user can select one of the configurations, to list the created profiles. Then, the user can select a profile to have an overview of its flows table, and he can then select the profiles he wants to import in GRIF.



16.2.3. Creation of the sources from the imported profiles

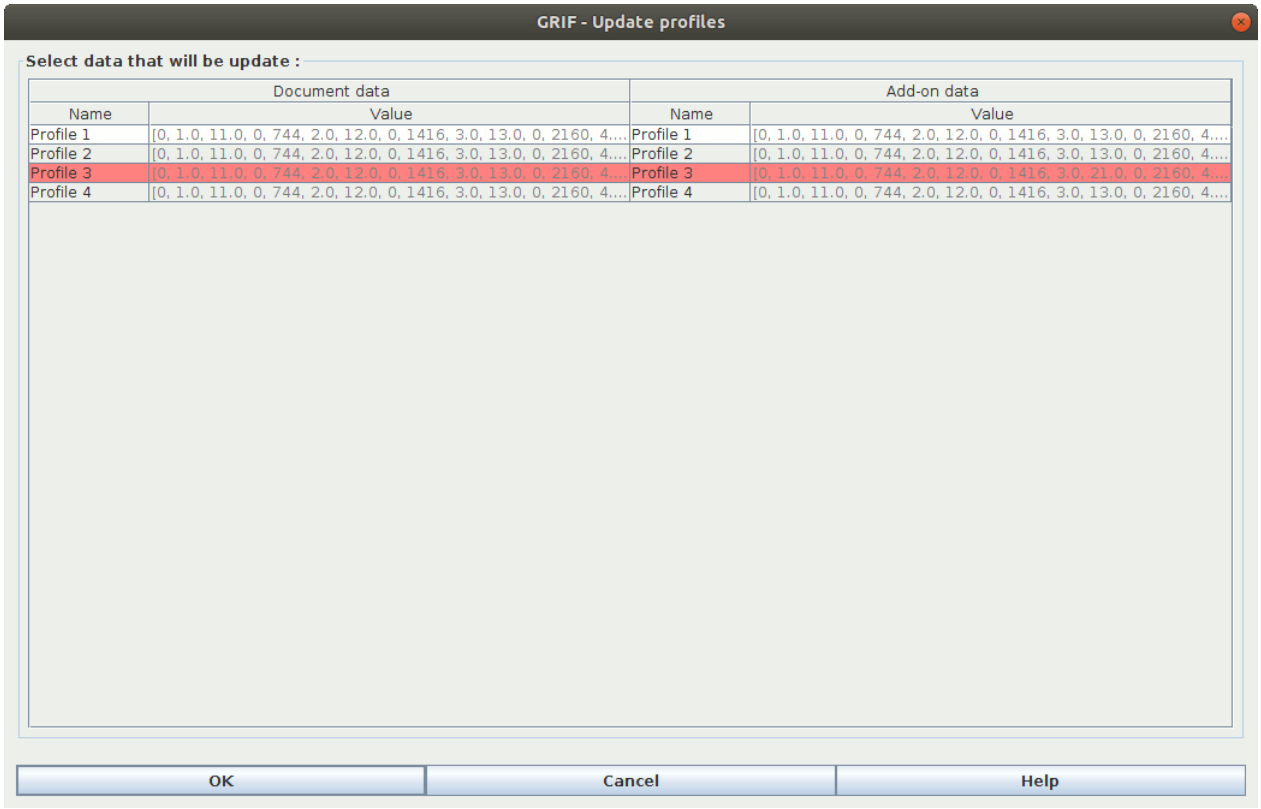
GRIF creates the sources out of the imported profiles:



To create the sources, tick the checkbox. The user may choose to specify a suffix to add to the created sources, the same way the user can add a suffix to the generated profiles. However, the suffix of the profiles and the suffix of the sources will not be cumulated: a generated source will not have the suffix of the profile.

16.3. Update of the data profiles

If the data profiles of the Excel file have been modified, and so GRIF's profiles are out of date, it is possible to update these profiles with the command in the menu **Data and Computations -> Import Profiles from an Excel File -> Update from the Excel...**. A window then opens:



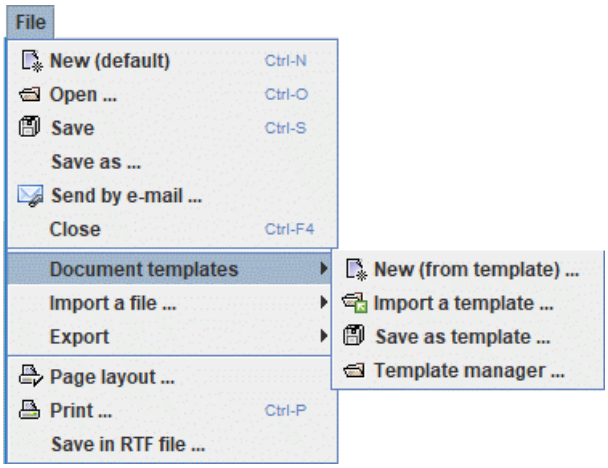
The profiles which have different data from their associated profiles in the Excel file are displayed in red. To update them, the user has to select the profiles he wants to update, and by a click on the **OK** button, GRIF updates these profiles and their associated sources.

17. Save

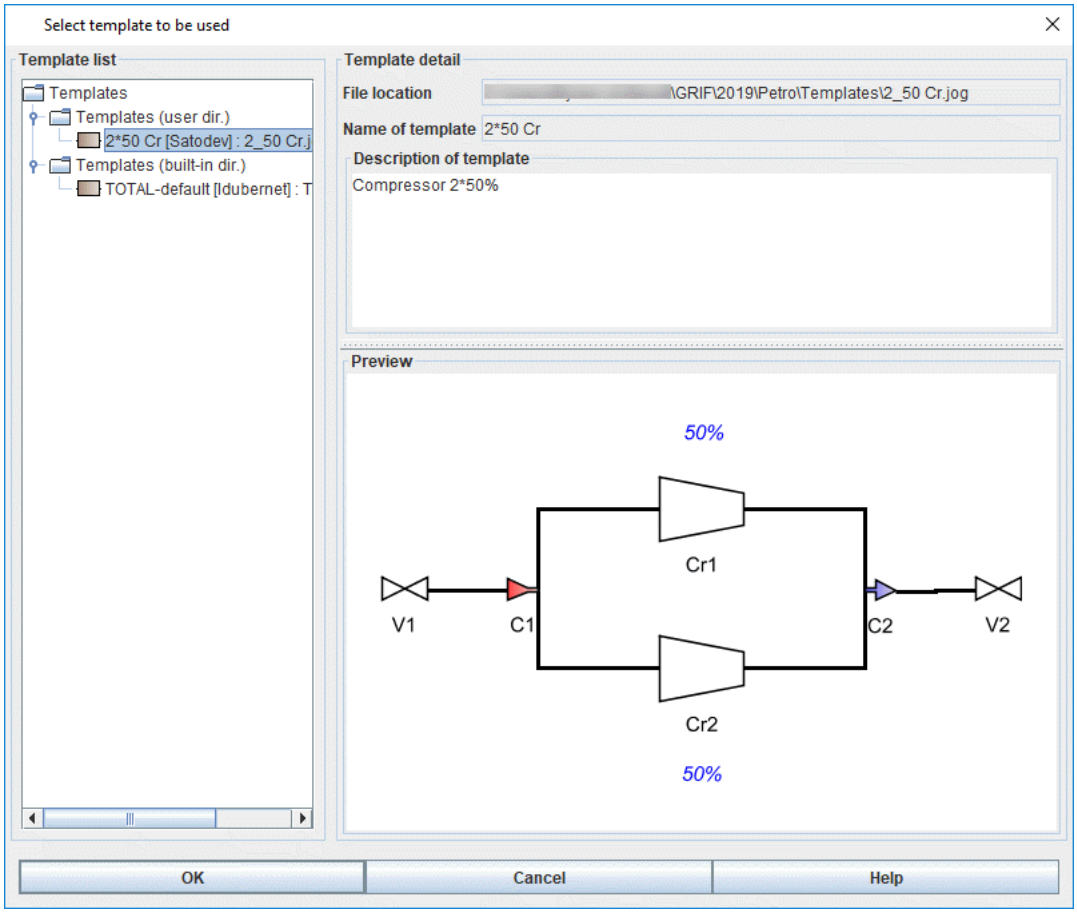
Here is a summary of all the information which can be saved from a same model.

17.1. Document template

It is possible to use an existing document as base to create a new document or as a part of a document. This functionality is accessible in **File - Document template** menu.

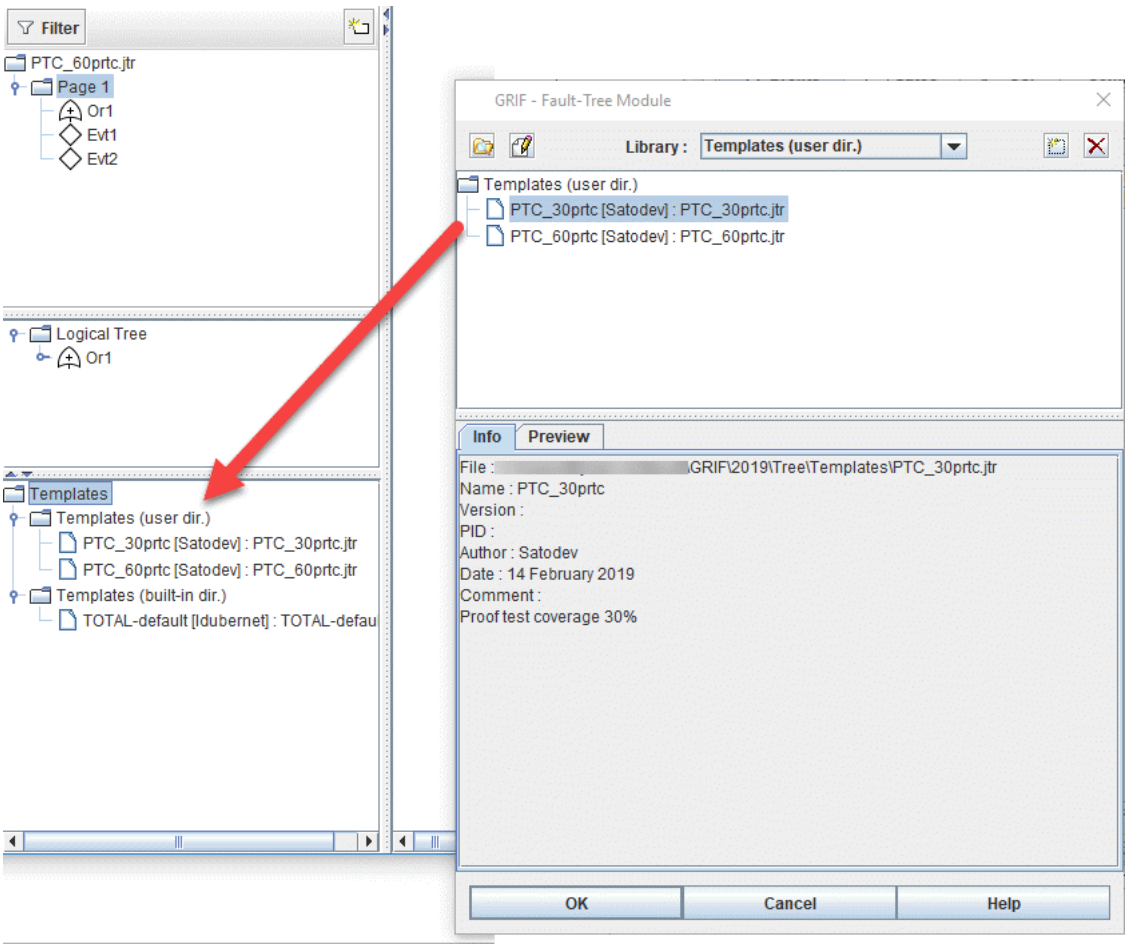


New (from template)... menu enables to open a new document and to initialize it with data from a model already build. A window appears to select the existing model.

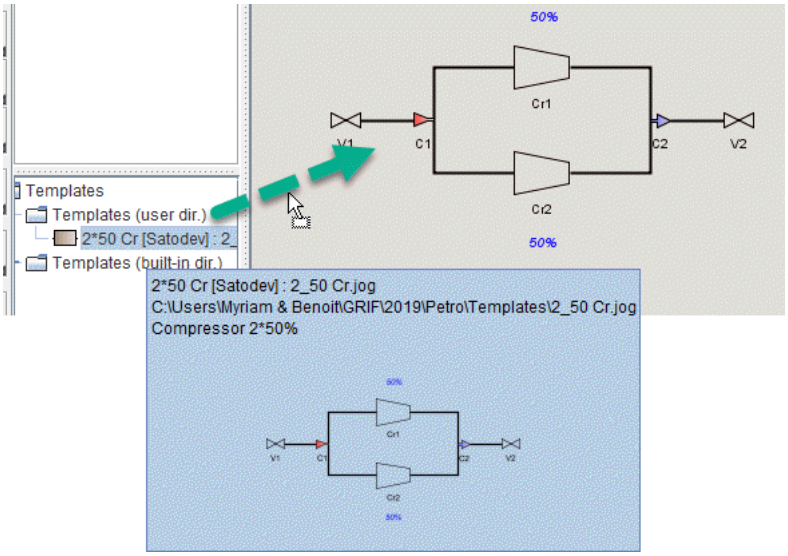


Import a template... menu enables to add to the current document data from a model already build.

Save as template enables to save the current document as template in the Template directory of the module. Once saved as a template, the document appears in the Template tree of the GRIF window as well as in the **Template Manager**.

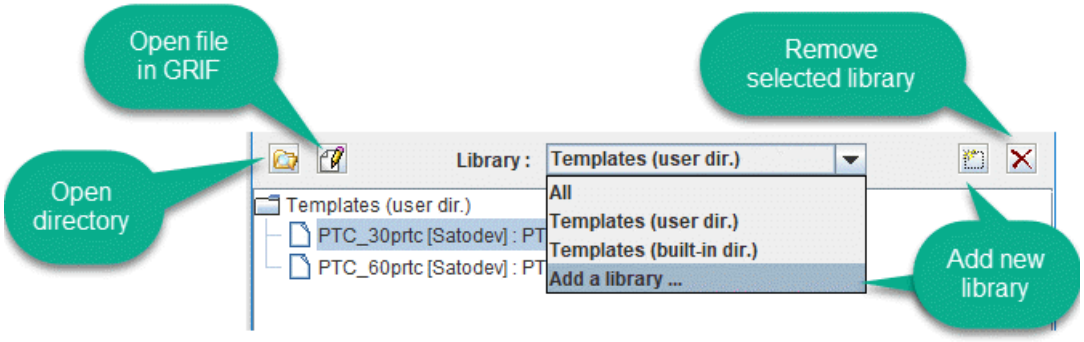


It is possible to create new files from this model using **New (from a template)...** action. A drag and drop to the templates from the input area enables to import the model quickly.



Save as default template menu enables to save the current document as default model in the module template directory. This model will also be the default model of the module. It will be used as base for creating a new document when **File - New (default)** action is used.

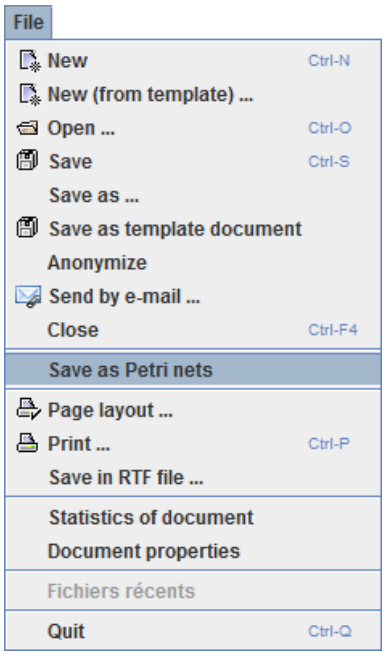
Template manager menu opens a window to manage the template of the document. New document libraries can be added/deleted. To add a new library it is necessary to select a directory of the file system. The tool analyzes the documents in this directory and builds a library that can be used by GRIF based on the compatible documents found.



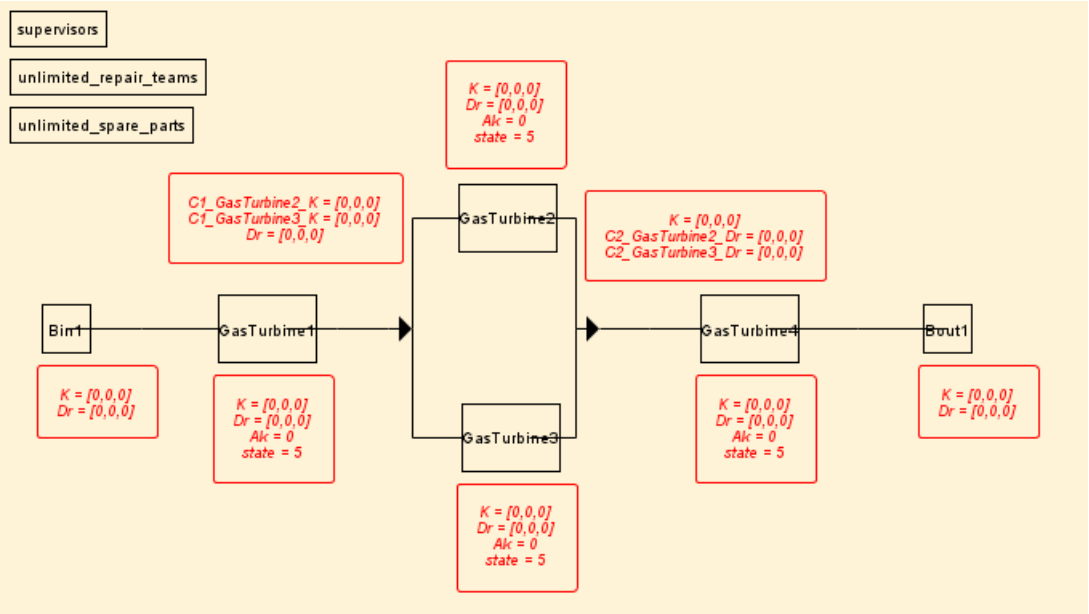
17.2. Petri Net model

The Petri net generated by the stochastic diagram can be saved. This ".jpp" file can then be reloaded and reworked using the Petri12 module. This can be very useful for different reasons:

- to understand the way in which the computations are performed (by looking at the variables automatically generated),
- to debug the model (using the Petri12 "Step by step simulation mode",
- to complete the model (addition of places, addition of variables, etc.),
- to access more parameters,
- ...

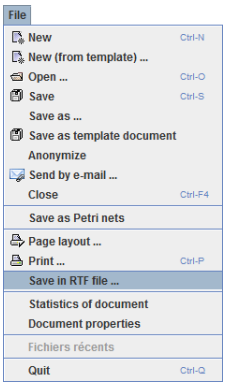


In the ".jpp" file, each block has become a group associated with a dynamic field which enables the user to observe the values from the different tables and variables for the block. The same applies to each connector. The data to be displayed in the dynamic fields can be configured in the document options.



17.3. RTF File

A model can also be saved in RTF format. This enables the saved model to be reloaded in WORD to insert the graphical part of the model in any document. To do this, go into menu **File - Save in RTF file...**.



There is another way to insert model in a report.

Select the part of the model, copy it, and paste it in Microsoft WORD or other software.

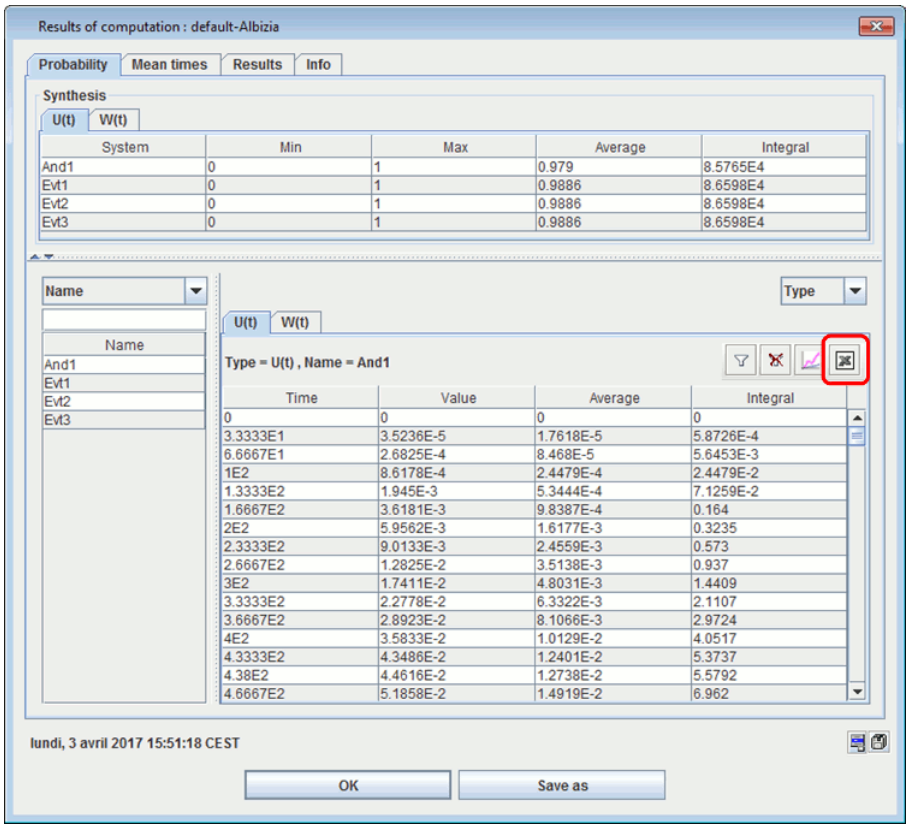
17.4. Input data

When the input data for the computation engine is generated, it can be saved. This type of file has the ".don" extension. These files can therefore be modified using a text editor then reloaded to launch computations on them (for example). This action should be made only by advanced user.

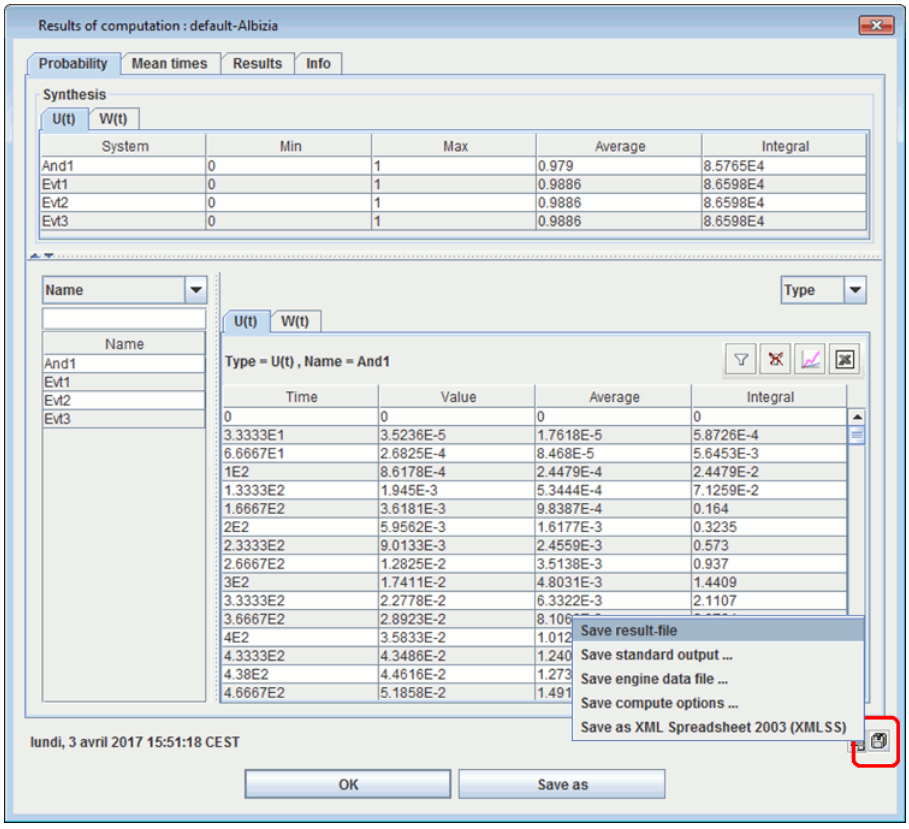
17.5. Results

Computation results can be saved in different formats:

1. Export of a table in particular in .csv format:



2. Export entire set of results:

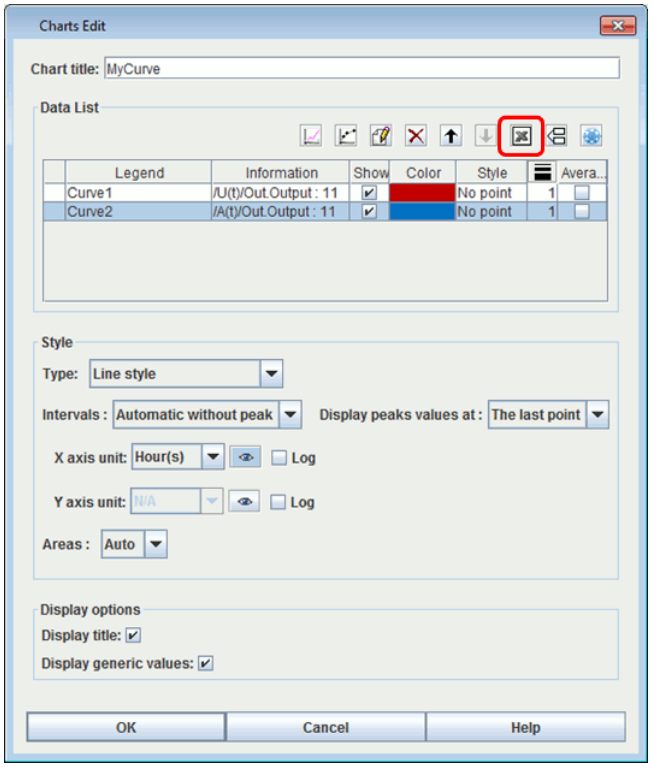


- **Save result file**: saves the contents of the Results tab (.xml format).
- **Advanced report**: generates reports using style sheets.
- **Save standard output**: saves the contents of the Info tab (.txt format).

- **Save engine data file**: saves the data file sent to the computation engine (.txt format).
- **Save as XML spreadsheet 2003 (XMLSS)**: saves all the results tables in an XML format compatible with Microsoft(r) Excel 2003 and later versions.

17.6. Curves

For each curve drawn, the points which have been computed in CSV format can be saved. This list of points can then be used to draw new curves or to perform further computations.



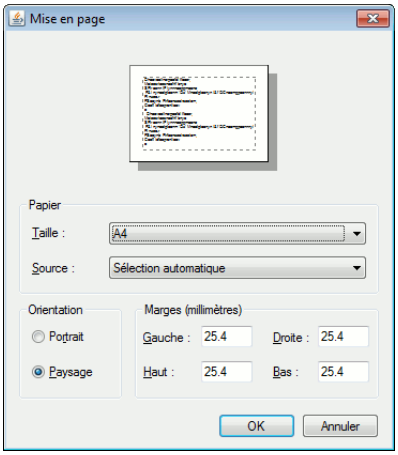
18. Printing

For printing, you have several commands at your disposal in the **File** menu **File**:

- **Page setup**
- **Print**
- **Save in RTF file**

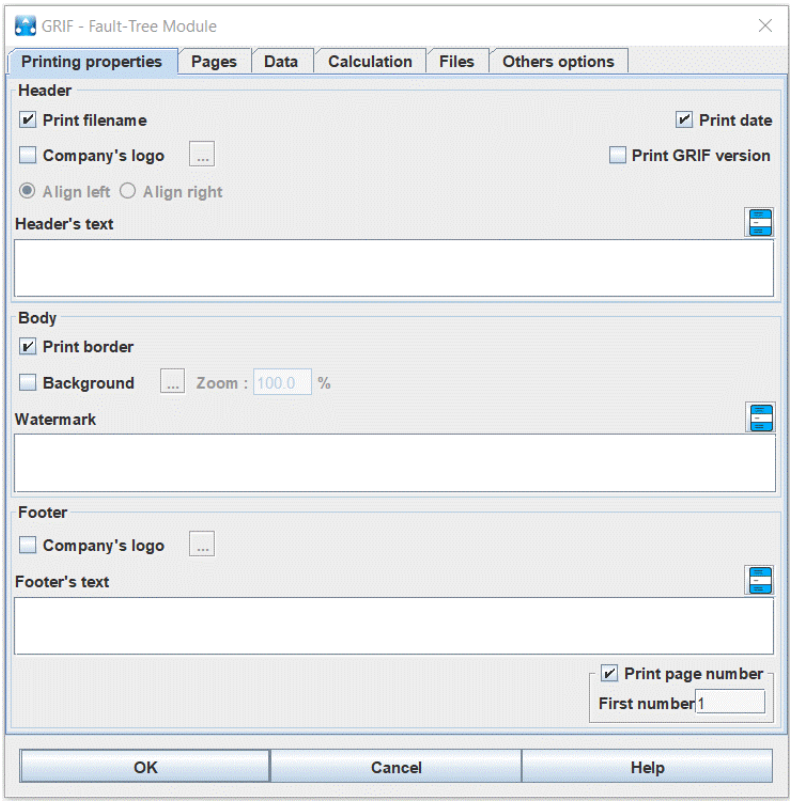
18.1. Page setup

The **Page setup** function enables you to choose the page orientation, the size of the margins, etc.



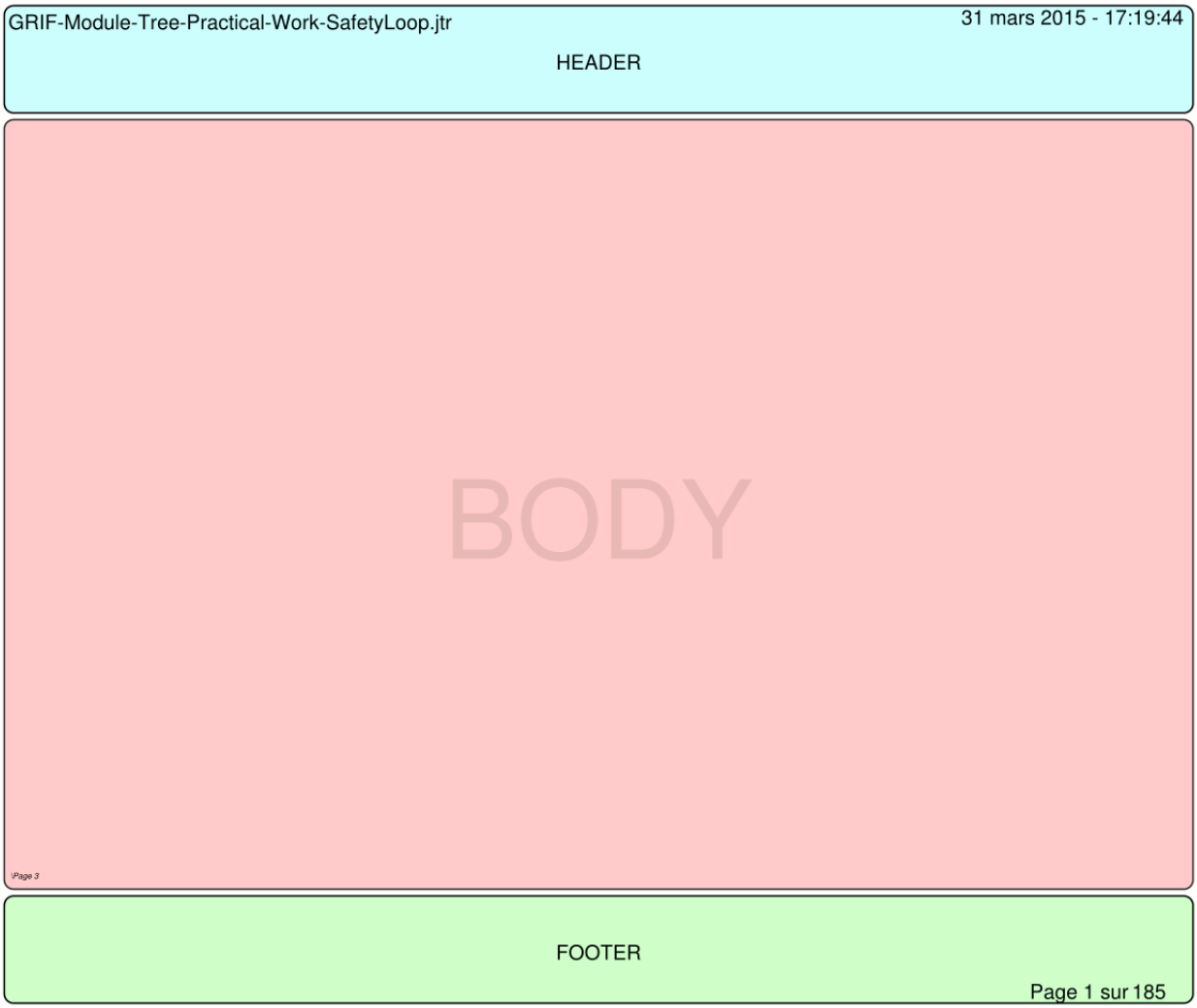
18.2. Print

The **Print** function enables you to export .pdf document pages. Graphics are exports in a vector graphics format in order to scale it without deterioration. All data table and the results of calculations can be also exported. Here is the window of configuration of the printing:

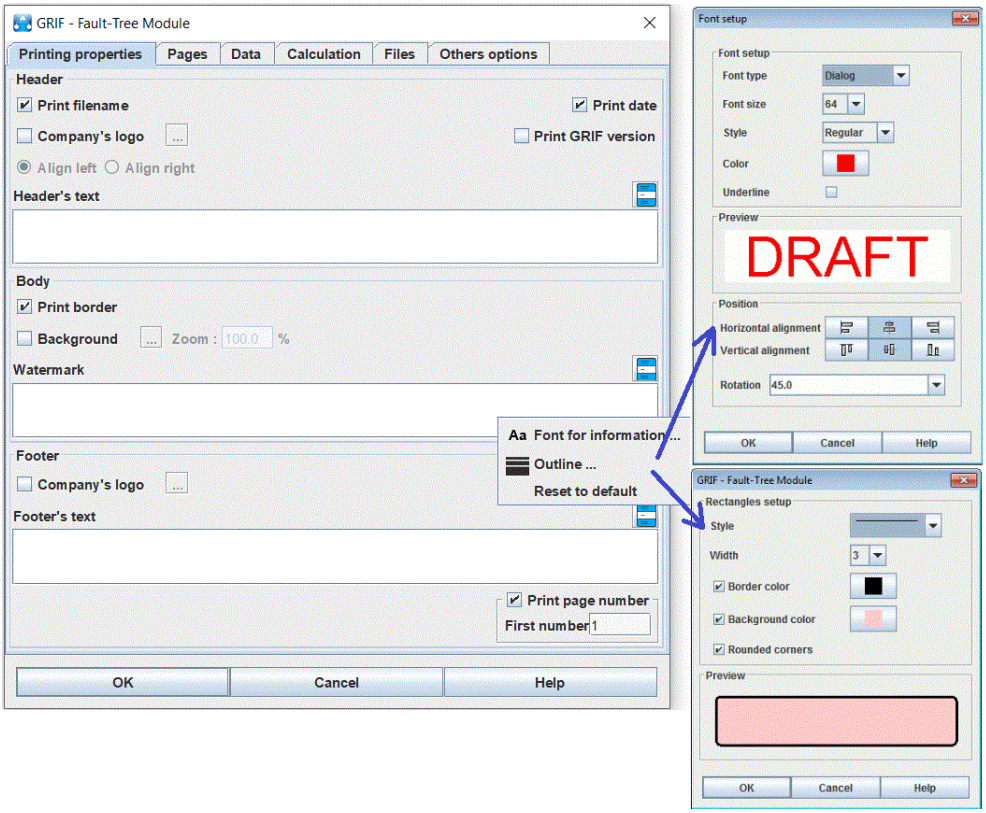


1. Printing properties

Printing properties tab gives the possibility of configuring what will be visible on all the printed pages. This tab contains three parts: the header, the body and footers. Below here is an example which illustrates the various zones on the PDF:



Each of three text fields is configurable. The user can so modify the police, the style, the color and the positioning of the text in the zone. He can also configure the zone itself by allocating a background color and a frame. A click-right on the text field shows a contextual menu: **Display settings**

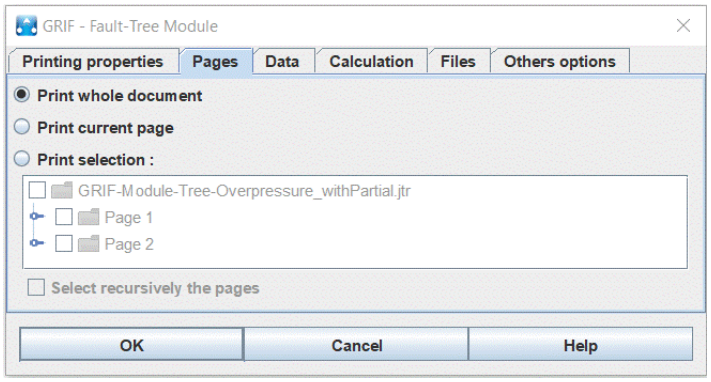


Several options are added:

- **Print file name** in the header at the left top.
- **Print date** in the header at the right top.
- **Print a border in the graphic pages** enables to bound visually the pages of graphs besides of the page of printing. Warning: this frame can be in conflict with a possible frame defines in the zone of body of page
- **Watermark** enables to seize an image to be printed on a background with colors which will be limited. The image can go out of the zone of body of page
- **Print number of page** in the footer in the lower right. The first number can be chosen to start not at 1.

2. Pages

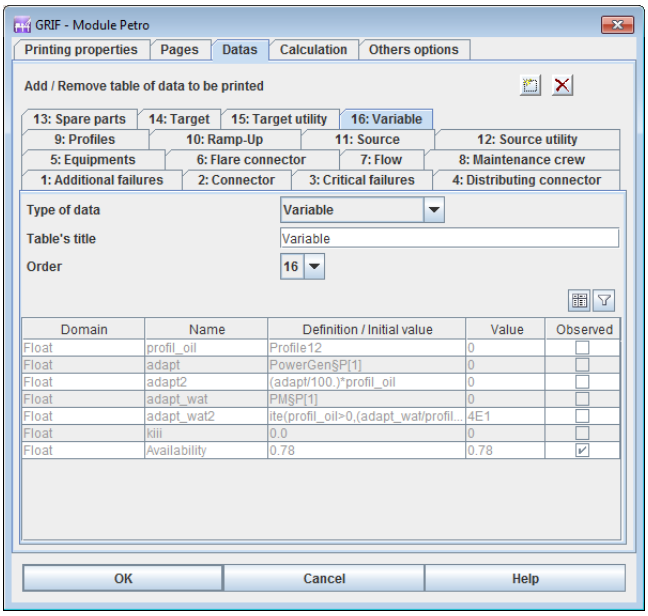
Pages tab enables to select the page will be printed.



- **Print all the page** : all the pages will be printed
- **Print current page** :only the current page will be printed.
- **Print selection** : select pages and groups will be printed. **Select recursively the pages** option is a help in order to select quickly sub-pages.

3. Data

Data tab enables to select date will be printed.

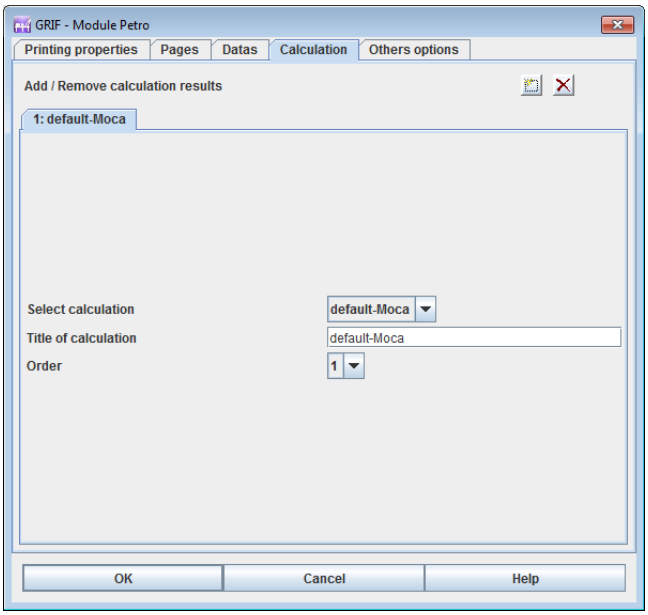


Each sub-tab will represent a table in the PDF document. The title and the print order of these tables can be modified.

By default all the types of data are represented (one by sub-tab). The upper right buttons allow to add it or to delete it. To note that we can print several times of the same data type in different tables. The data table can be filter using the usual filter. The Column manager can be also used.

4. Calculation

Calculation enables to select the calculation will be printed.

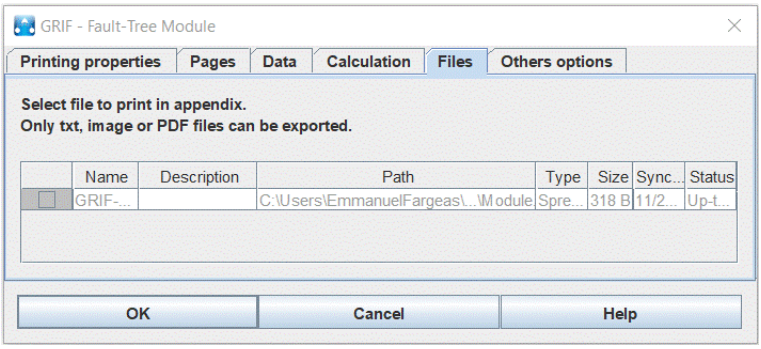


Each sub-tab will represent a part containing the various tables present in the results of the calculation. This part has a modifiable title and order which is the order of printing of the various results.

By default all the calculation results are represented (one by sub-tab). The upper right buttons allow to add to it or to delete it.

5. Files

Files enables to select the files related to the document to print in appendix. Only txt, image or PDF files can be exported.



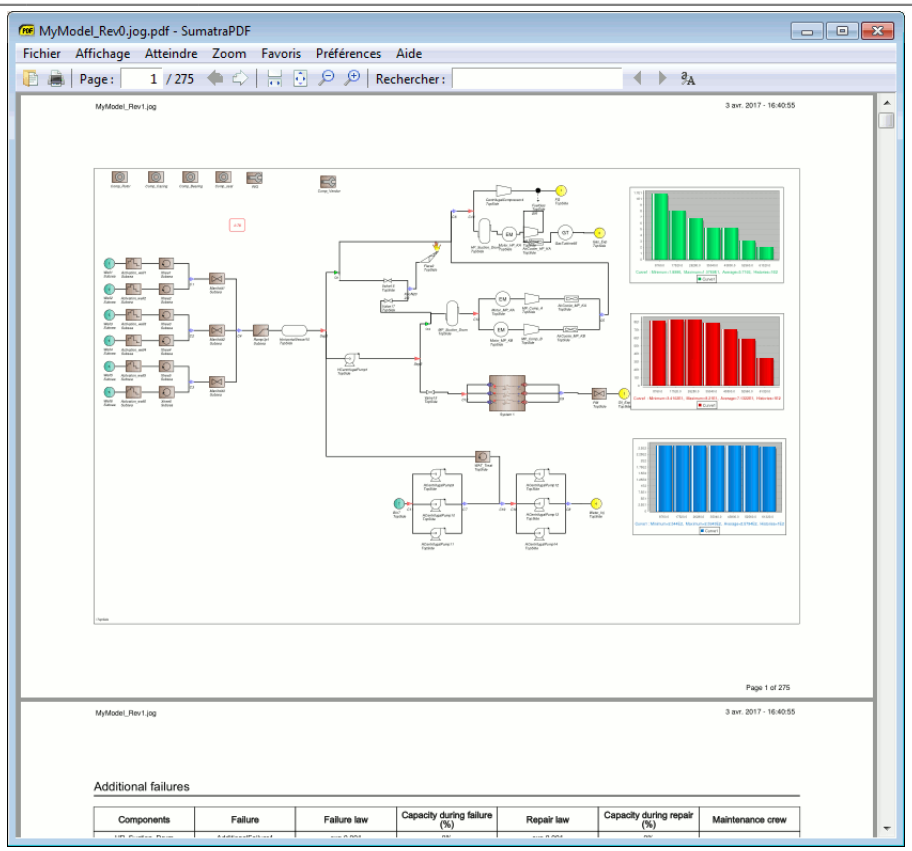
6. Others options

Others options .

- **print table of contents** : Allows to print the table of content of document, very useful for big documents.
- **Print modification tracking** : Allows, if the modification tracking optuion is selected, to print comment, author and date of modifications for each saving.
- **Print index of graphic pages** : Allows to print an index of graphic pages, very useful for big documents.
-
-
-
-
-

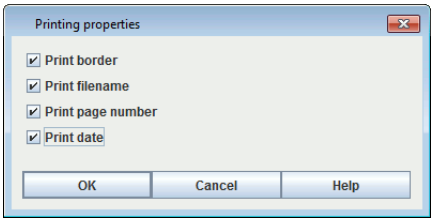
7. Example

To give an overview of a PDF printing:

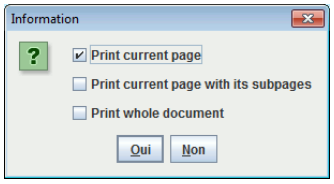


18.3. Save in RTF file

The **Save in RTF file...** function initially gives access to a window called **Printing properties** . Then to another called **Information** . And thirdly, a window is displayed allowing you to choose the folder in which the RTF file is to be saved.



When you select the **Save in RTF file** function, the first box to appear is that shown above. You can then select your preference: Print border, Print filename, Print page number and/or Print date.



Secondly, an **Information** window appears. It enables you to indicate whether you wish to print the current view, print the current page or print the whole document.

19. Options of GRIF - Petro

Tools - Application Options menu opens a window containing the following tabs:

19.1. Options

Options tab enables to tune application behavior :

- **Save the options of the current document as default options in the application** : Save options of current doc as application default options.
- **The application manages the default options of the documents, apply the default options to the current document** : Apply -Application options- to current document.
- **Delay of automatic document saving (in minutes)** : Delay of automatic document saving (in minutes). A null value disables automatic saving.
- **Number of undo** : Specifies number of possible undo/redo.
- **Number of recent files** : Specifies number of files in recent files list.
- **Window display** : Enables separate tables (external) or linked tables (internal).
- **Columns to be resized in tables** : Enables to specify the columns on which space will be taken for resizing.
- **Ask for confirmation before deletion outside the input area** : When deleting an element in the graphic tree or in the table date, a dialog box will be displayed.
- **Manage new names to avoid name conflict** : Tries to avoid name conflict, creating new objects whose name is unique (when pasting for example).
- **Synchronize view with tables** : Select objects in tables (on the right) when they are selected in view.
- **Synchronize view with explorer** : Select objects in explorer (on the left) when they are selected in view.
- **Ask for confirmation if closing with close button** : When closing with the button at the top-right of a dialog box, the software will ask for a confirmation. Use OK or CANCEL buttons if you don't want to confirm closing.
- **Modification tracking when saving** : When saving, if tacking is activated, you can add a comment about modifications made on the document.
- **Ask for configuration of observed variables** : Ask for computation setup when variable becomes observed.

19.2. Executables

Executables tab enables to specify path to external executables :

- **Mail client** : Enable you to set the mail client to use
- **Automatically open PDF files** : Specifies if PDF reports must be opened with generation.
- **Moca-RPC path** : Specifies path of Moca 10.

19.3. Graphics

Graphics tab enables to modify GUI look :

- **Use Windows look and feel** : Use the look and feel of your operating system instead of java look and feel (GRIF restart is needed).
- **Element Zoom** : Changes graphics size.
- **Filling and outline for dynamic fields** : Object outline configuration (line color, line width, background color, ...).
- **Font for dynamic fields** : Enables font configuration (color, size, italic ...) for information that are displayed under objects.
- **Shape filling and outline for commentaries** : Object outline configuration (line color, line width, background color, ...).
- **Font for commentaries** : Enables font configuration (color, size, italic ...) for information that are displayed under objects.
- **Shape filling and outline for groups** : Object outline configuration (line color, line width, background color, ...).
- **Shape filling and outline for utility groups** : Object outline configuration (line color, line width, background color, ...).

- **Font for groups** : Enables font configuration (color, size, italic ...) for information that are displayed under objects.
- **Activate cross hair** : Activate cross hair which enables object alignment.
- **Activate smoothing for texts** : Activate anti-aliasing (smoothing) for texts, it can slow the display.
- **Activate smoothing for images** : Activate anti-aliasing (smoothing) for images, it can slow the display.
- **Activate tooltips** : Activate tooltip-system.
- **Display the name of the observed links** : Enables to display name or not

19.4. Digital format

Digital format tab enables to customize digits display :

- **Display of parameters** : Specifies the display of parameters (number of digits, ...).

19.5. Computations / Results

Computations / Results :

- **Light Batch** : Deletes files used for each computation of batch computations, it decreases memory/disk use.
- **Preferred frequency unit** : Unit that will be used for displaying result which dimension is "frequency" in - main view, - data tables, - and some result synthesis.If no unit is displayed (especially in detailed results) the unit is (h-1).
- **Preferred duration unit** : Unit that will be used for displaying result which dimension is "duration" in - main view, - data tables, - and some result synthesis.If no unit is displayed (especially in detailed results) the unit is (h).

19.6. Blocks

Blocks :

- **Font set-up** : Enables font configuration (color, size, italic ...) for information that are displayed under objects.
- **Display name** : Enables to display name or not
- **Display number** : Enables to display number or not.
- **Display description** : Enables to display description or not.
- **Outline set-up** : Object outline configuration (line color, line width, background color, ...).
- **Display SIMOPS** : Display or not display the SIMOPS components list (those impacted by operations on the component)
- **of names** : of names

19.7. Connectors

Connectors tab contains options for connectors. :

- **Font set-up** : Enables font configuration (color, size, italic ...) for information that are displayed under objects.
- **Outline set-up** : Object outline configuration (line color, line width, background color, ...).
- **Display connector name** : Enable to display connector name.
- **Display K/N** : Enables to display K/N or not.
- **Display output connector name** : Enable to display output connector name.
- **Display name of report source** : Enable to display name of report source.
- **Display page of report source** : Enable to display page of report source.

19.8. Simulation - settings

Options settings interactive simulation :

- **Automatic firing of dirac 0** : Automatic firing of transition with Dirac 0 law.
- **Automatic firing when simulation starts** : Automatic firing of transition when simulation starts.
- **Transitions fireable before loop detection** : Specifies maximum number of fire before loop detection.
- **Limit duration for "fast forward" (s)** : Specifies the number of seconds before the detection of a loop during the "fast forward".

- **Graphical Rendering of simulation** : Activate or not graphical rendering of simulation or each modification.
- **Keep history during "fast forward"** : It enables history-saving of triggered transitions, the ones automatically triggered when using Go until time/transition functions
- **Constant time interval between two automatic fires** : When using automatic forward during step by step simulation, the (user) time between two transition fires will be constant. If this option is not checked, time will be proportional to the simulated time.
- **Delay between transition fire (ms)** : Delay between two automatic fire.
- **Simulated time / user time coefficient** : Delay between two automatic fire.
- **Simulation trace** : Enable simulation trace.
- **Path to trace file** : File where simulation is saved.
- **Follow scheduler order for deterministic transitions** : Only the first deterministic transition of scheduler can be fired.
- **Choose delay of fired transitions** : Displays a dialog box to choose delay of fired transition.

19.9. Simulation - display

Display options for interactive simulation :

- **Show table K (real flow) on flow tooltip** : Show table K (real flow) on flow tooltip
- **Show table P (potential flow) on flow tooltip** : Show table P (potential flow) on flow tooltip
- **Show table N (nominal flow) on flow tooltip** : Show table N (nominal flow) on flow tooltip
- **Show table Dr (real request) on flow tooltip** : Show table Dr (real request) on flow tooltip
- **Show Dp value (potential request) on flow tooltip** : Show Dp value (potential request) on flow tooltip
- **View Ak value (current capacity) on component tooltip** : View Ak value (current capacity) on component tooltip
- **View Pk value (potential capacity) on component tooltip** : View Pk value (potential capacity) on component tooltip
- **View Nk value (nominal capacity) on component tooltip** : View Nk value (nominal capacity) on component tooltip
- **View the states of the components on component tooltip** : View the states of the components on component tooltip
- **Show start condition on component tooltip** : Show start condition on component tooltip
- **Show stop condition on component tooltip** : Show stop condition on component tooltip
- **Show utilities condition on component tooltip** : Show utilities condition on component tooltip
- **Show repair condition on component tooltip** : Show repair condition on component tooltip
- **Show references values of profiles on tooltips** : Show references values of profiles on tooltips
- **Display the values of observed flows on the outputs** : Display the values of observed flows on the outputs
- **Display the values flows on the tapping points** : Display the values flows on the tapping points

19.10. Curves

Charts tab enables to change charts drawing :

- **Set graphics borders** : Add borders to charts.
- **Set generic values borders** : Add borders to generic values under charts.
- **Display grid** : Display grid on curves area.
- **Display legends** : Display legends under curves.
- **Drawing zone transparency** : Activate curves area transparency.
- **Graphic transparency** : Activate charts transparency.
- **Title size** : Specifies charts title font size.
- **Generic values size** : Specifies generic values font size.
- **Point size** : Specifies point size on curves.
- **Coordinates size** : Specifies coordinates font size.
- **Legend size** : Specifies legends font size.

19.11. Prototypes

Path configuration prototypes :

- **Empty bloc Petri path** : Empty bloc Petri path
- **Rotating and static machines Petri path** : Rotating and static machines Petri path
- **Maintenance crews Petri path** : Maintenance crews Petri path
- **Spare parts Petri path** : Spare parts Petri path
- **CCF Petri path** : CCF Petri path
- **Flare Petri path** : Flare Petri path
- **Ramp-up Petri path** : Ramp-up Petri path
- **Statistic Petri path** : Statistic Petri path
- **Tank Petri path** : Tank Petri path
- **Path of timer prototype** : Path of timer prototype

19.12. Petri export

Export options in Petri net :

- **Precision of crc supervisors** : Precision of crc supervisors
- **Tolerance for production interval** : Adds an epsilon in interval definitions, in order to include 49.99999% production level in [50%,60%[interval
- **Standard tolerance to redirect the flow to the flare** : Standard tolerance to redirect the flow to the flare (= P - min(delta, 0.01 * P))
- **Create a supervisor for connector flare** : Create a supervisor for connector flare
- **Optimize export for serial blocks** : Optimize export for serial blocks
- **Turn flow tables into variables** : In exported PetriNet, each flow-table will be replaced by as much variables as defined flow.
- **Export dynamic fields for K table** : Export dynamic fields for K table
- **Export dynamic fields for P table** : Export dynamic fields for P table
- **Export dynamic fields for N table** : Export dynamic fields for N table
- **Export dynamic fields for Dr table** : Export dynamic fields for Dr table
- **Export dynamic fields for DP variable** : Export dynamic fields for DP variable
- **Export dynamic fields for Ak variable** : Export dynamic fields for Ak variable
- **Export dynamic fields for state variable** : Export dynamic fields for state variable
- **Export dynamic fields on inputs** : Specifies whether dynamic fields (K, Dr, Ak) of inputs must be exported.
- **Export dynamic fields on outputs** : Specifies whether dynamic fields (K, Dr, Ak) of outputs must be exported.
- **Export dynamic fields on components** : Specifies whether dynamic fields (K, Dr, Ak) of components must be exported.
- **Export dynamic fields on connectors** : Specifies whether dynamic fields (K, Dr, Ak) of connectors must be exported.

19.13. Verifications

Verifications :

- **Maximal threshold for lambdas** : Allow the verification of the lambda in the application
- **Null transition rate produces a warning** : Displays a warning message if a transition rate is equal to 0.
- **Minimal threshold for MTTR (0.0 authorized)** : Allow the verification of the MTTR in the application

19.14. Cost management

Allow the configuration of the cost calculation :

- **Activate setting and cost calculation** : If checked, setting and cost calculation will be available
- **Activate detailed cost calculation** : If checked, the costs will be detailed by component.
- **Currency** : Currency used in the to document
- **Equipment cost** : Type of equipment cost
- **Cost of maintenance crews (excluding repairs)** : Type of maintenance crews costs related to mobilization and waiting for repairs
- **Cost of maintenance crews (under repair)** : Type of cost of maintenance crews during repairs
- **Cost per unit of spare parts in the initial stock** : Type of Cost per unit of spare parts in the initial stock
- **Cost per unit of spare parts during replenishment** : Type of cost per unit of spare parts during replenishment

- **Cost of Preventive maintenance 1** : Type of cost for the preventive maintenance that are Type 1
- **Cost of Preventive maintenance 2** : Type of cost for the preventive maintenance that are Type 2
- **Cost of Preventive maintenance 3** : Type of cost for the preventive maintenance that are Type 3
- **Cost of Preventive maintenance 4** : Type of cost for the preventive maintenance that are Type 4
- **Cost of Preventive maintenance 5** : Type of cost for the preventive maintenance that are Type 5
- **Currency discount rate** : Annualized currency discount rate
- **Inflation rate** : Annualized inflation rate
- **Governmental tax rate** : Governmental tax rate
- **Capex recovery rate** : Capex recovery rate