Getting started with ProSimPlus®

Use Case 1: Main features overview

Software & Services In Process Simulation

We guide You to efficiency

ProSim

Introduction

ProSimPlus is a process engineering software that performs rigorous mass and energy balance calculations for a wide range of industrial steady-state processes. It is used in design as well as in operation of existing plants for process optimization, units troubleshooting or debottlenecking, plants revamping or performing front-end engineering analysis.

This document gives a general overview of ProSimPlus features. It is based on the "*Simple example*", available on ProSim web site: www.prosim.net.

The steps to create and analyze the flowsheet are the following:

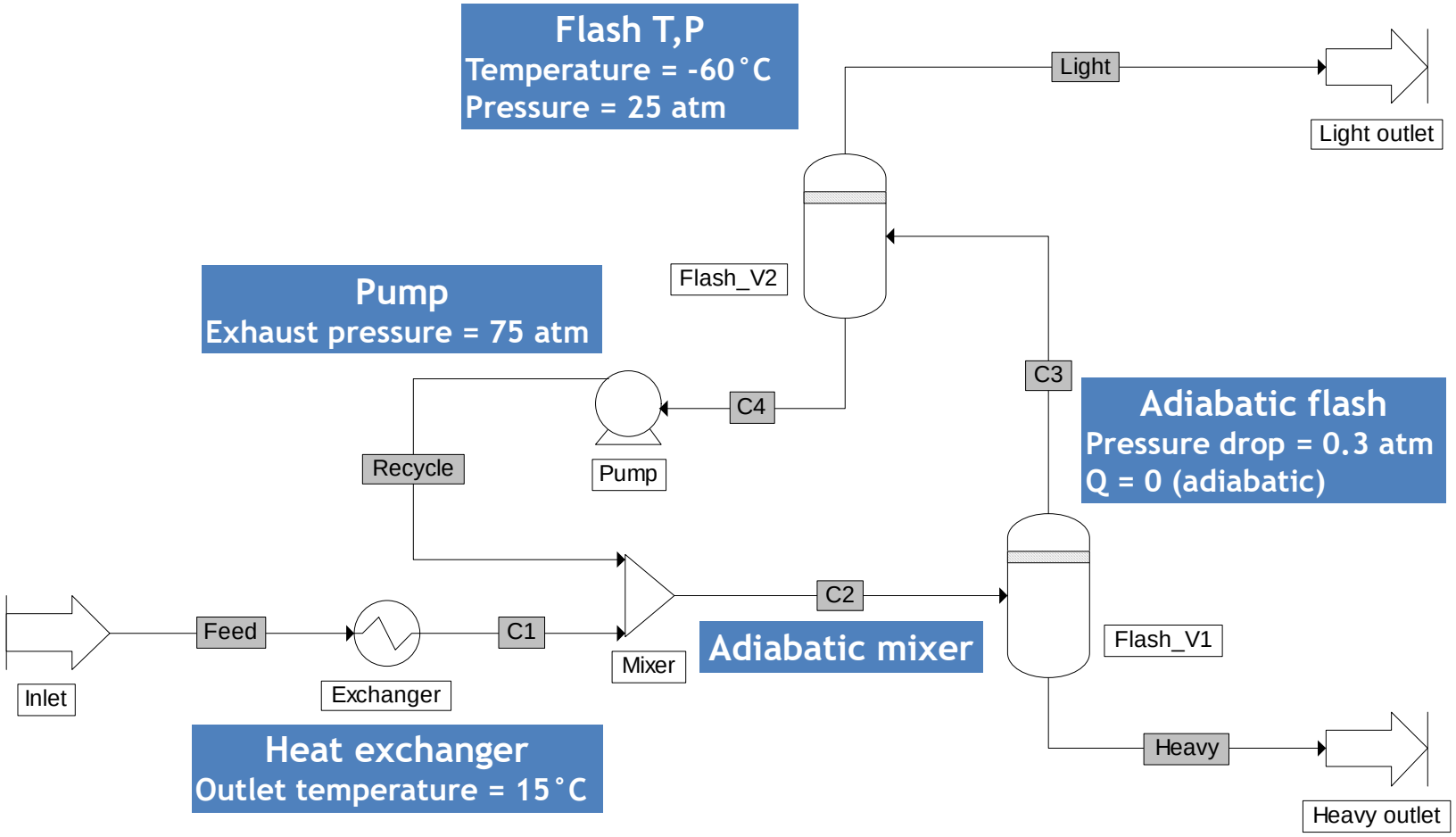
- Step 1 – Select the components
- Step 2 – Select the thermodynamic model
- Step 3 – Create the flowsheet
- Step 4 – Run the simulation
- Step 5 – Analyze generated reports
- Step 6 – Analyze results from the flowsheet
- Step 7 – Graphical User Interface

Presentation of the example

The “simple example” consists in separating a hydrocarbons mixture into heavy and light components:

Process Feed
Temperature = 40°C
Pressure = 75 atm

Partial molar flow rates (kmol/h)
- Nitrogen : 9.0
- Methane : 41.7
- Ethane : 11.2
- Propane : 6.2
- n-Butane : 5.4
- n-Pentane: 3.0
- n-Hexane : 8.1
- n-Heptane: 13.3
- n-Octane : 2.1



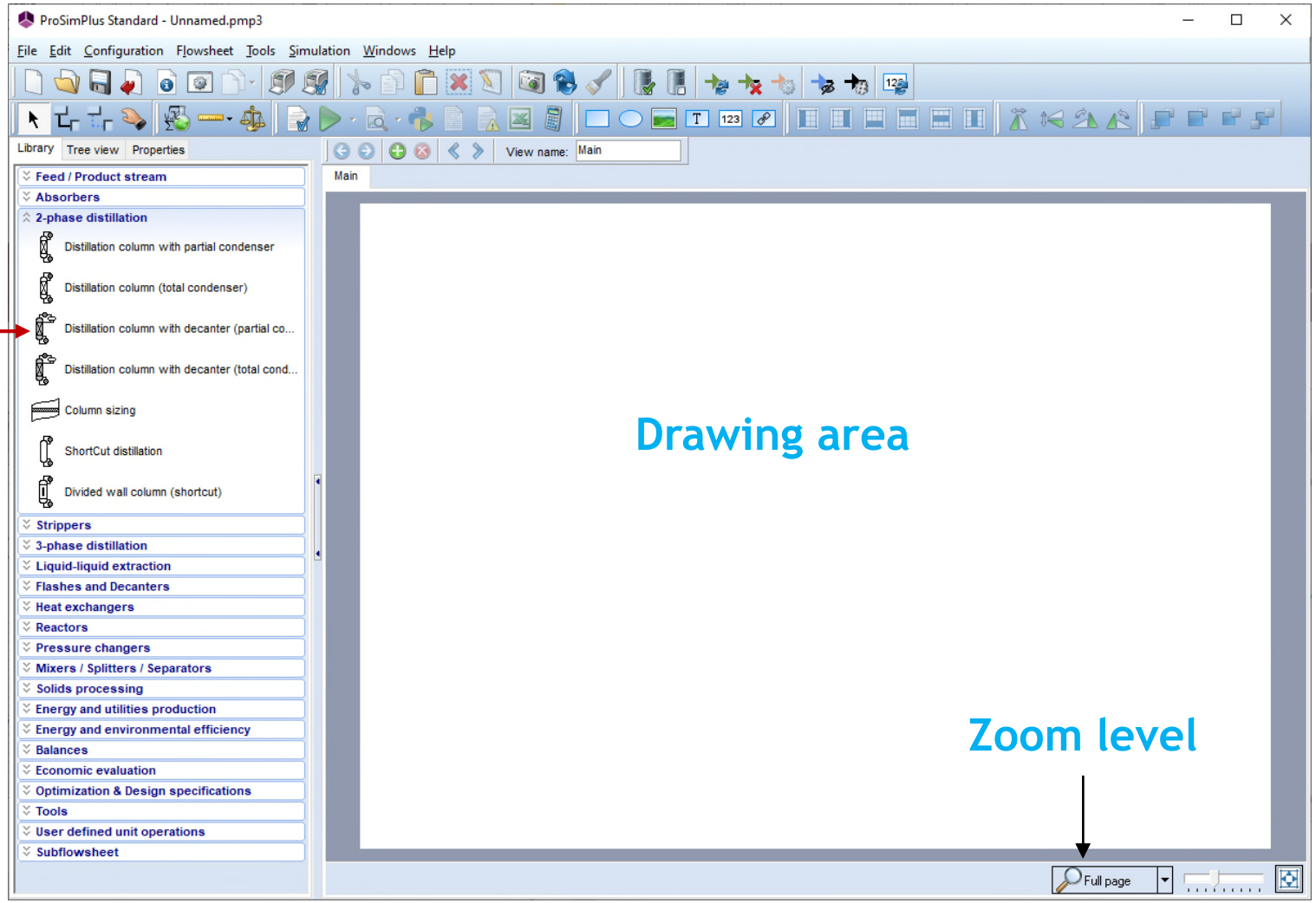
Before we start: Graphical User Interface

Menu

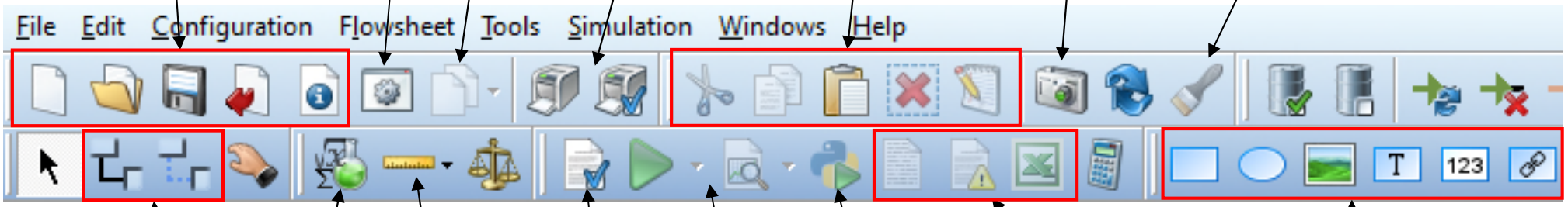
Toolbars

Unit operations within the selected categories

Library of unit operations



Before we start: Graphical User Interface



File management (new, open, save, close, edit synopsis)

ProSimPlus Preferences

Open the files linked to the simulation

Setup the drawing area, Print

Object management (cut, copy, paste, delete, open configuration window)

Copy the flowsheet (to paste in another document)

Copy the format

Streams (material, information)

Thermodynamic profile(s) (components, models, binary interaction parameters…) and Set(s) of reactions

Unit systems (unit operations description and reports)

Simulation report parameters

Run the simulation (with options) and Case study

Call to an external solver

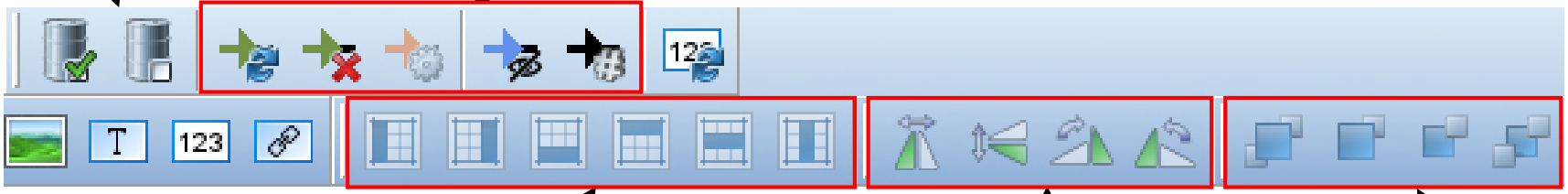
Last simulation results (report, history file, MS-Excel workbook)

Graphical objects (text, image, shapes, tags, links)

Before we start: Graphical User Interface

TBP/ASTM Assay curves of material streams (select, deselect)

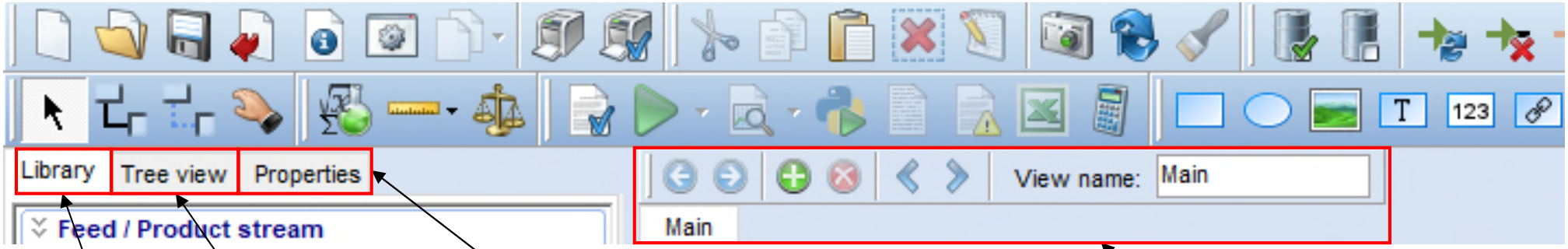
Streams management (link, initialization, number…)



Arrange the graphical objects on the flowsheet (align, center…)

Objects position (mirror, inverse, rotate…)

Arrange the objects layout in the flowsheet (front, back…)



Access to the library of unit operations

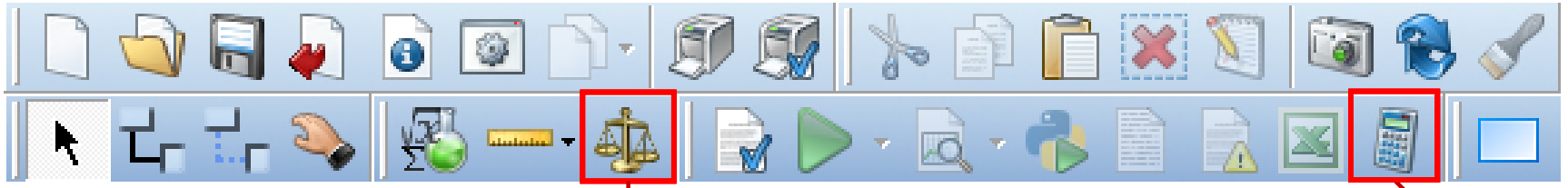
Access to the existing object on the flowsheet

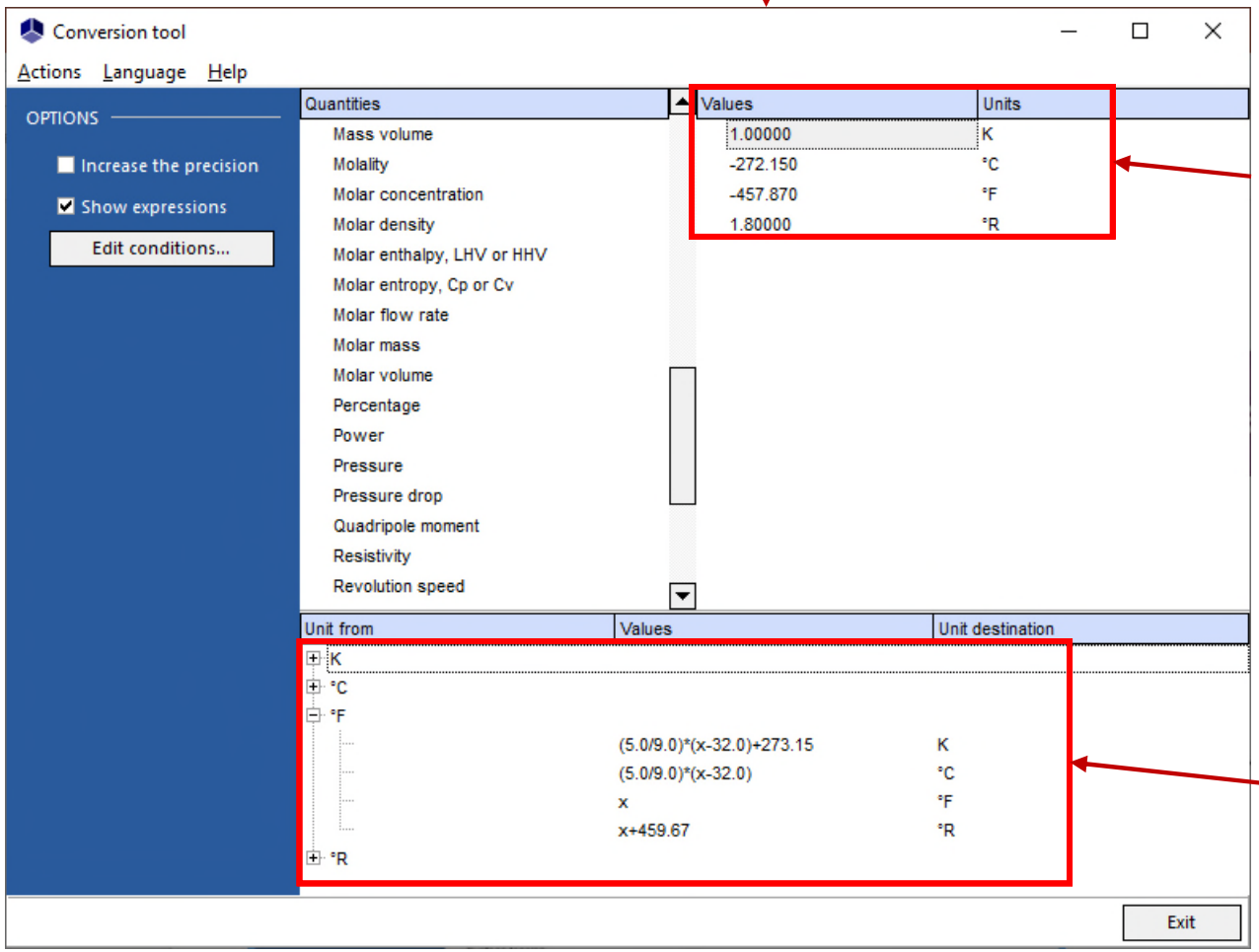
Access to the graphical properties and the name of the selected object

Management of the views of the flowsheet (creating, moving, renaming…)

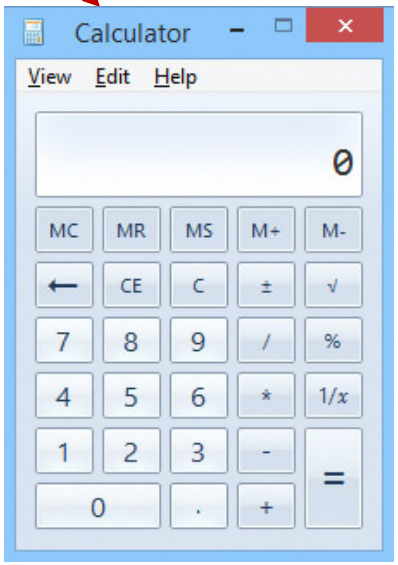
Before we start: Graphical User Interface

Tools: conversion tool and calculator





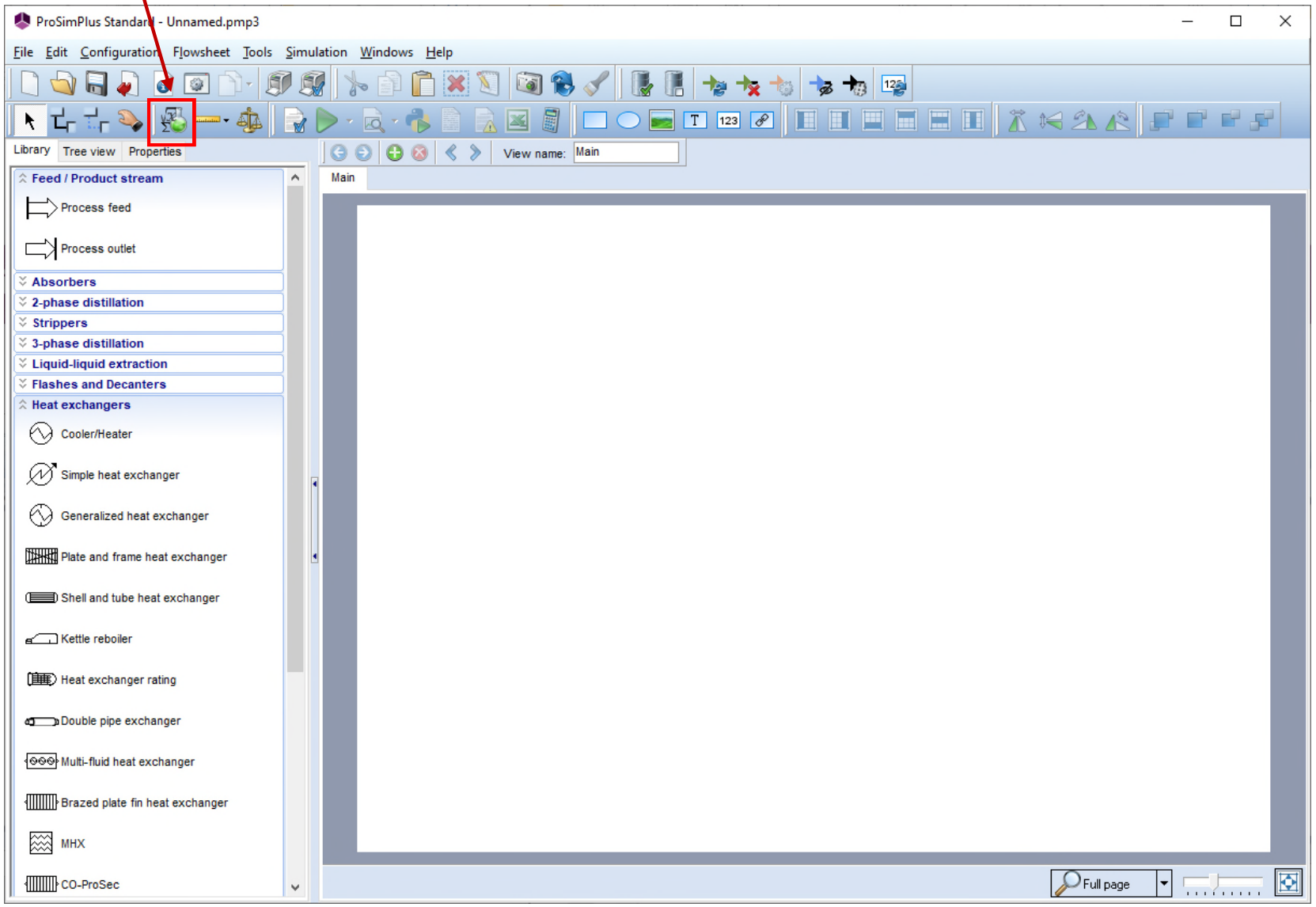
Conversions



Mathematical expressions used by the conversion tool

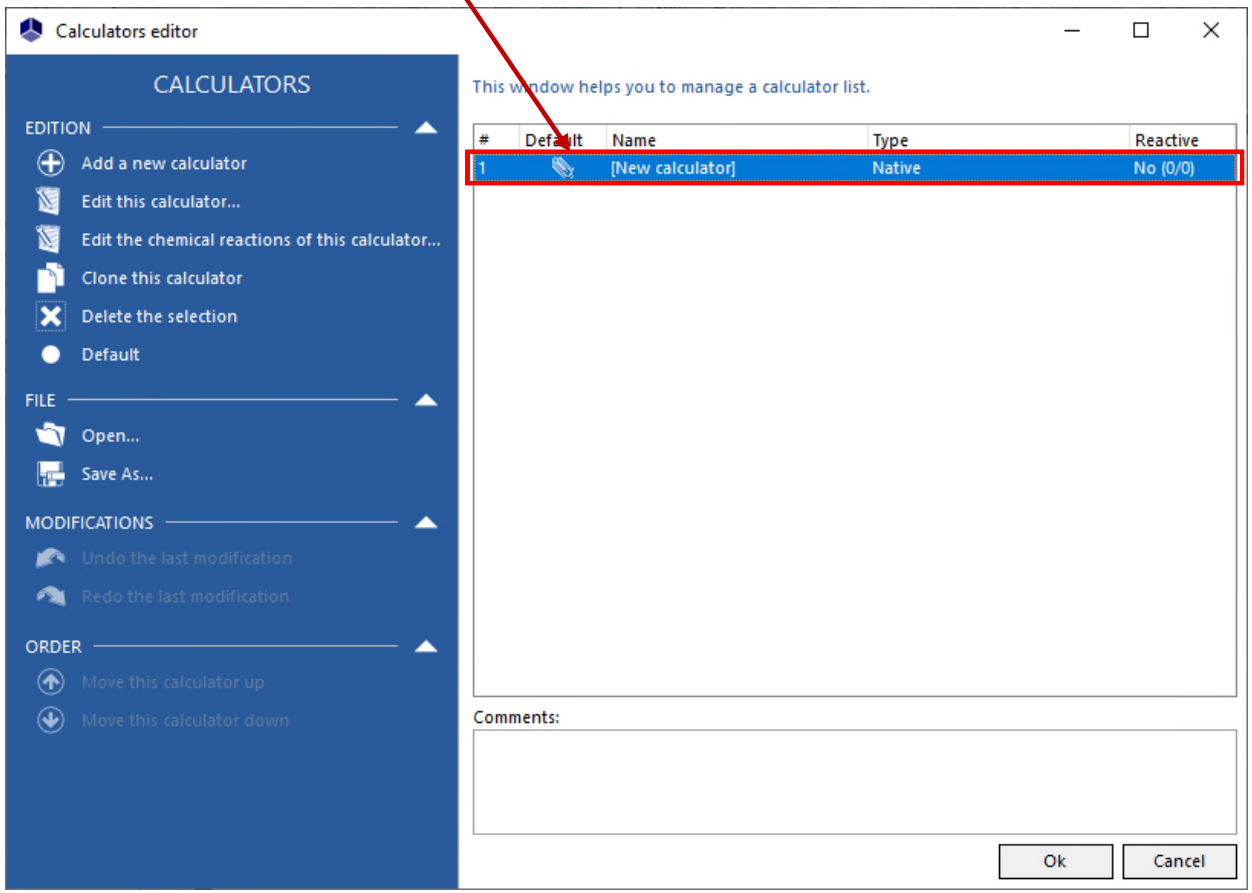
STEP #1: Select your components

Click on the “Thermodynamics and Compounds” icon



STEP #1: Select your components

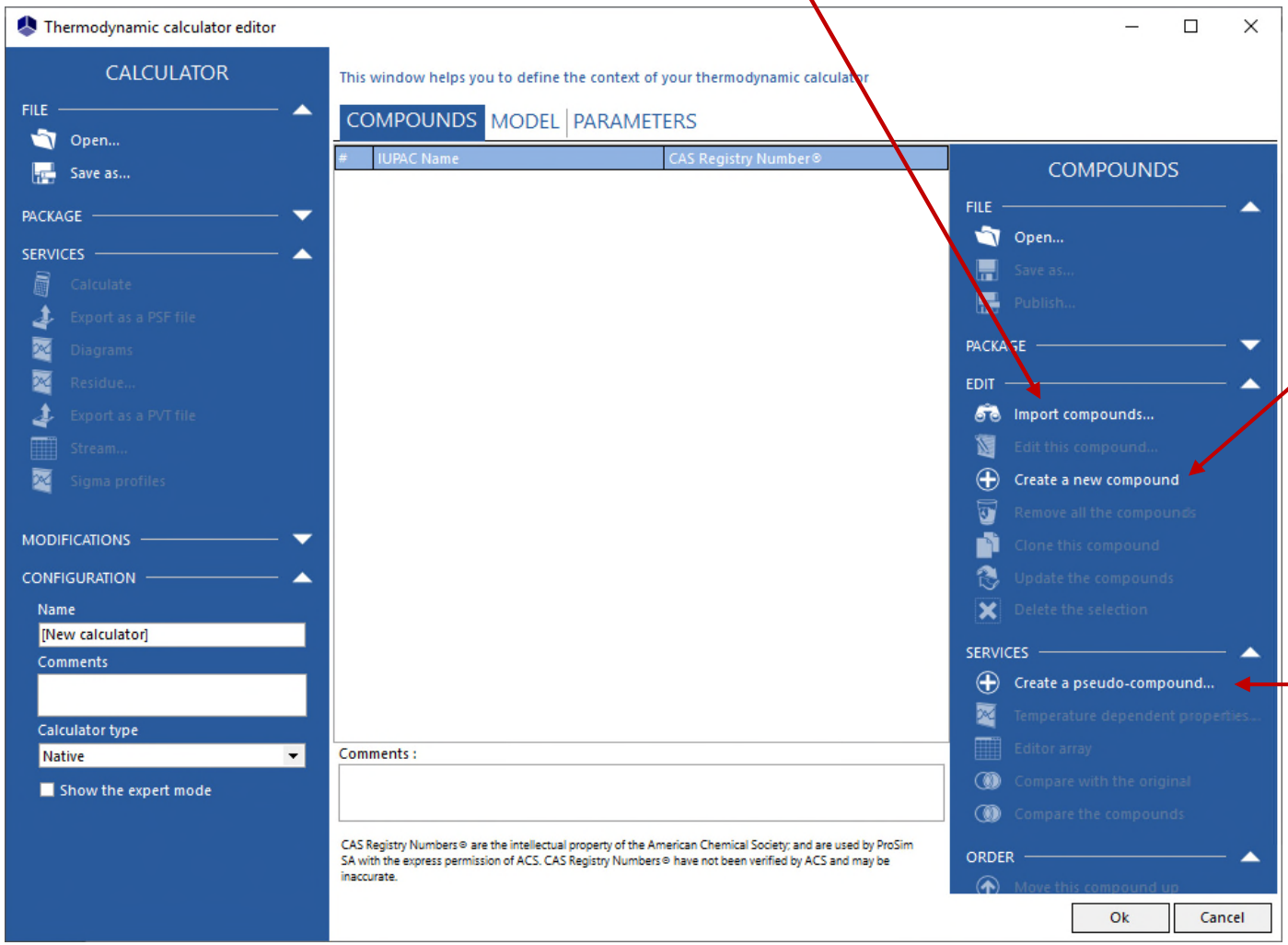
Double click on the calculator to open its configuration window



For more information about ProSimPlus thermodynamic package (called Simulis Thermodynamics), please consult « Getting Started with Simulis Thermodynamics » documents

STEP #1: Select your components

Click on “Select compounds” to import the compounds from the databases available on your computer



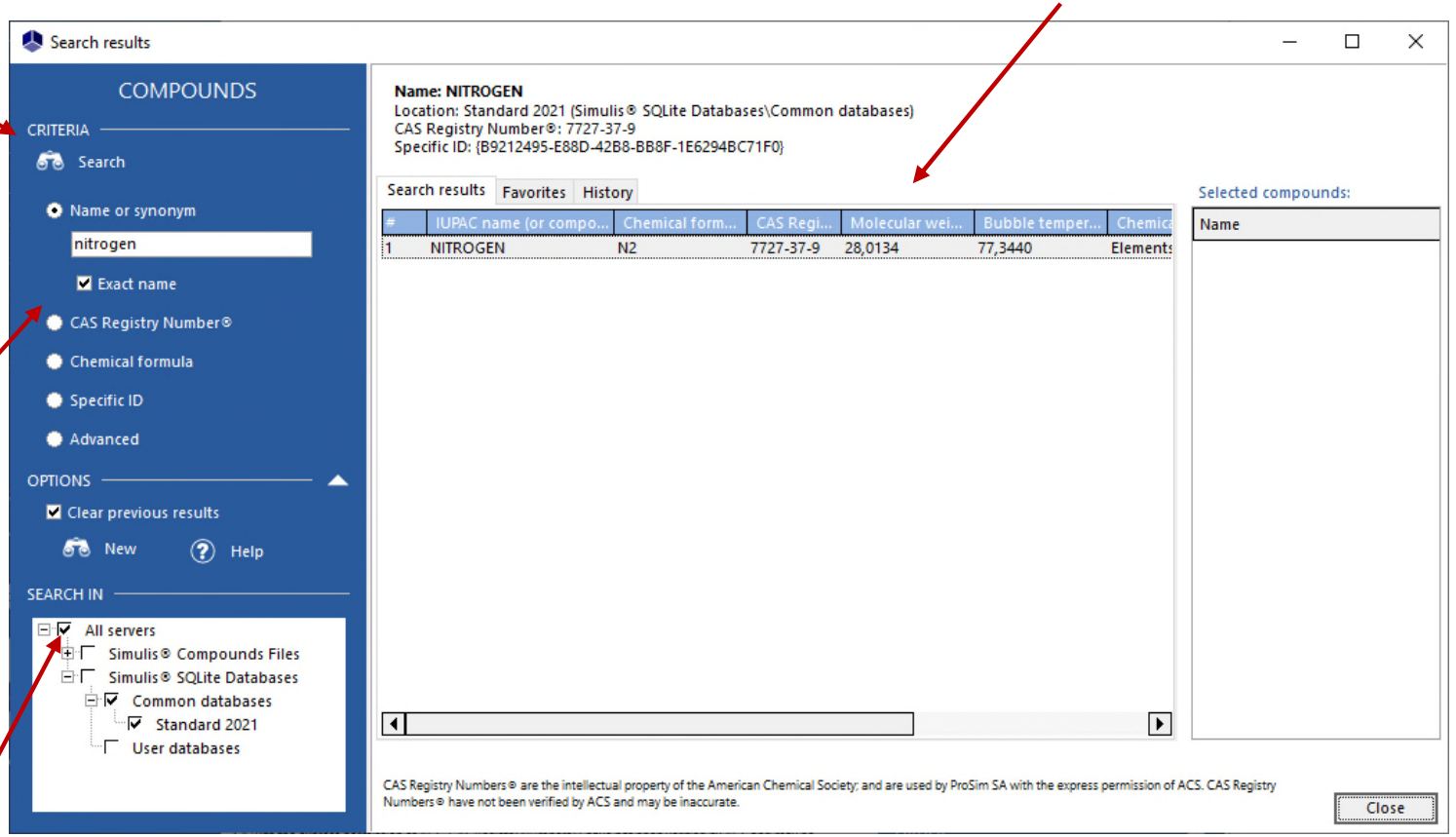
You can create your own compounds, by clicking on “Add a new compound”

You can create pseudo-compounds corresponding to a petroleum cut, by clicking on “Create a pseudo-compound”

STEP #1: Select your components

3. Click on the “Search” button to get the list of compounds that match your criteria

4. The search results are shown in this area



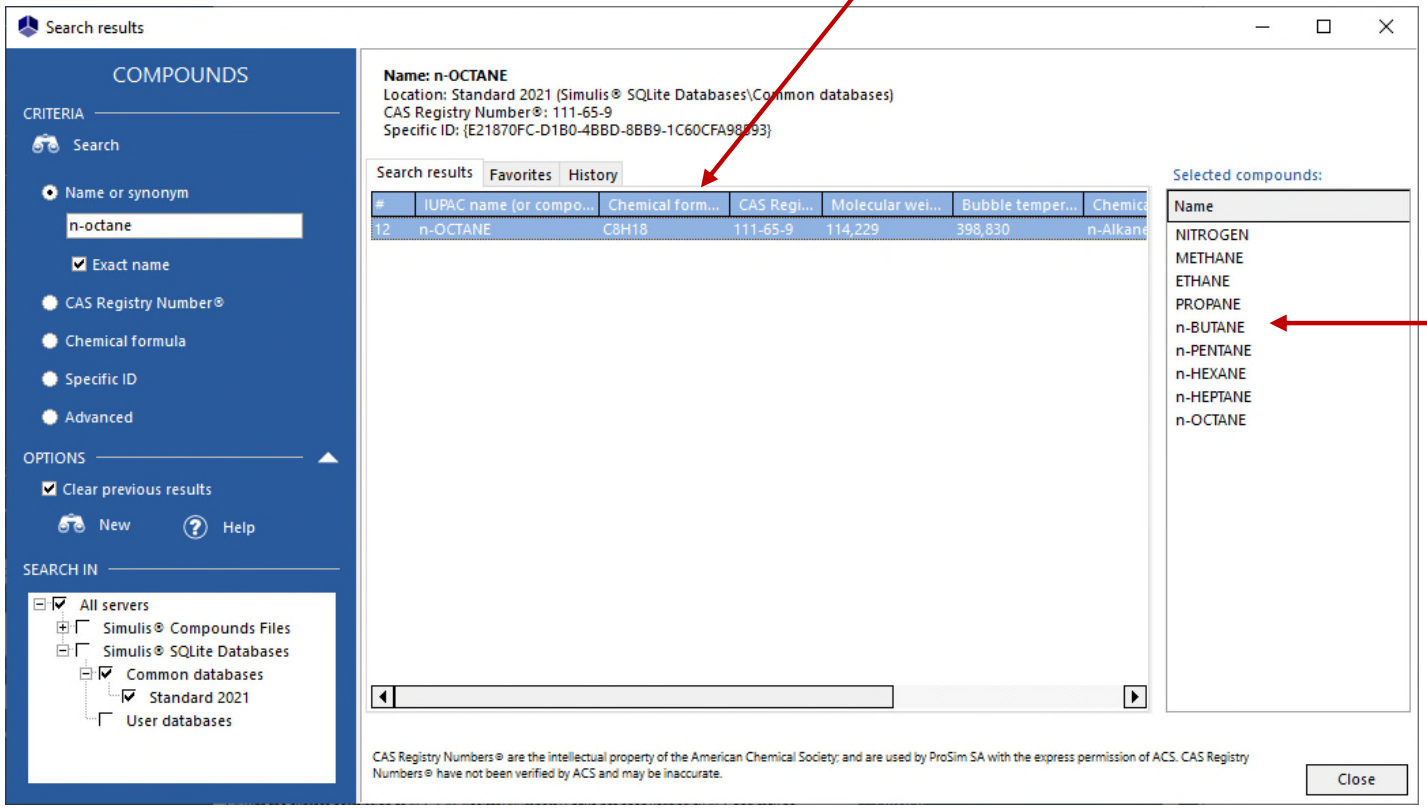
2. You have access to multiple search criteria (in this example, search “Nitrogen” by name)

1. Select the compounds server(s) (databases or packages) in which you want to search the compounds

You can run multiple searches without closing this window

STEP #1: Select your components

1. Double click to add the compound (Nitrogen) to your final selection



2. Repeat this operation for the other compounds (Methane, Ethane, Propane, n-Butane, n-Pentane, n-Hexane, n-Heptane, n-Octane)

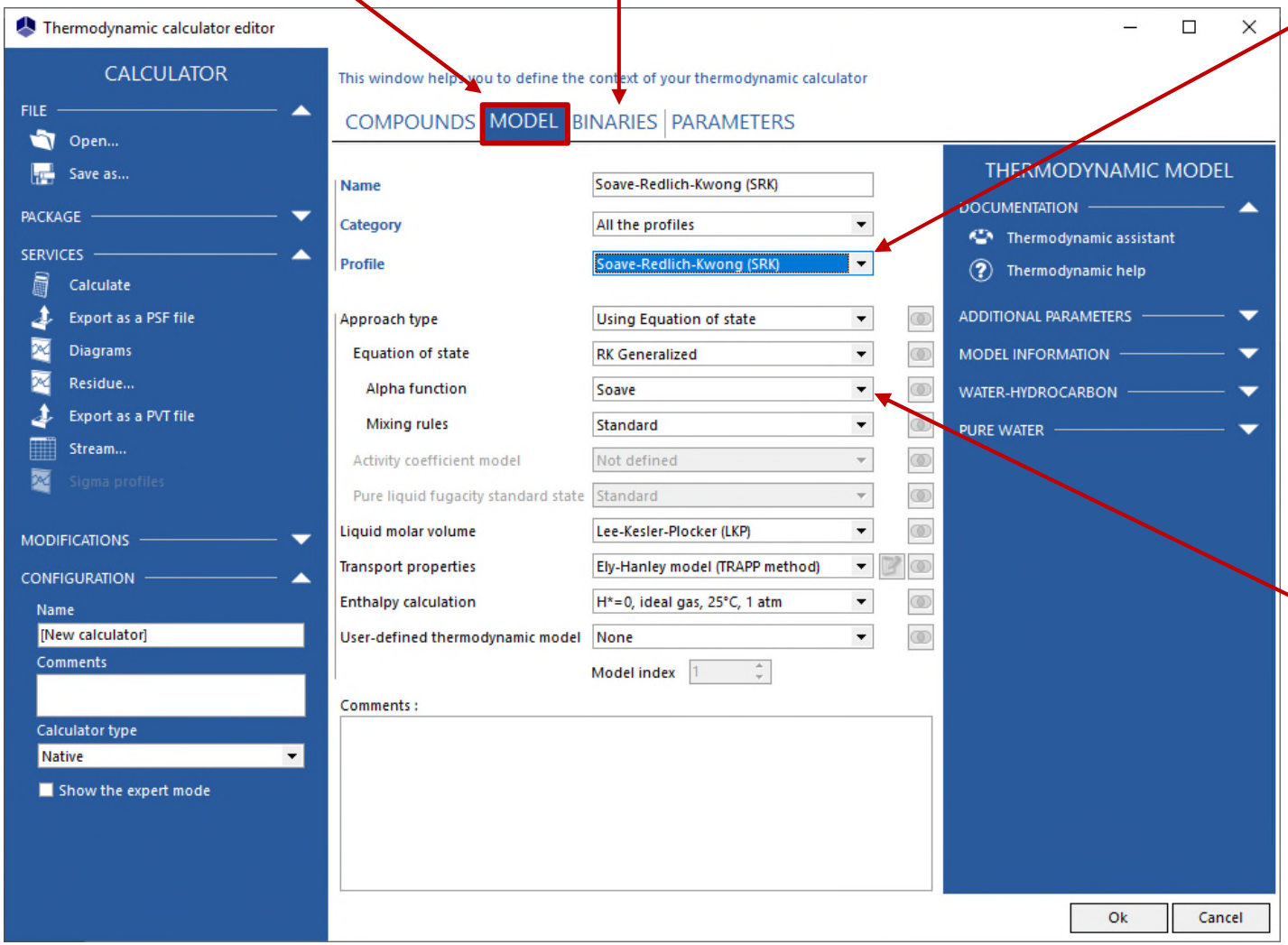
3. Click on "Close" to end the compounds selection process

STEP #2: Select your thermodynamic model

1. Click on the "Model" tab to enter the thermodynamic models editor

The "Binaries" tab appears automatically when you select a model that requires binary interaction parameters

2. Select the Thermodynamic profile (in this example, SRK)

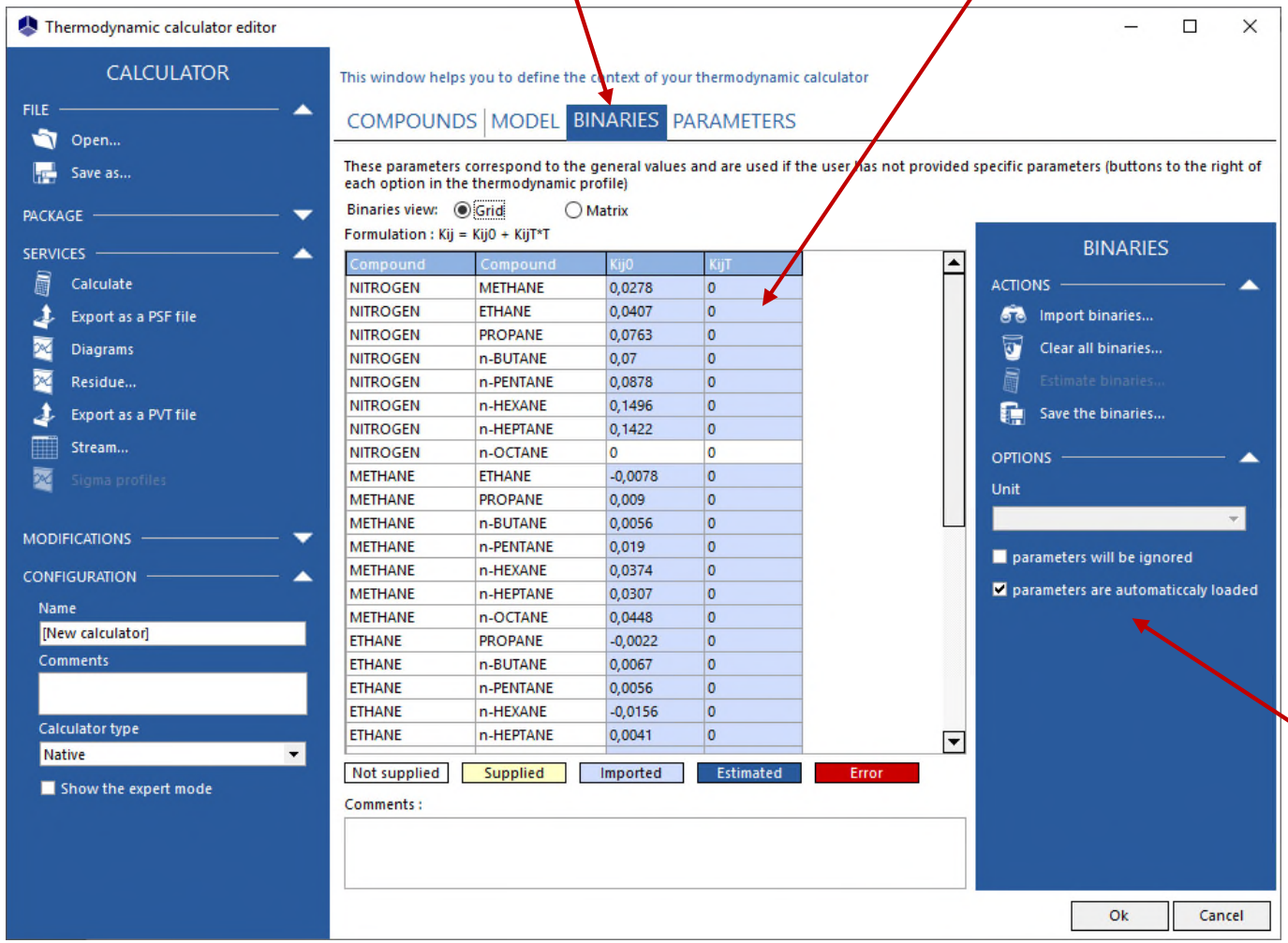


3. You can adjust the thermodynamic profile if it is necessary

STEP #2: Select your thermodynamic model

1. Click on the “Binaries” tab to enter the binaries search window (if required by the model)

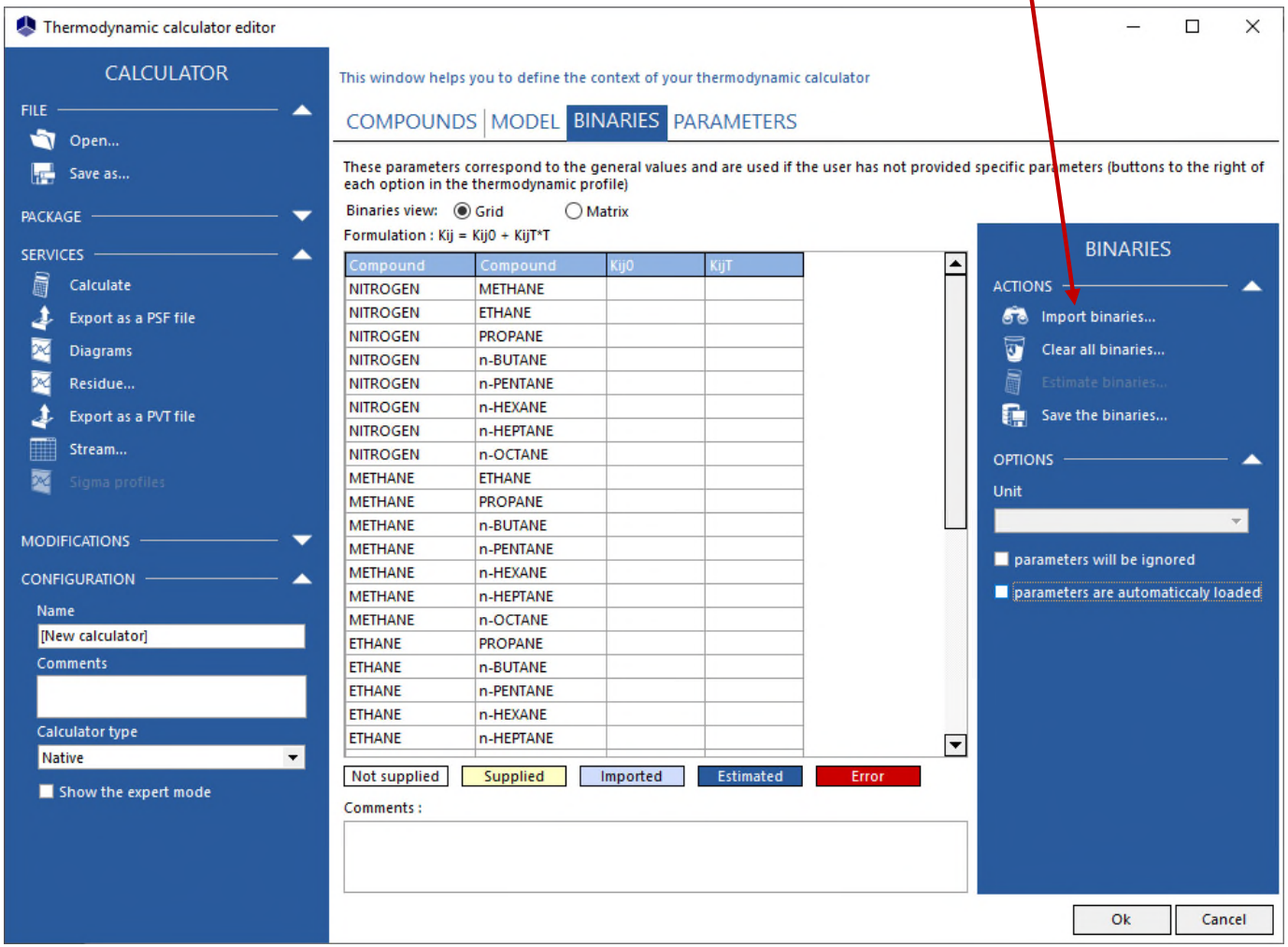
If binary interaction parameters are available, they are automatically loaded.



Possibility to inactivate the automatic loading of the binary interaction parameters

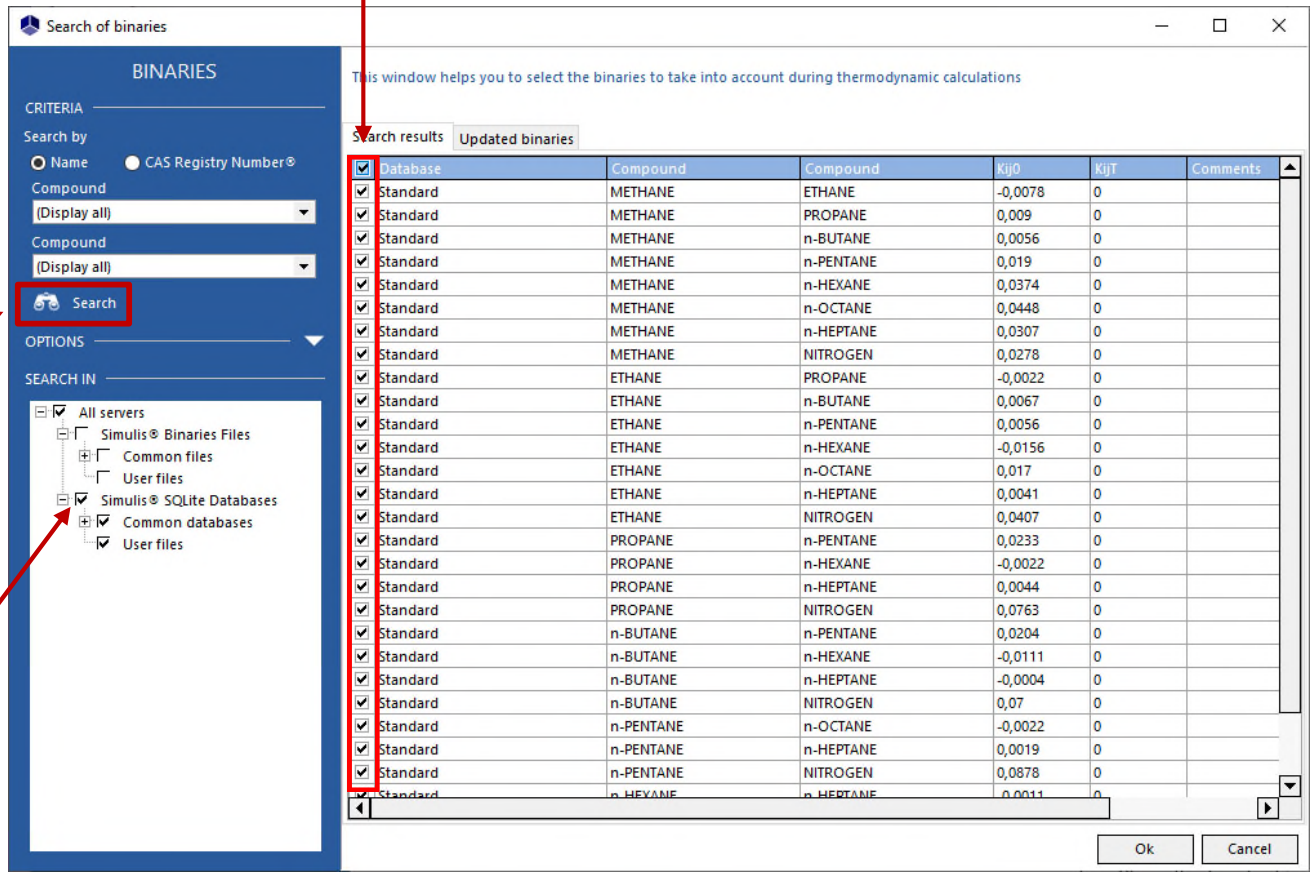
STEP #2: Select your thermodynamic model

Click on “Import binaries…” to search for the binary interaction parameters in the databases if some parameters are missing, the automatic loading is inactivated, etc.



STEP #2: Select your thermodynamic model

3. Select the binary interaction parameters to be used and click on “OK”

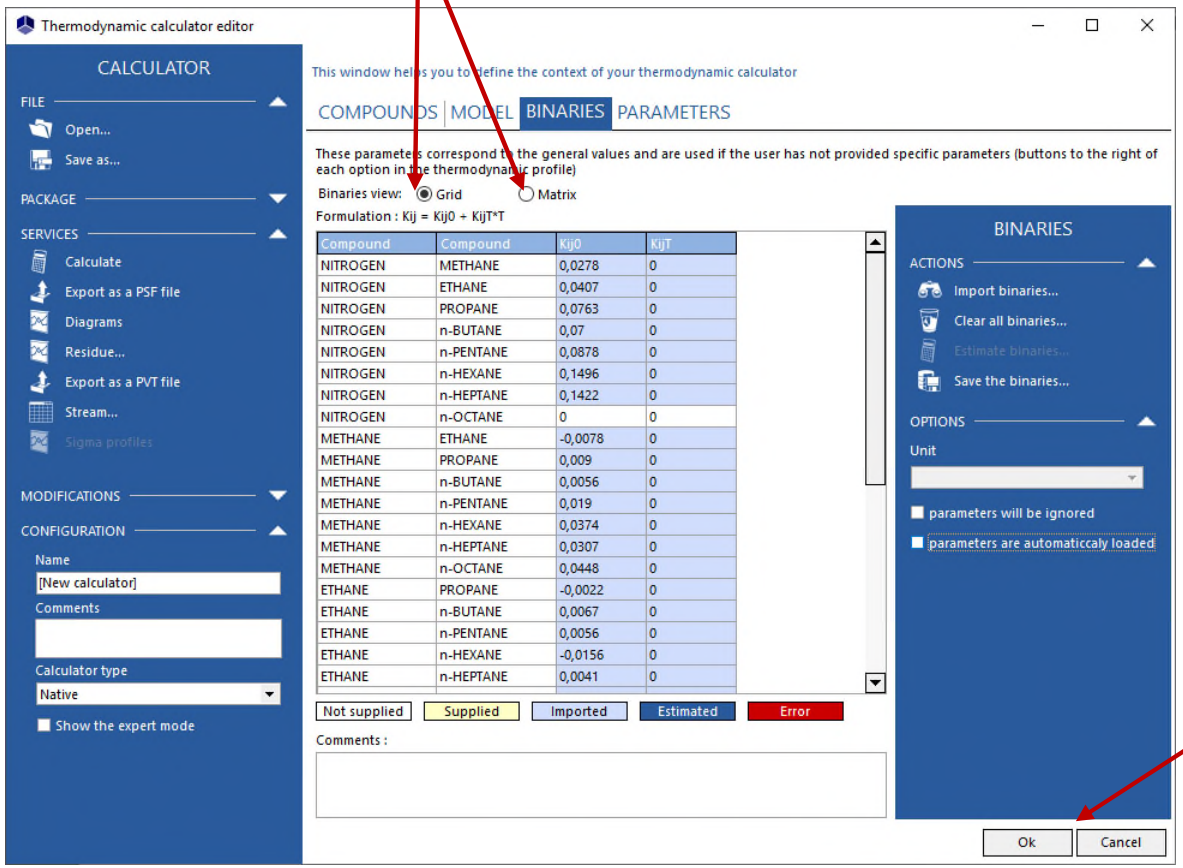


2. Select the compounds and click on “Search”

1. Select the binaries server(s) that you want to use for your research

STEP #2: Select your thermodynamic model

You can display the binaries as a grid or a matrix



Click on “OK” to validate

Your thermodynamic calculator is now defined:

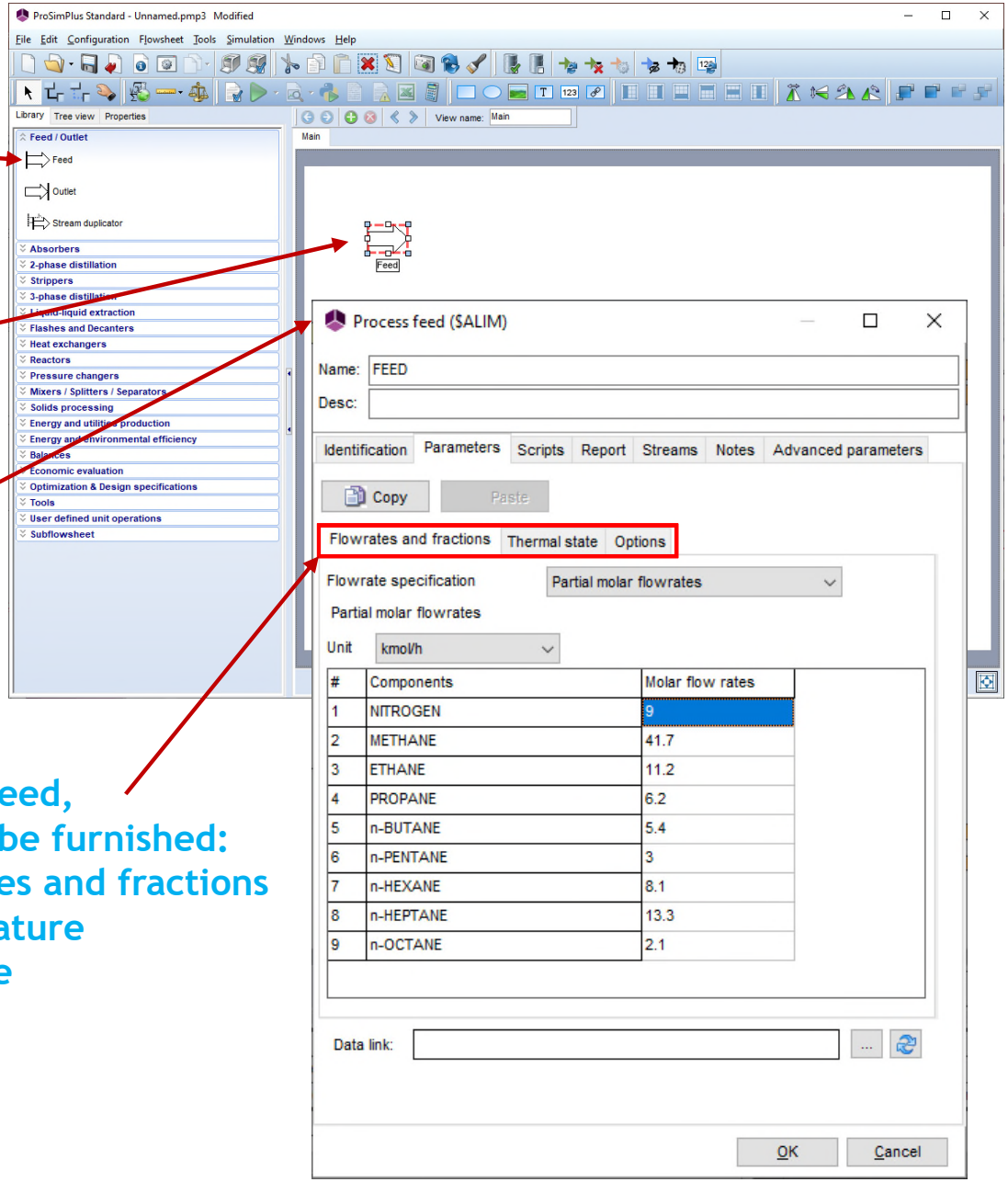
- Components
- Thermodynamic model
- Binary interaction parameters (if necessary)

STEP #3: Create your flowsheet

1. Select the process inlet icon (simple click)

2. Drop the icon in the flowsheet (simple click)

3. Double click on the unit to describe it



For a process feed, parameters to be furnished:

- Flowrates and fractions
- Temperature
- Pressure

At least one Process feed (inlet) and one Process outlet are necessary to run a simulation

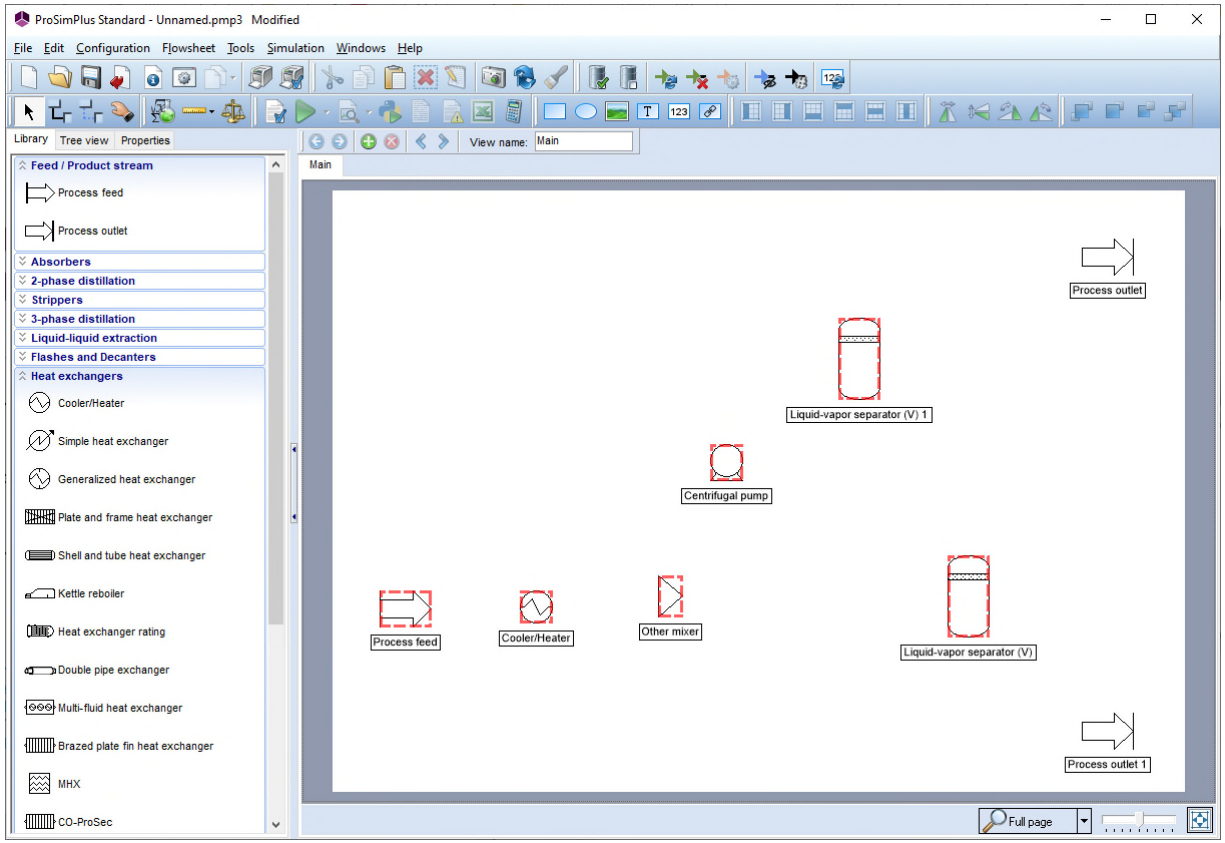
STEP #3: Create your flowsheet

Repeat the operation for all the modules:

1. Click to select a unit operation

2. Move the mouse on to the drawing sheet and reach the desired position

3. Click again, to release the unit. Operating parameters of the unit operation can be set at any time

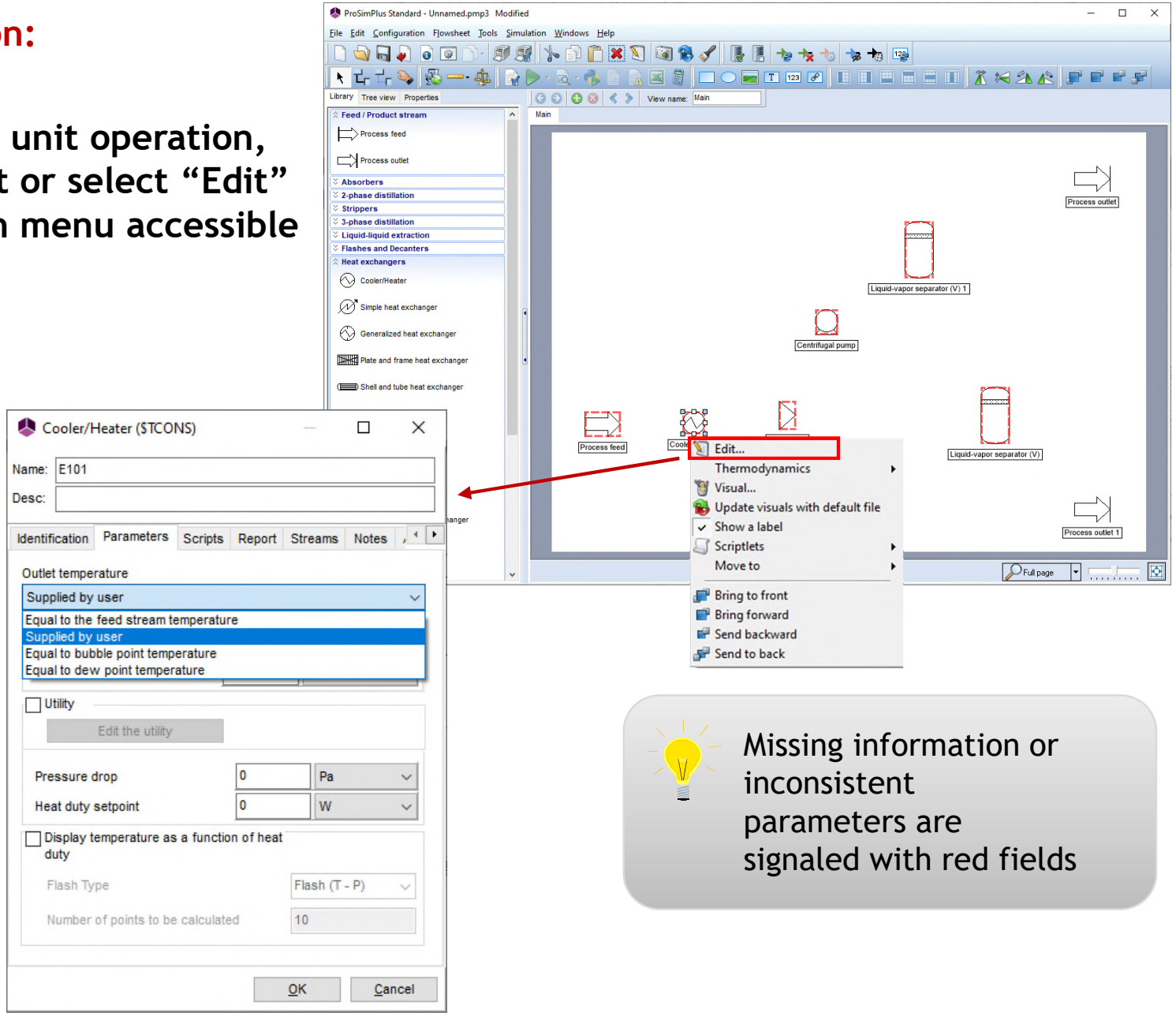


A comprehensive set of features allows you to resize, rotate, reposition, align, etc. the element on the drawing area

STEP #3: Create your flowsheet

Configure each unit operation:

1. To configure a unit operation, double click on it or select “Edit” in the drop-down menu accessible by a right-click



Missing information or inconsistent parameters are signaled with red fields

STEP #3: Create your flowsheet

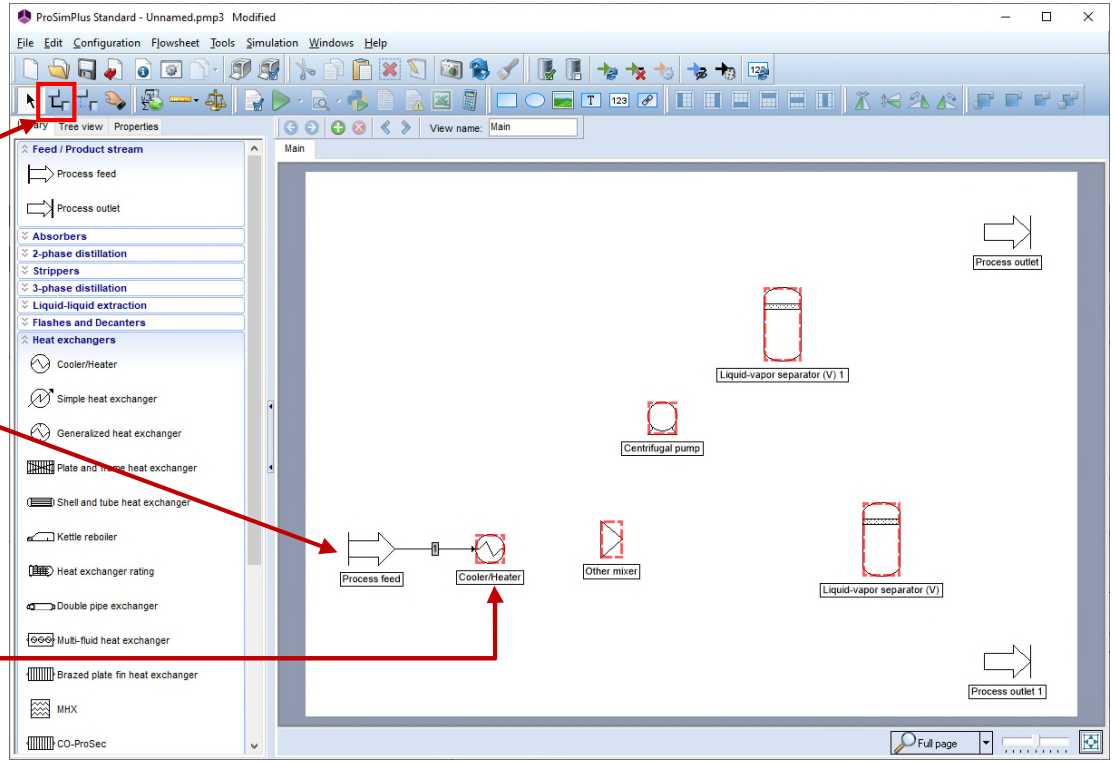
Connect the unit operations:

1. Select the “Material Stream” icon

2. Select the first unit operation (source) by clicking on it

3. Select the second unit operation (target) by clicking on it

4. Both unit operations are automatically connected



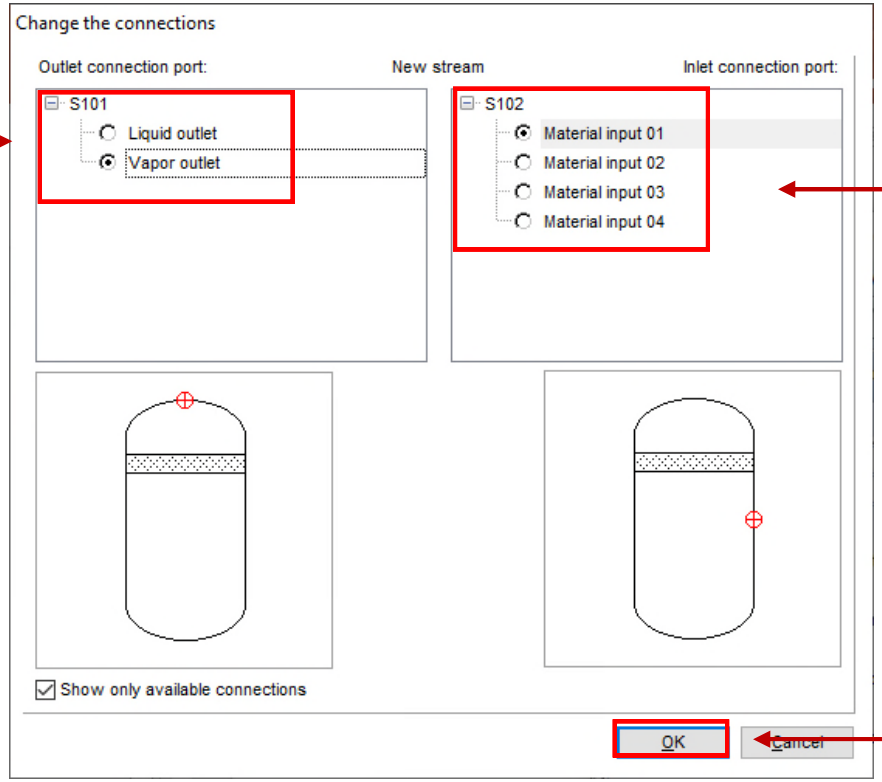
TIP : Press “shift” and click on the “Material stream” icon to connect successively several unit operations.

Material streams can be colored in order to ease the reading of the flowsheet. Right click on it to access this feature.

STEP #3: Create your flowsheet

When several connections are available, the following window appears allowing to select the right connection point:

1. Select the unit operation outlet



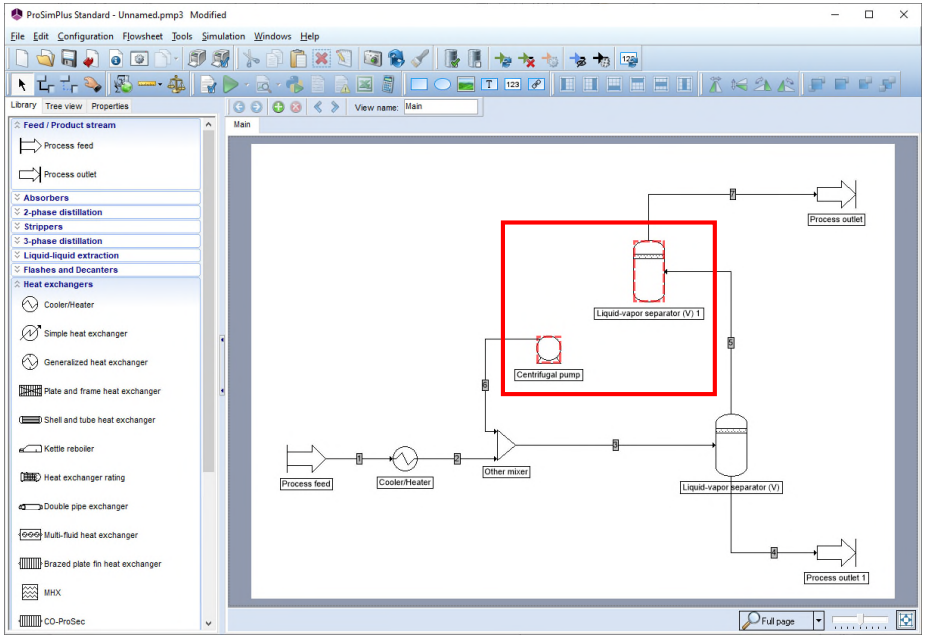
2. Select the unit operation inlet

3. Confirm by clicking on “OK”

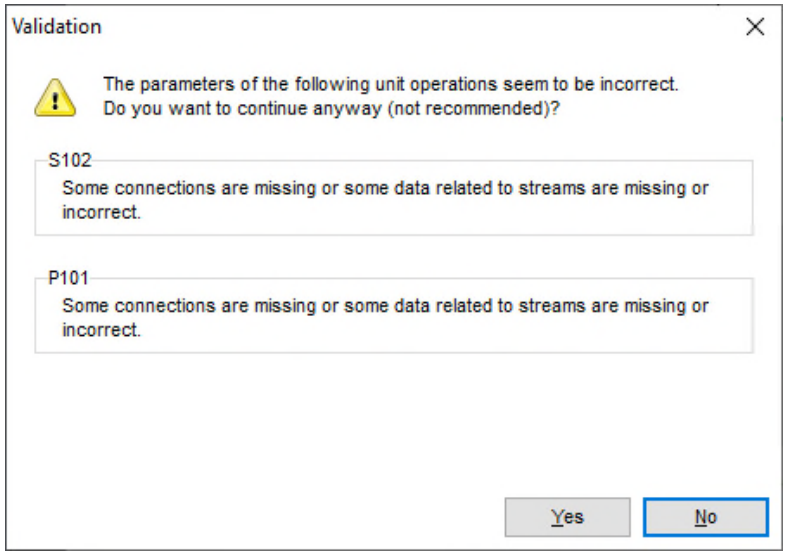
STEP #4: Run the simulation

Checking the flowsheet:

- At the flowsheet level, you can easily verify the consistency of your connections. If for a unit operation, a connection is missing it is identified with a red square. Placing your mouse on the unit operation will display a warning message



- If the simulation is run and an error is detected, a warning message will appear, with the name of the incorrect unit operation

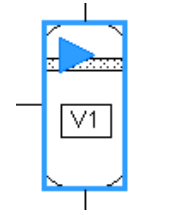


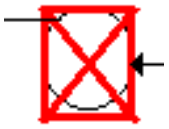
STEP #4: Run the simulation

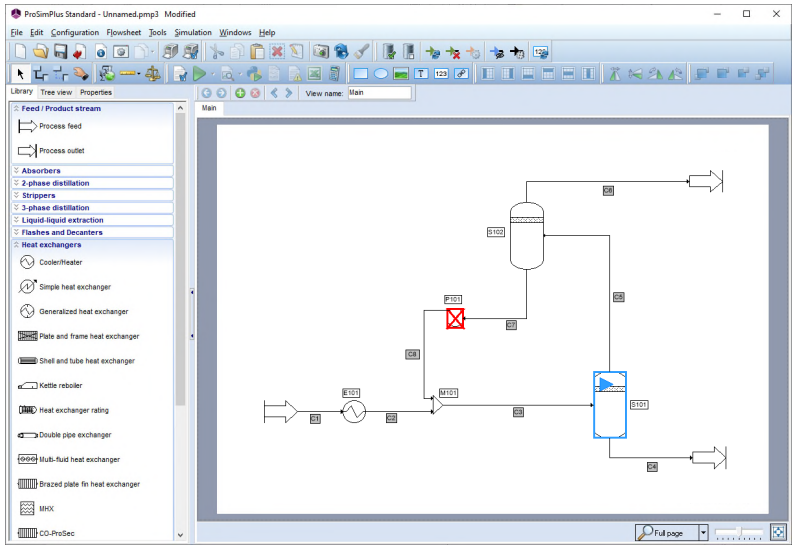
Click on the green arrow or press “F9” to run the simulation

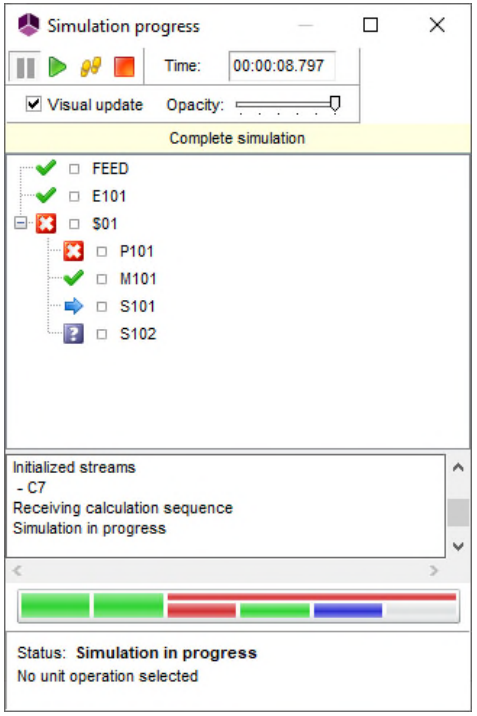
During the calculation, different symbols and indications will appear and disappear in the *“Simulation progress”* window and in the drawing area:

- A blue question mark indicates that the module has not been calculated yet
- A blue arrow indicates that calculation is in progress
- A green validation mark indicates that the module has been correctly calculated
- A red cross indicates a convergence error
- A yellow frame indicates a warning on the module results





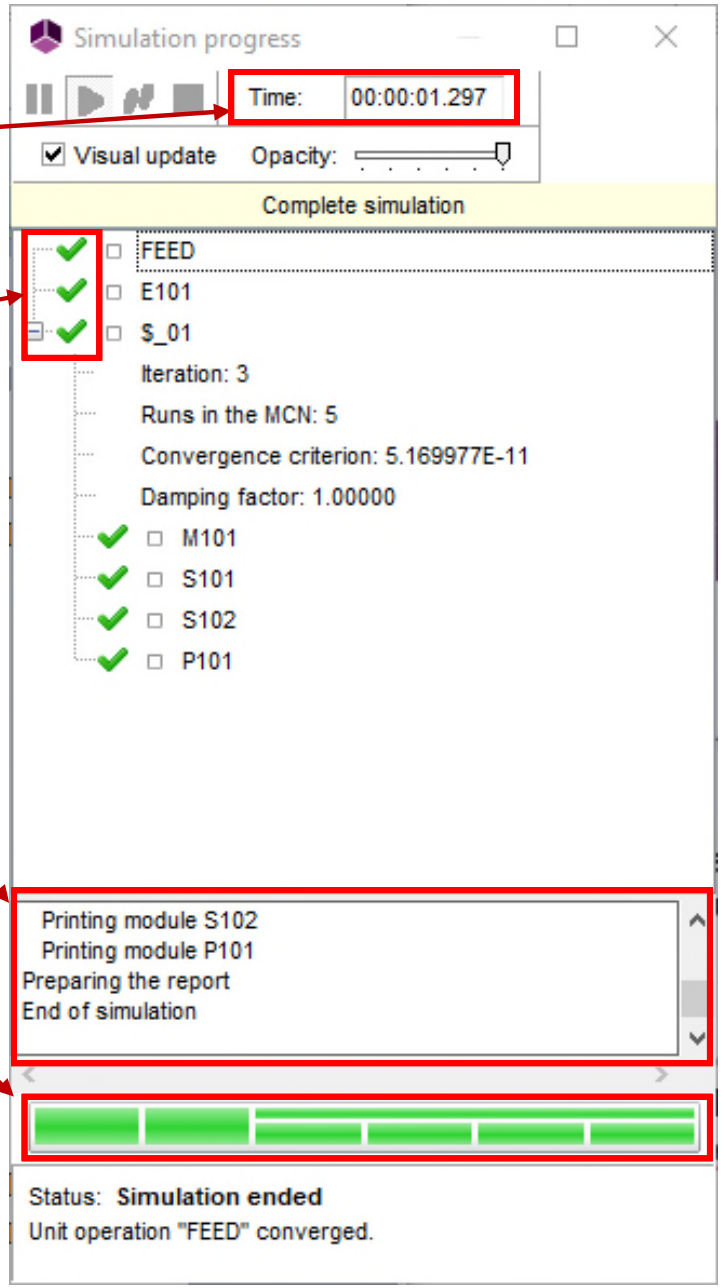




STEP #4: Run the simulation

Running the simulation:

- Simulation time
- If all modules go with a green validation, the simulation is successful
- Display of modules simulation progress, results prints (streams, modules, HCurves, TBP/ASTM curves…)
- Progress bars of the simulation



Closing the simulation progress window allows to access the simulation results

STEP #5: Simulation reports

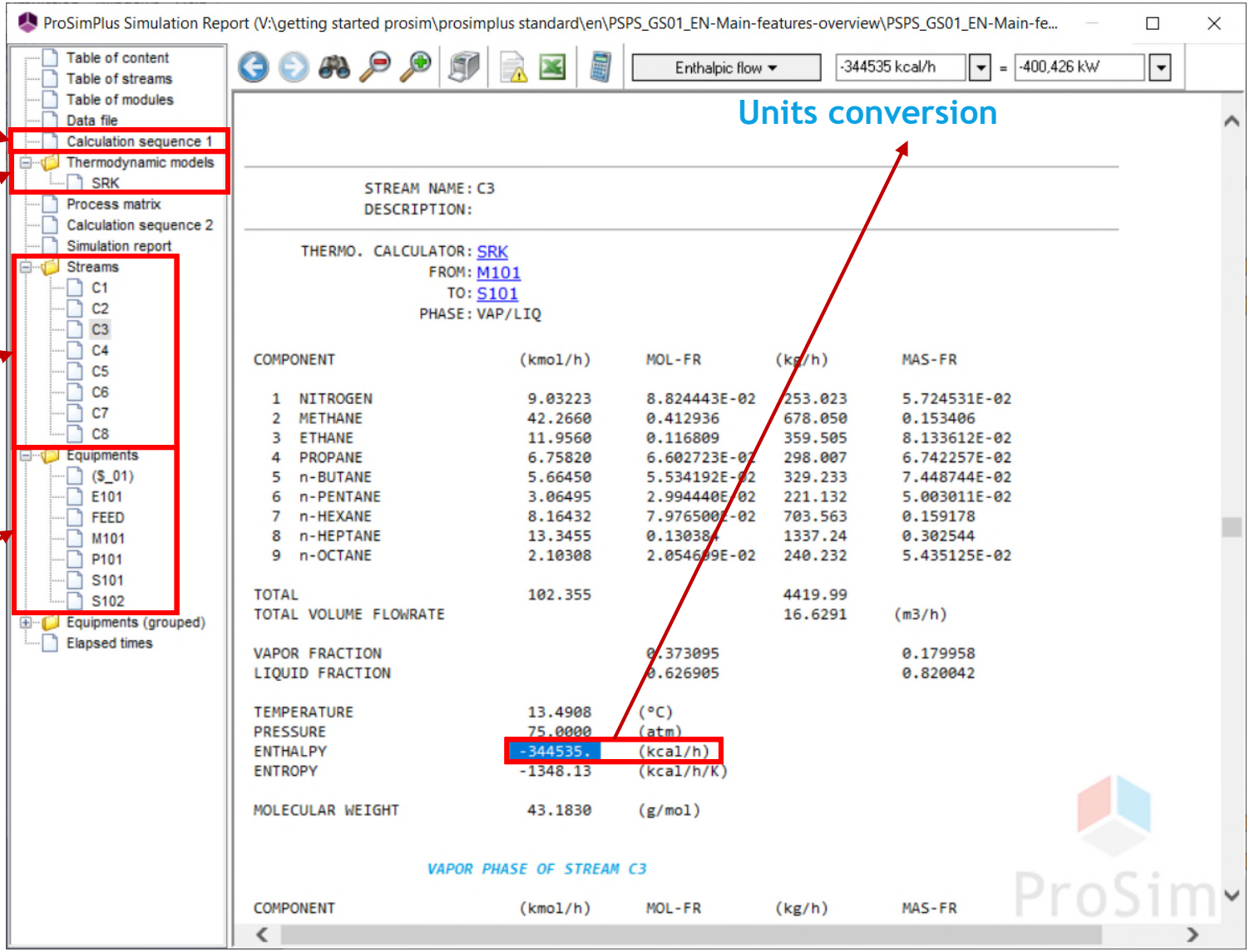
An HTML report is automatically generated and gives you a convenient access to the results:

Calculation sequence

Thermodynamic models

Stream properties

Module results

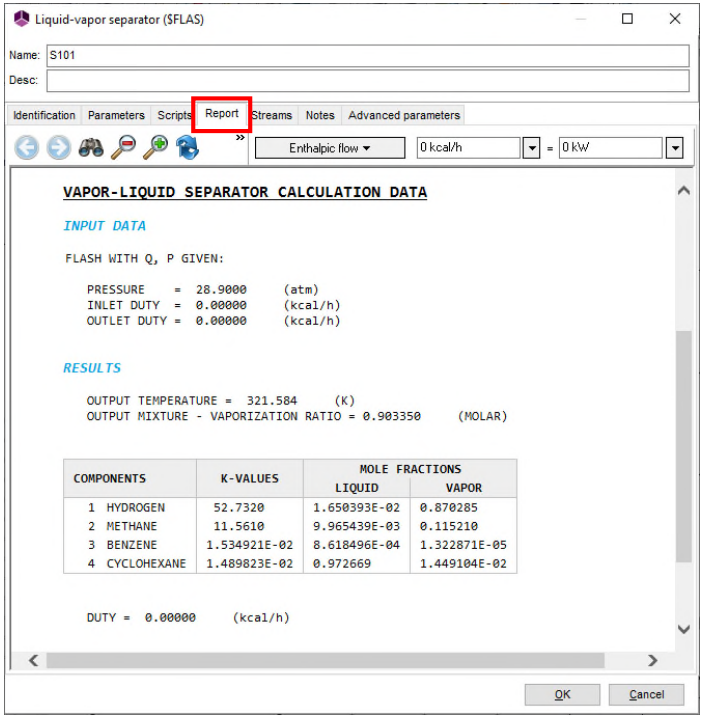


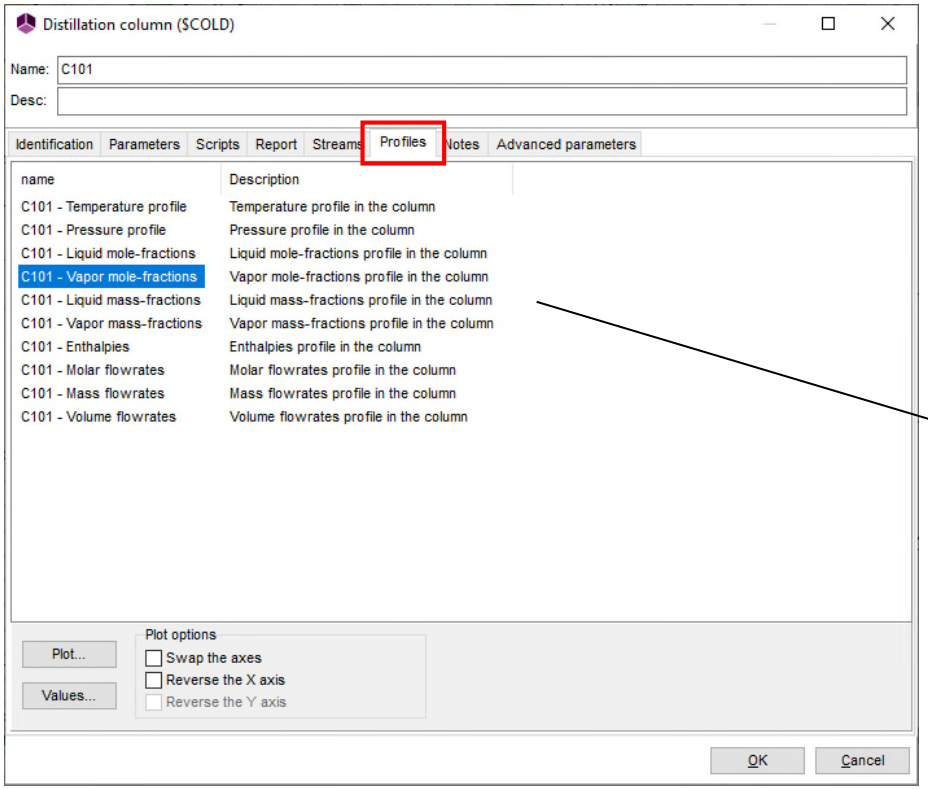
STEP #5: Simulation reports

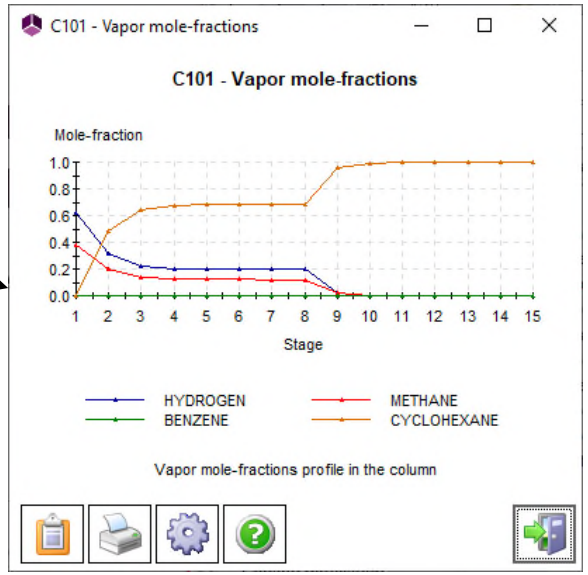
Unit operations

- You can double-click on any module of the flowsheet, and then click on the “Report” tab to access the results for this specific module.

- For some modules (columns for example), a “Profiles” tab is also available.





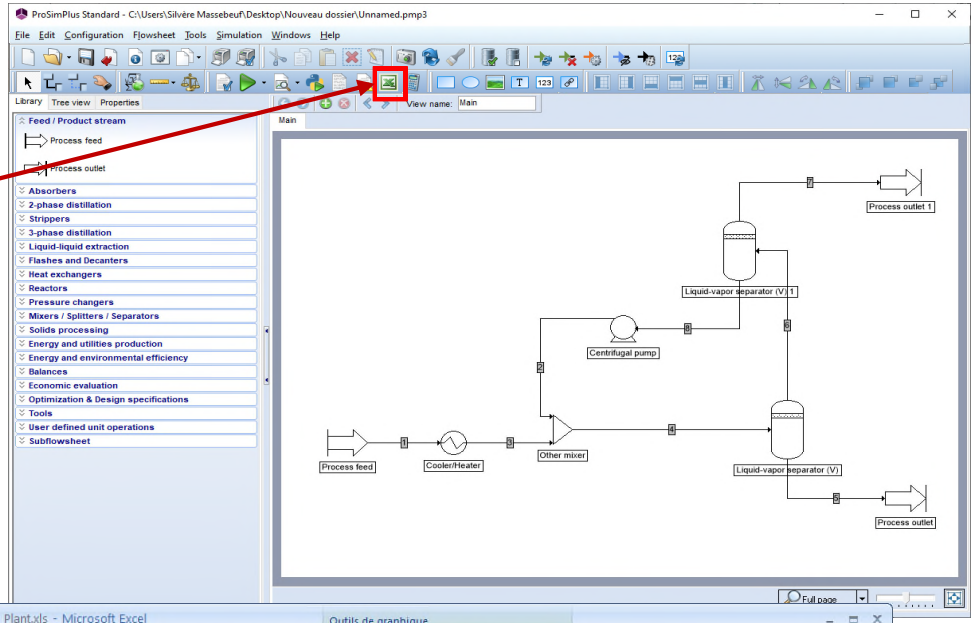


STEP #5: Simulation reports

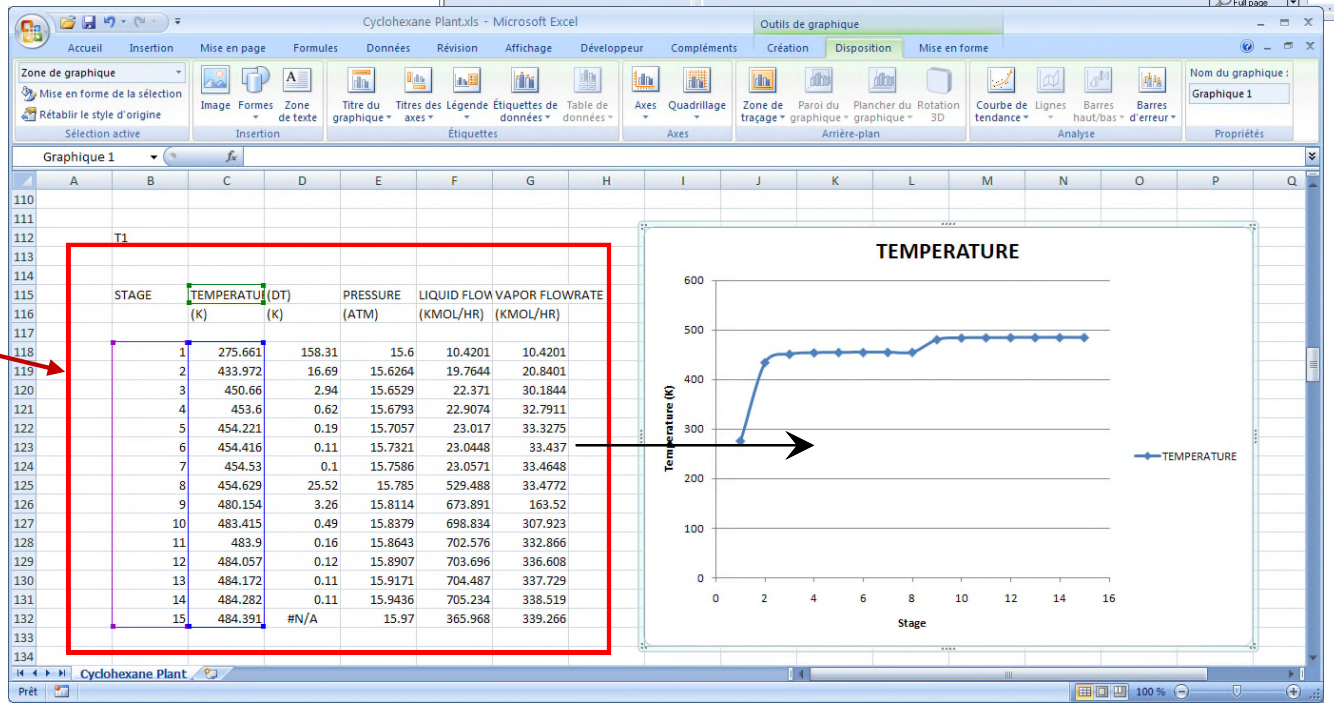
1. Click on the “Excel file” icon to open the Excel report

Excel file (csv format)

- Balances on process streams
- Equipment profiles
- Etc…



2. Recover the results in order to exploit them in Excel

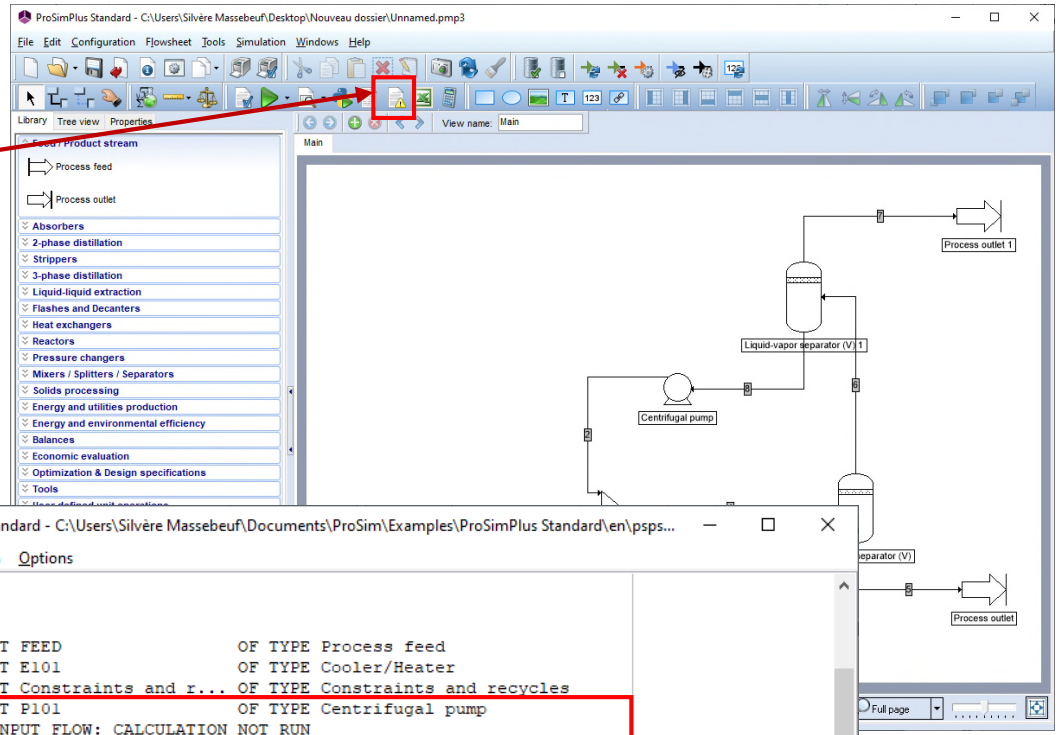


STEP #5: Simulation reports

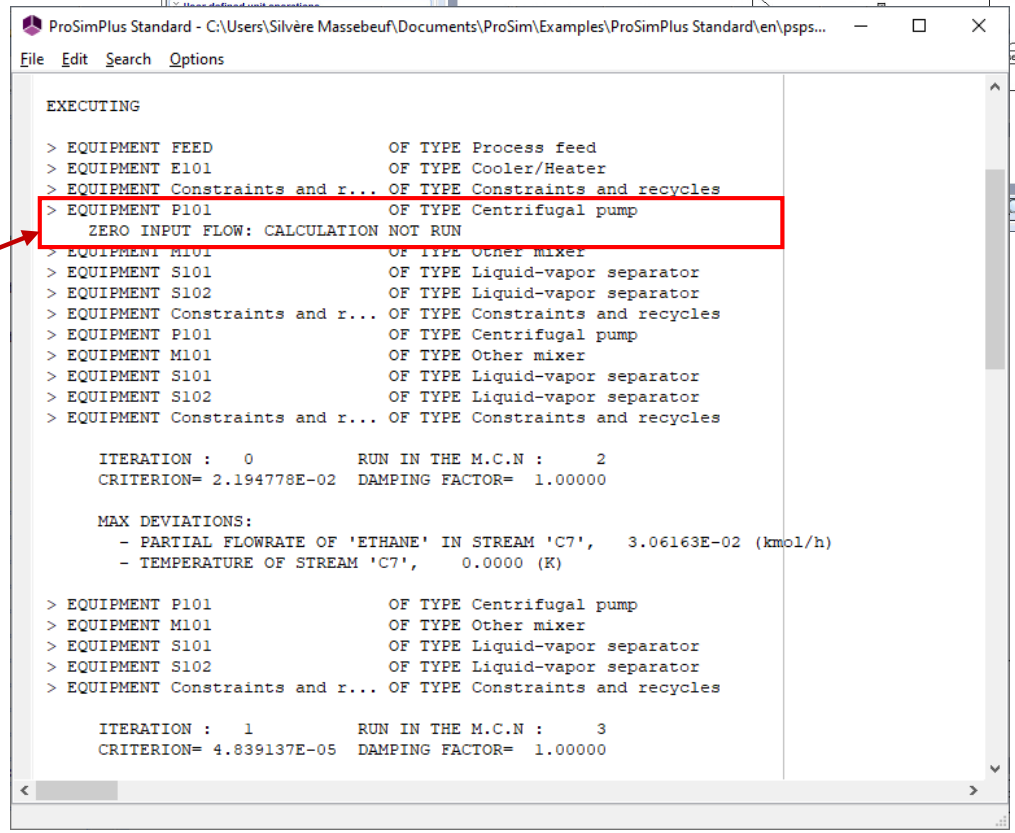
1. Click on the “History file” icon

History file

- List of calculations
- List of encountered errors



2. For each module, display of the possible errors or warning messages



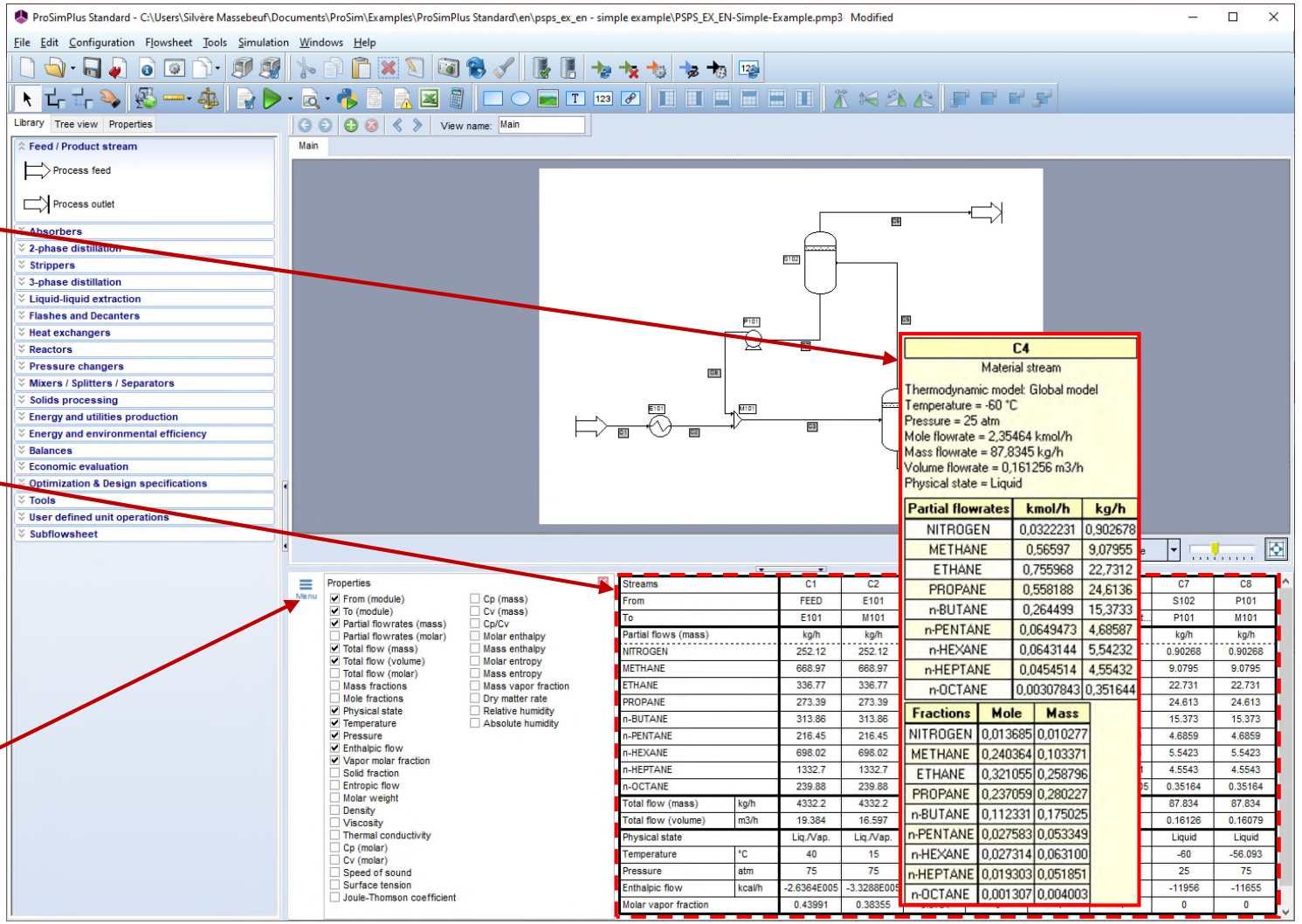
STEP #6: Results analysis

You can analyze the results directly from the flowsheet

Hints: move the cursor over a stream to display its properties

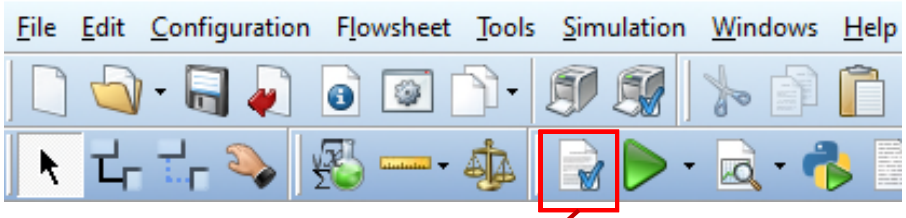
Streams summary: Summary of streams properties

Click on Menu to configure the streams properties for the display and the export in MS-Excel

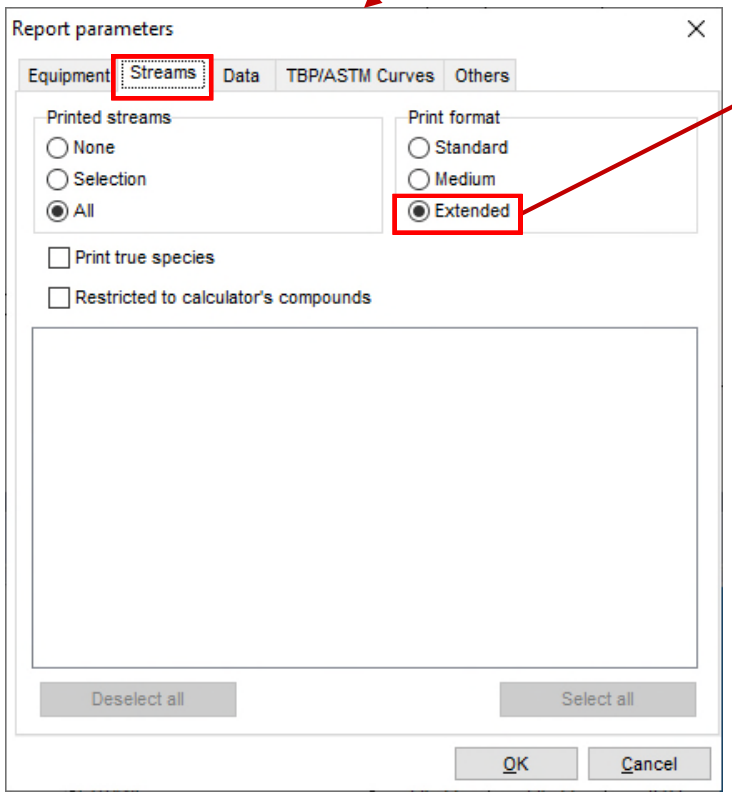


STEP #6: Results analysis

Streams summary: Displaying physical properties



"Streams" tab



By default, all the physical properties are calculated

Streams		C1	C2	C3	C4	C5	C6	C7	C8
From		FEED	E101	M101	S101	S101	S102	S102	P101
To		E101	M101	S101	Process out...	S102	Process out...	P101	M101
Total flow (mass)	kg/h	4332.2	4332.2	4420	3622.6	797.37	709.54	87.835	87.835
Total flow (volume)	m3/h	19.384	16.597	16.629	6.4803	10.211	21.561	0.16126	0.16079
Physical state		Liq./Vap.	Liq./Vap.	Liq./Vap.	Liquid	Vapor	Vapor	Liquid	Liquid
Temperature	°C	40	15	13.491	13.451	13.451	-60	-60	-56.093
Pressure	atm	75	75	75	74.7	74.7	25	25	75
Enthalpic flow	kcal/h	-2.6364E005	-3.3288E005	-3.4453E005	-3.2579E005	-18746	-32320	-11956	-11655
Molar vapor fraction		0.43991	0.38355	0.37309	0	1	1	0	0
Density	kg/m3	223.49	261.02	265.8	559.02	78.087	32.908	544.69	546.26
Viscosity	Pa.s	(*)	(*)	(*)	0.00019512	1.4091E-005	9.6913E-006	0.00017653	0.00017764
Thermal conductivity	W/m/K	(*)	(*)	(*)	0.12498	0.039226	0.026939	0.14875	0.1497
Cp (mass)	J/kg/K	(*)	(*)	(*)	2371.2	2432.3	2118.2	2449.9	2414.9

(*): Indicates a missing or undefined value.
Set the print format option to Extended to calculate more properties.

Physico-chemical properties are printed in the grid only for monophasic streams liquid or vapor

Physico-chemical properties for di- or triphasic streams are available in the report

Print format "Standard" or "Medium":

Streams		C1	C2	C3	C4	C5	C6	C7	C8
From		FEED	E101	M101	S101	S101	S102	S102	P101
To		E101	M101	S101	Process out...	S102	Process out...	P101	M101
Total flow (mass)	kg/h	4332.2	4332.2	4420	3622.6	797.37	709.54	87.835	87.835
Total flow (volume)	m3/h								
Physical state		Liq./Vap.	Liq./Vap.	Liq./Vap.	Liquid	Vapor	Vapor	Liquid	Liquid
Temperature	°C	40	15	13.491	13.451	13.451	-60	-60	-56.093
Pressure	atm	75	75	75	74.7	74.7	25	25	75
Enthalpic flow	kcal/h	-2.6364E005	-3.3288E005	-3.4453E005	-3.2579E005	-18746	-32320	-11956	-11655
Molar vapor fraction		0.43991	0.38355	0.37309	0	1	1	0	0
Density	kg/m3	(*)	(*)	(*)	(*)	(*)	(*)	(*)	(*)
Viscosity	Pa.s	(*)	(*)	(*)	(*)	(*)	(*)	(*)	(*)
Thermal conductivity	W/m/K	(*)	(*)	(*)	(*)	(*)	(*)	(*)	(*)
Cp (mass)	J/kg/K	(*)	(*)	(*)	(*)	(*)	(*)	(*)	(*)

(*): Indicates a missing or undefined value.
Set the print format option to Extended to calculate more properties.

STEP #6: Results analysis

Add a tag (label, process value or simulation result)

1. Click on the “tag” icon and drop it on the flowsheet

2. Double-click on the “tag” object to open its configuration window

3. Select the object and parameter to display

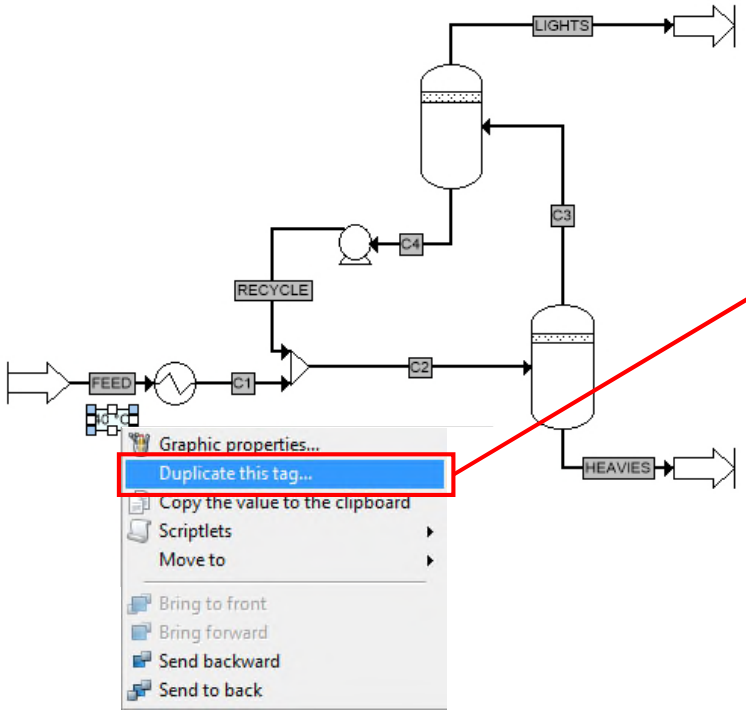
4. You can add text

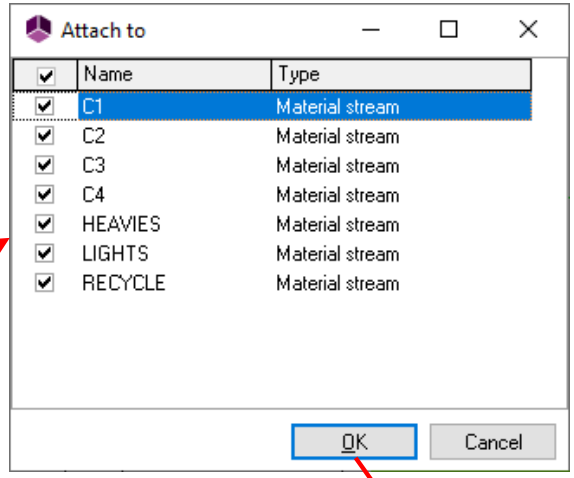
In this example, the pressure value of the “C5” stream will be displayed

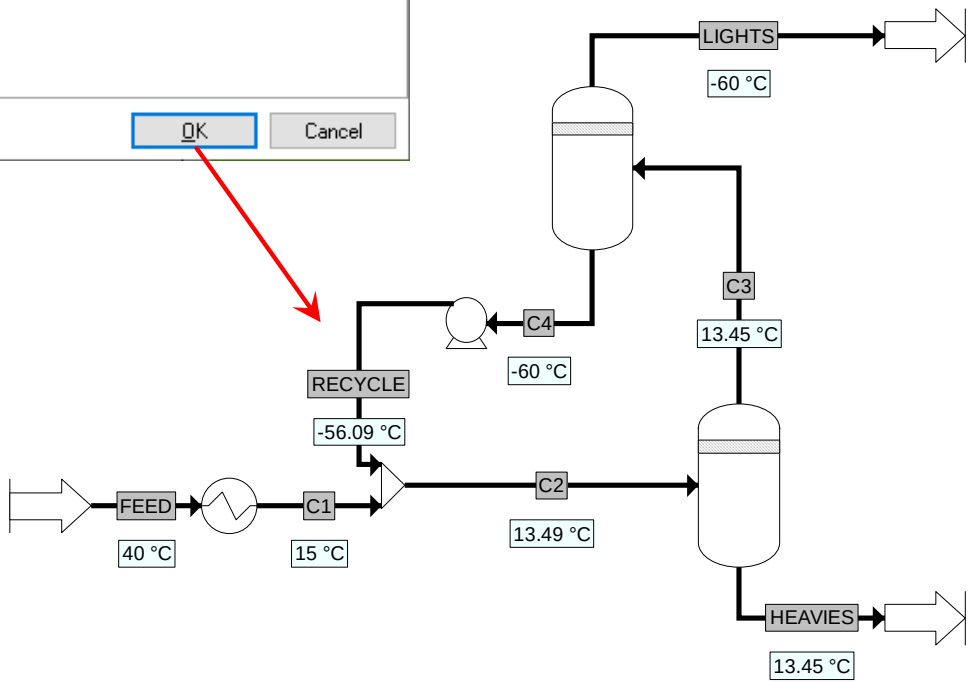
STEP #6: Results analysis

Duplicate a tag for a stream or a module property

1. Right click on the “tag”
2. Chose “Duplicate the tag”
3. Select the streams od the modules to which link this tag

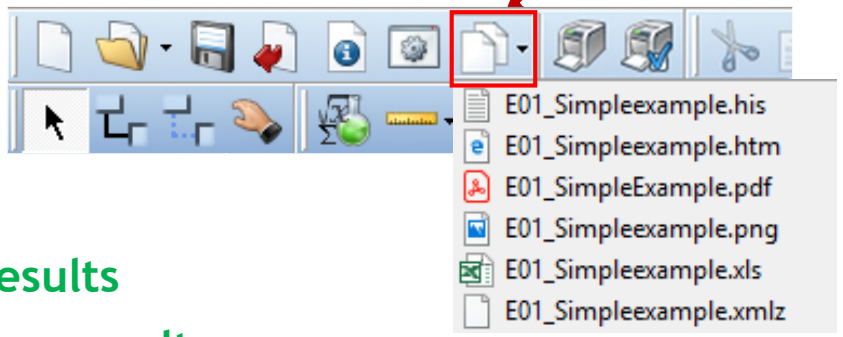






STEP #6: Results analysis

List of files that are generated in the same directory as the simulation file



- *.pmp3 : ProSimPlus3 simulation file
- *.his : history file
- *.htm : html file of the simulation results
- *.xls : MS-Excel file of the simulation results
- *.xmlz : results file corresponding to the summary grid
- *.don : data file with the generated keywords
- *.sim : data file with the generated keywords in restart mode
- *.tem : temporary file which manages the restart mode
- *.views : file which manages the graphical interface (flowsheet printing)
- …
- *.~: backup copy of all the previous files

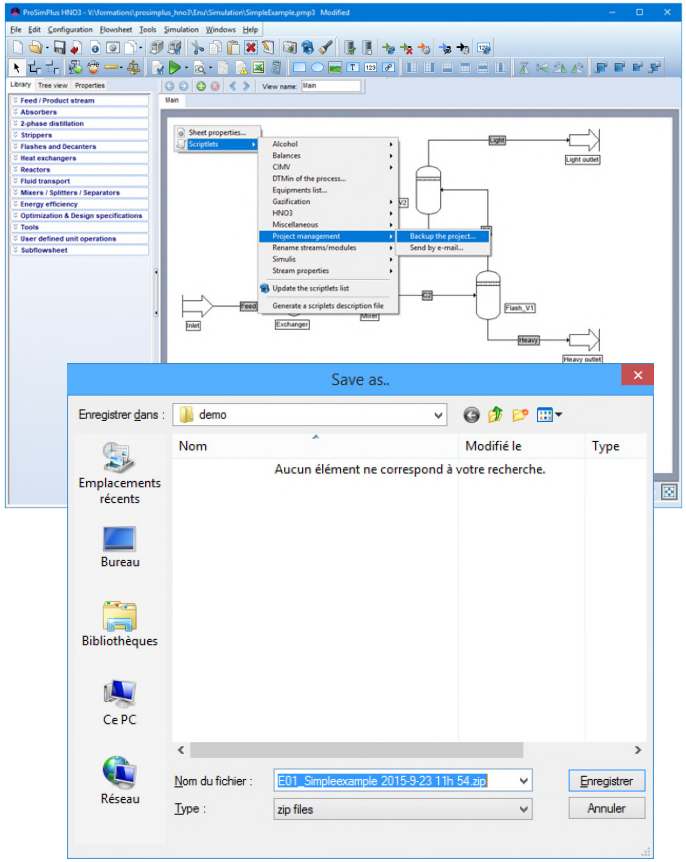
STEP #6: Results analysis

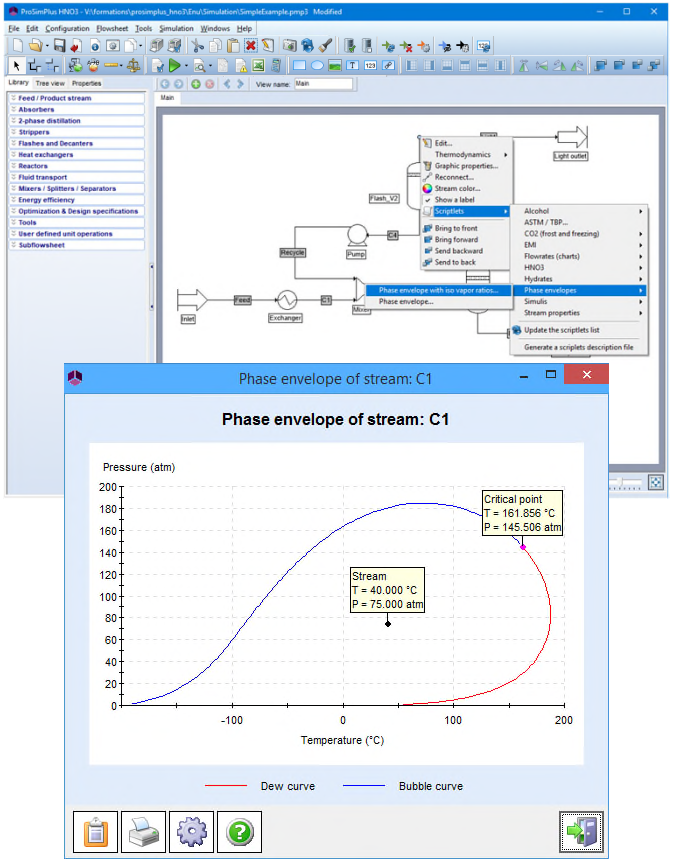
“Scriptlets”: right click on an object (project, stream, module, set of modules) to access specific scriptlets dedicated to results analysis

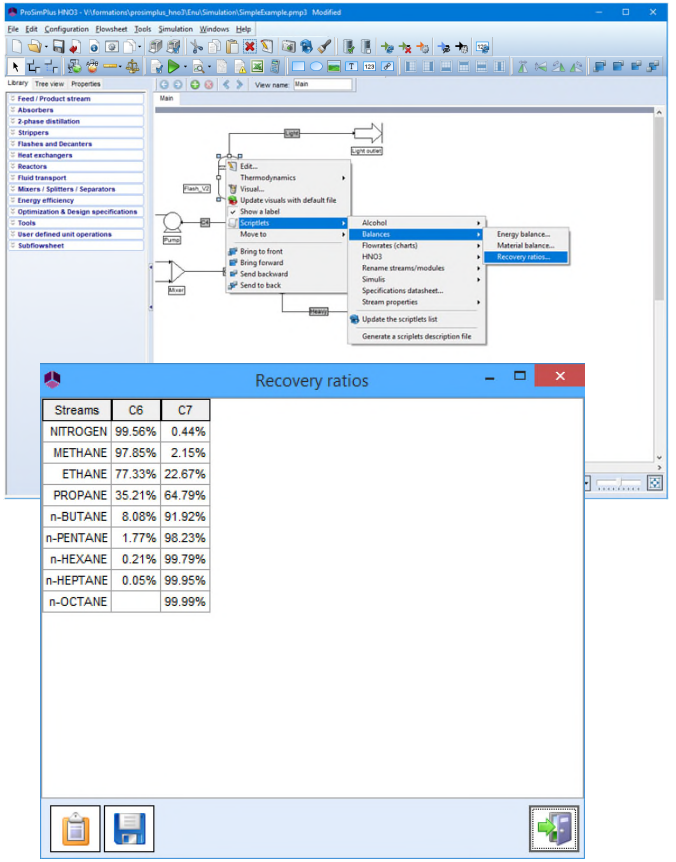
Project: Backup the project, Send by email, Material balance, Alcohol properties, Stream color and thickness…

Stream: Properties, Calculation service, Combustion, Graphs, Phase envelope…

Module(s): Material balance, Recovery ratio, Graphs, Specification data sheet (Columns, heat exchanger…), Columns export…





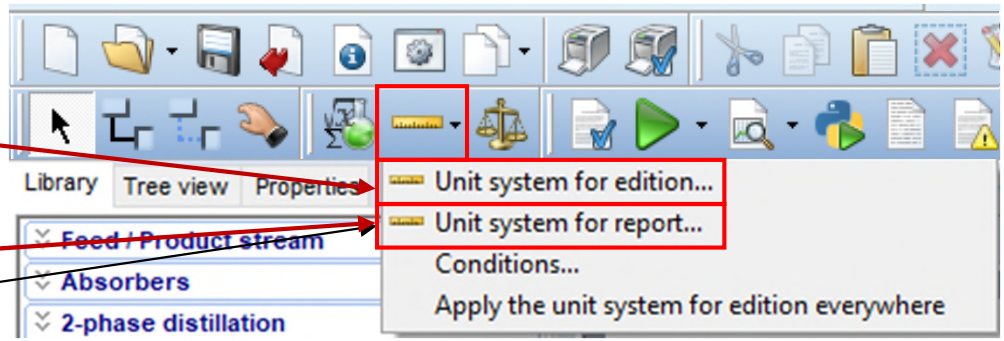


Streams	C6	C7
NITROGEN	99.56%	0.44%
METHANE	97.85%	2.15%
ETHANE	77.33%	22.67%
PROPANE	35.21%	64.79%
n-BUTANE	8.08%	91.92%
n-PENTANE	1.77%	98.23%
n-HEXANE	0.21%	99.79%
n-HEPTANE	0.05%	99.95%
n-OCTANE		99.99%

STEP #7: Graphical User Interface

1. Click here to access the unit system used for input data (unit operations, streams…)

2. Click here to access the unit system used in the simulation reports (report, summary, hint,…)

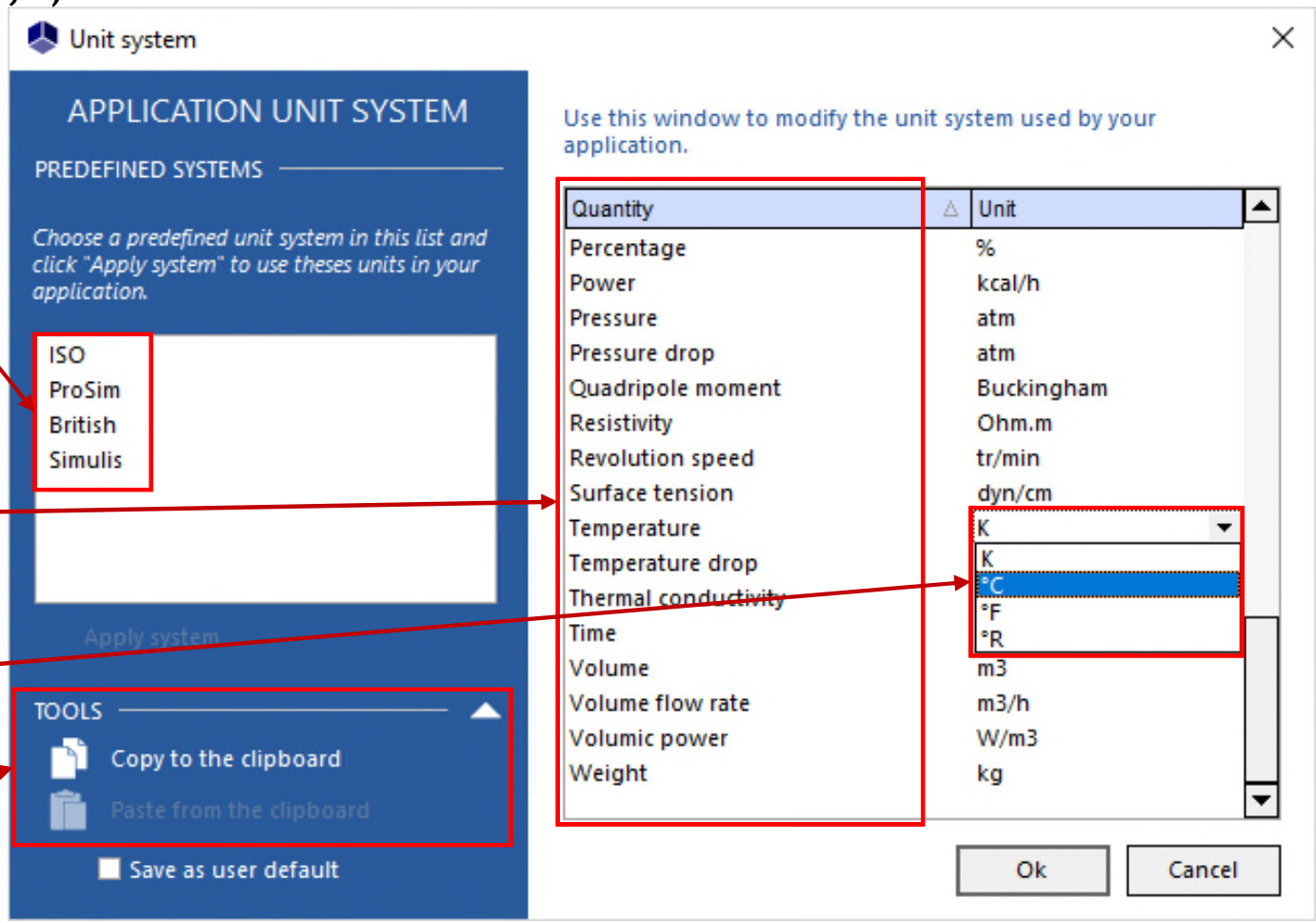


Predefined unit system

List of all the available quantities

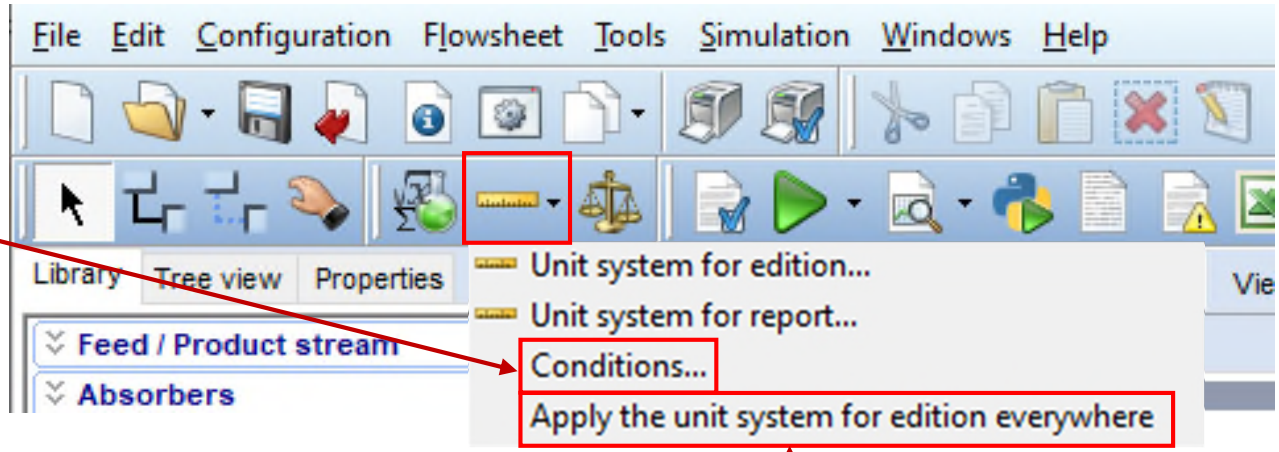
For each quantity, list of the available units

Copy/Paste all the unit system

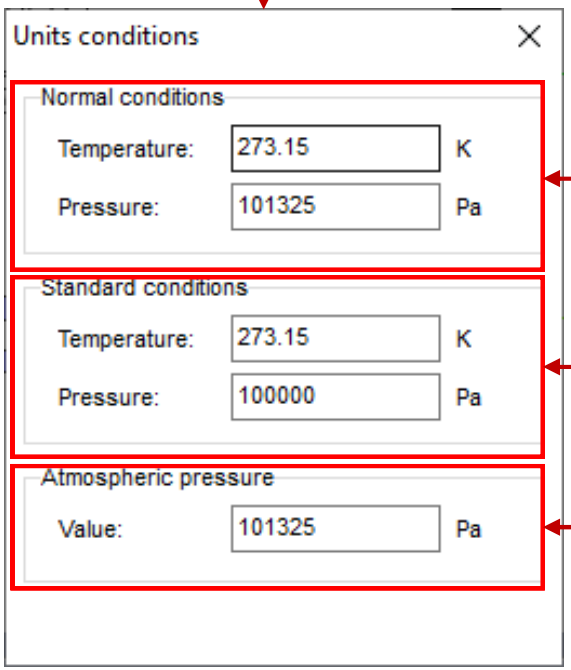


STEP #7: Graphical User Interface

1. Click on “Conditions” if you want to modify the default normal and standard conditions



2. Click here if you want to apply the updated units system to the modules already on the flowsheet



Normal conditions (T and P)

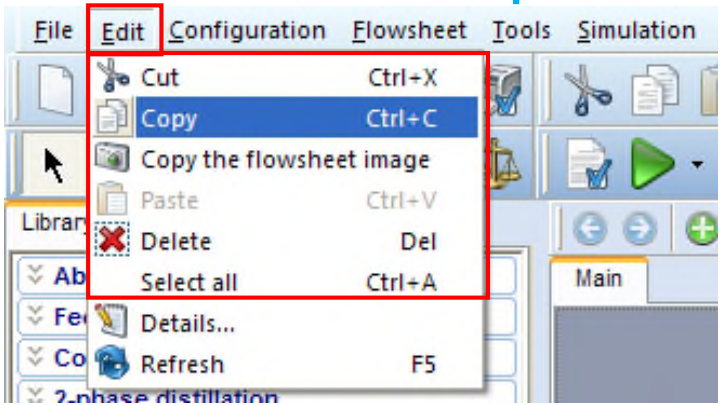
Standard conditions (T and P)

Atmospheric pressure for pressure conversion in relative unit

STEP #7: Graphical User Interface

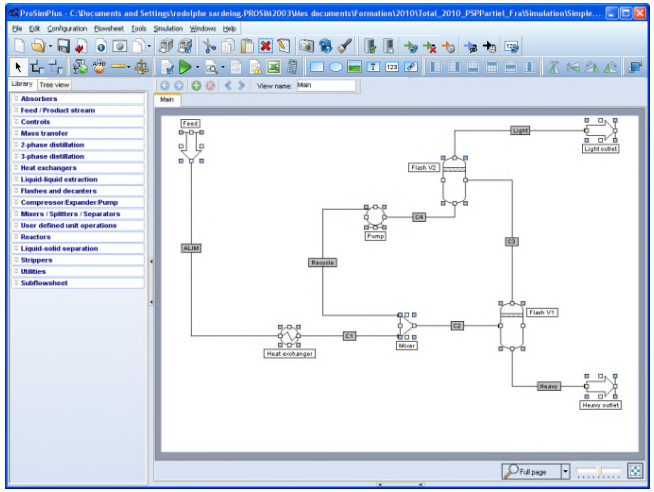
You can apply following functions on one or more unit operations:

- Cut (CTRL+ X)
- Copy (CTRL+ C)
- Paste (CTRL+ V)
- Select all (CTRL+ A)

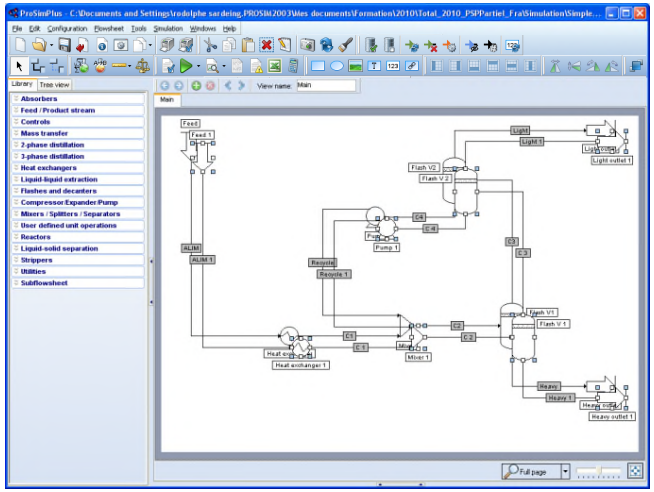


Example:

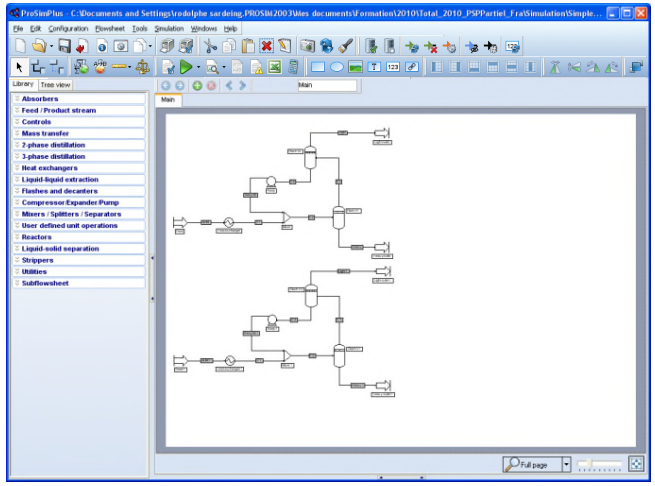
1. Press “CTRL+A” to select all the units in the flowsheet



2. Press “CTRL+C” then “CTRL+V” to duplicate the flowsheet



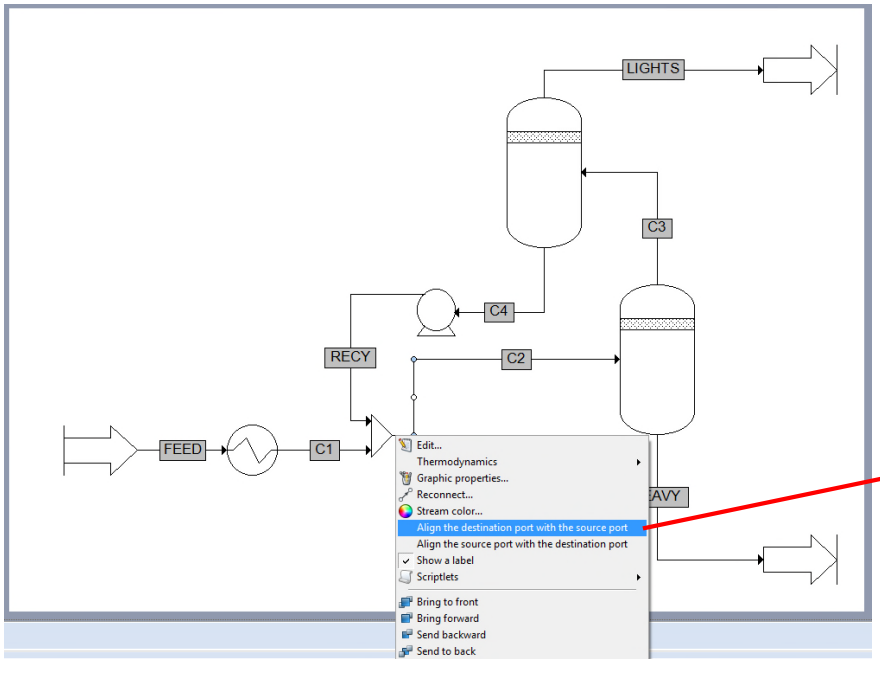
3. Move the new flowsheet and run the simulation (all unit operations parameters are also copied)

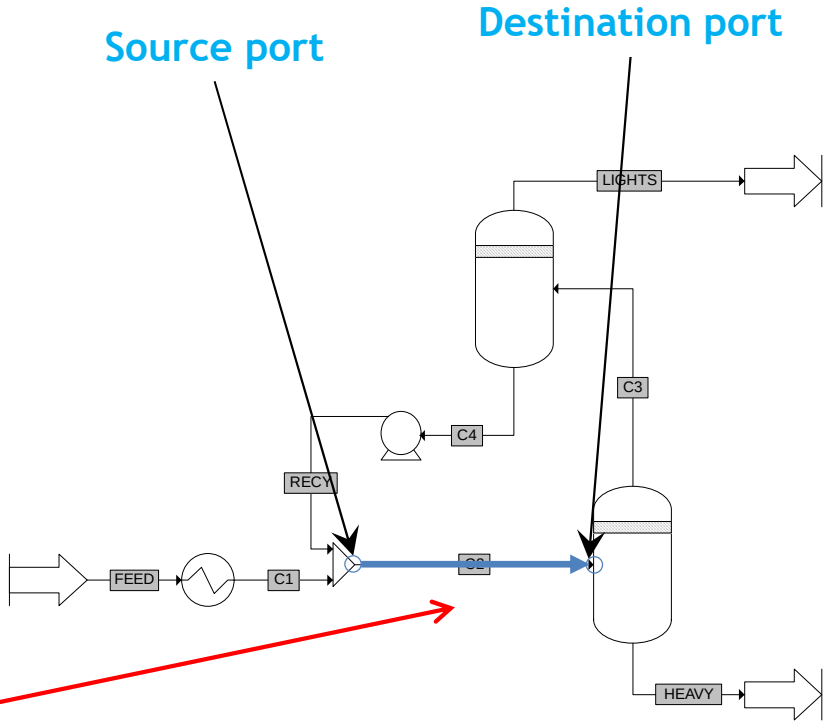


STEP #7: Graphical User Interface

You can obtain rectilinear lines:

1. Right click on the stream
2. Select:
   - Align the destination port with the source port
   - Align the source port with the destination port



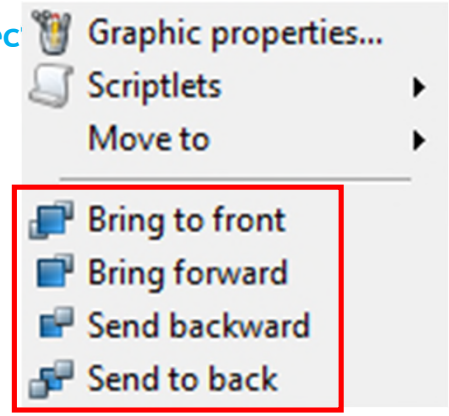


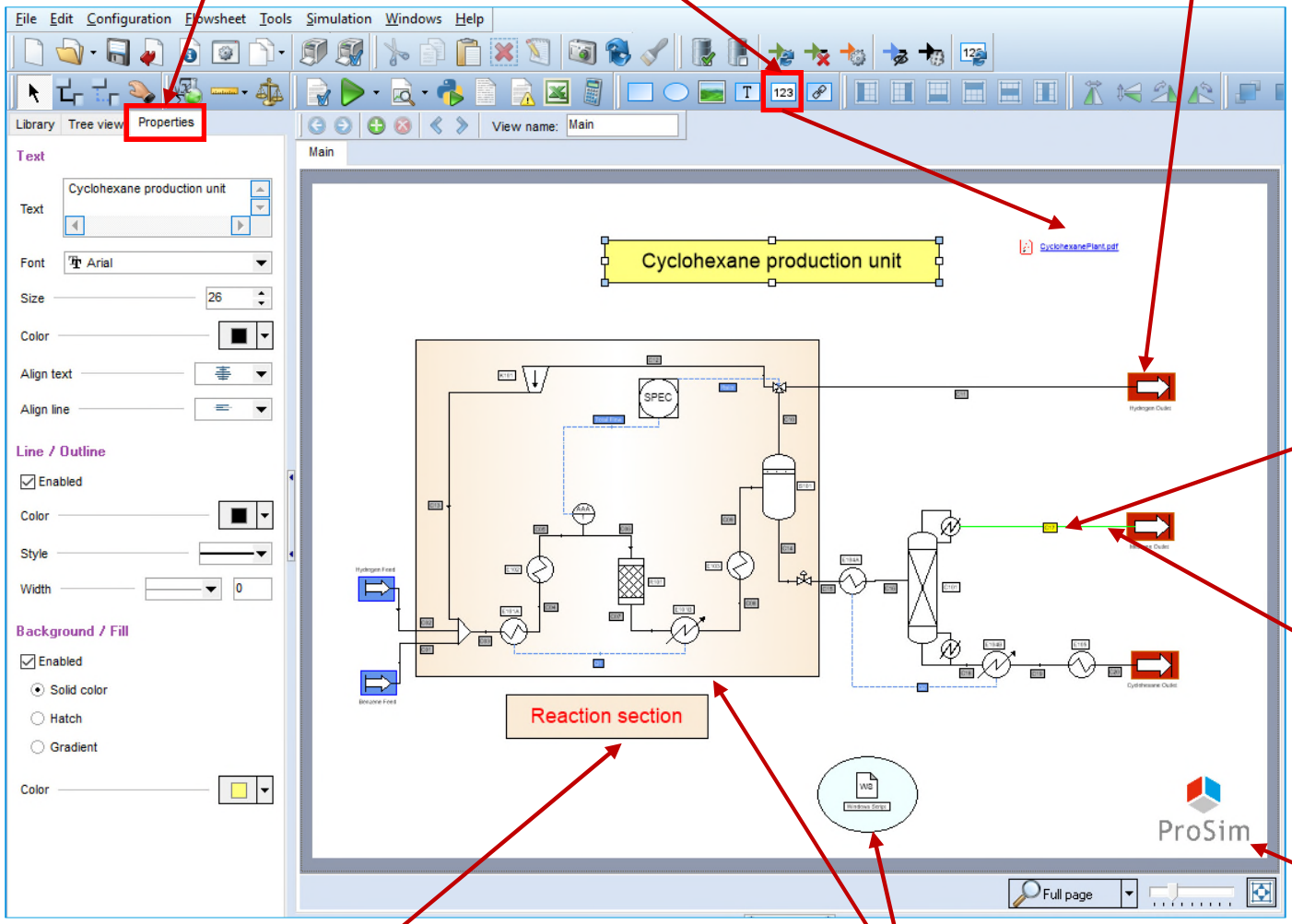
STEP #7: Graphical User Interface

Graphical properties of objects

Link to a document (pdf, web site…)

Layout of the graphical objec (front, background…)





Color, width… modification of the streams and equipments tags

Color and thickness modification of the streams

Inserting text blocks

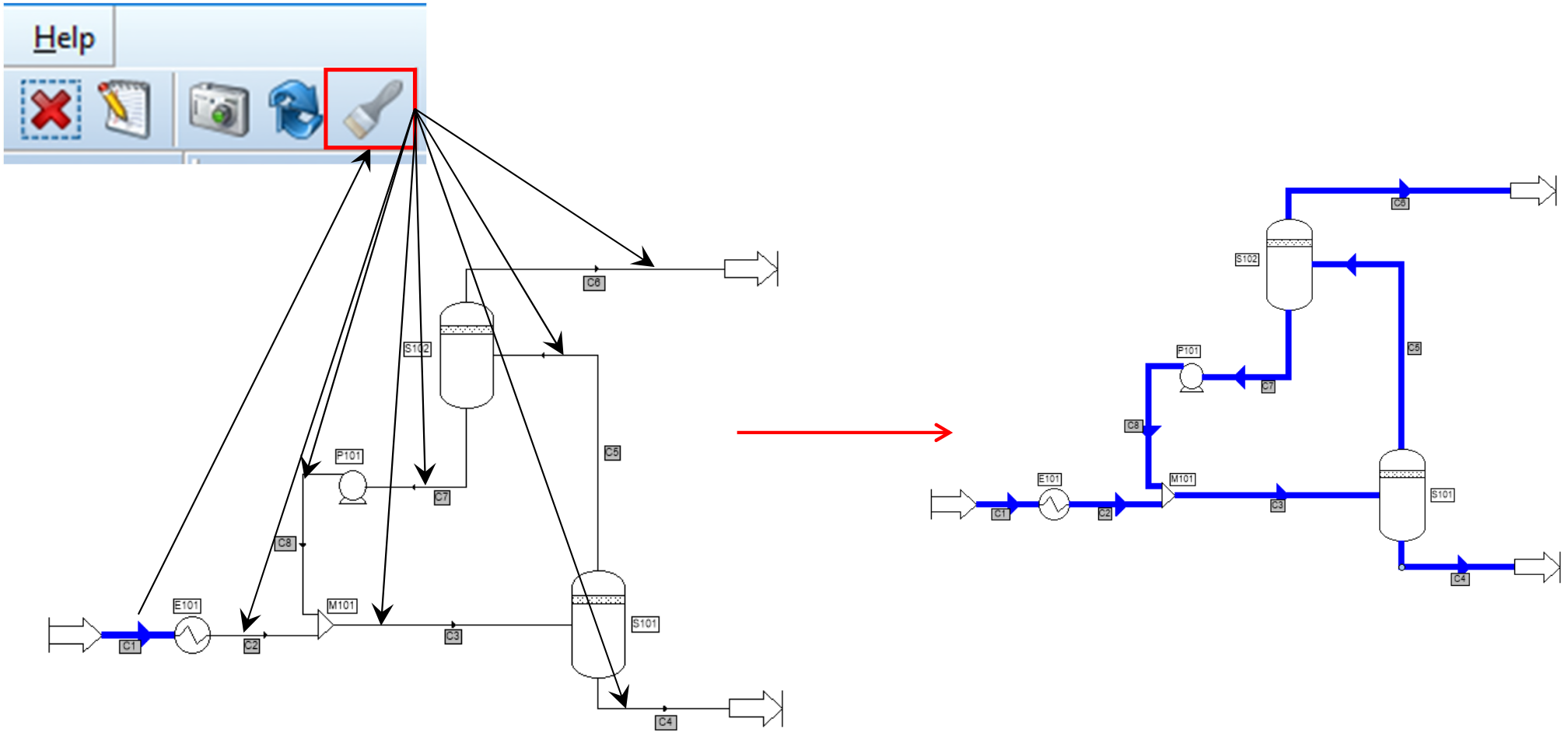
Inserting predefined shapes

Inserting images (logo…)

STEP #7: Graphical User Interface

You can reproduce the graphical aspect of the selected item (stream, label, form) on other elements

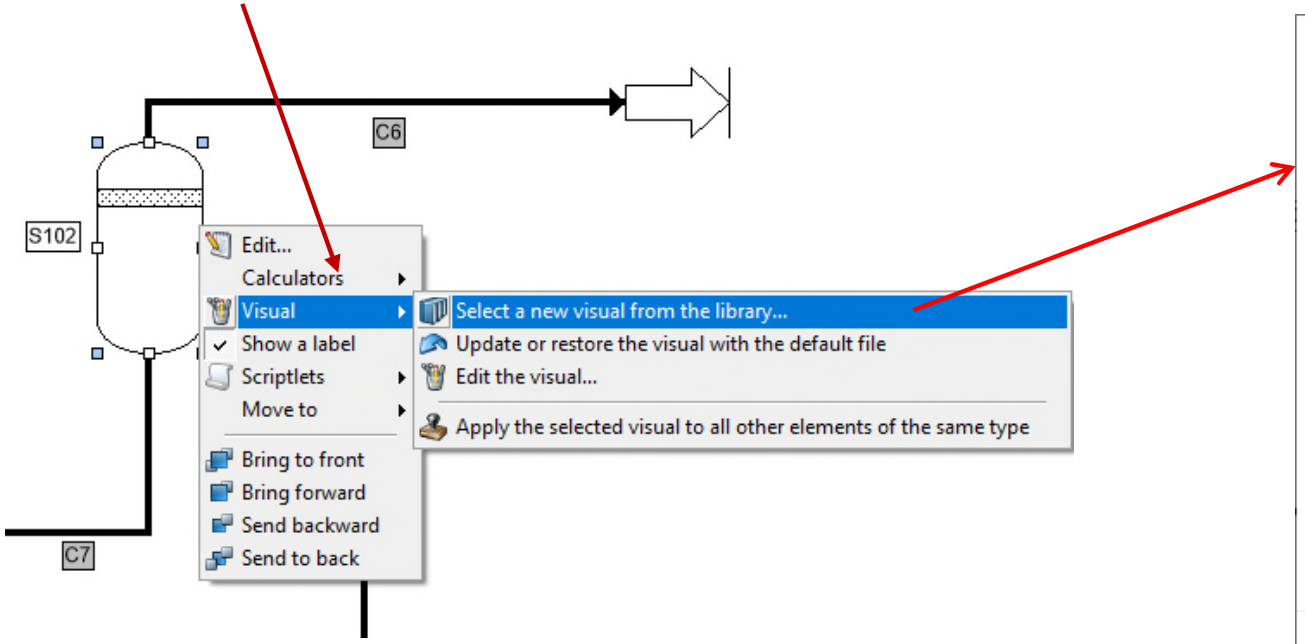
1. Click on the element from which you want to reproduce the graphical aspect
2. Click on the “Paint brush” icon
3. Click on the element for which you want to change the graphical aspect

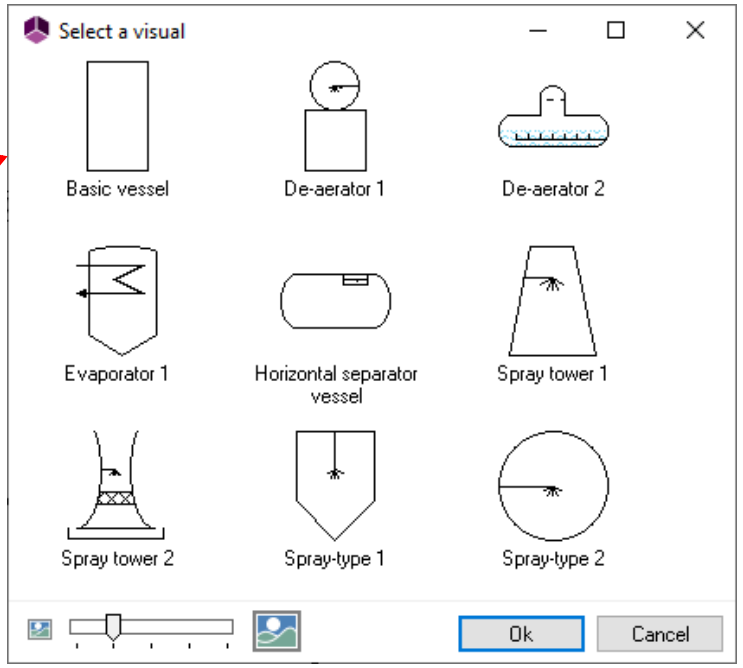


STEP #7: Graphical User Interface

You can modify the icon by another picture and modify the connection ports

1. Done a right click on the module, then, select “Visual”



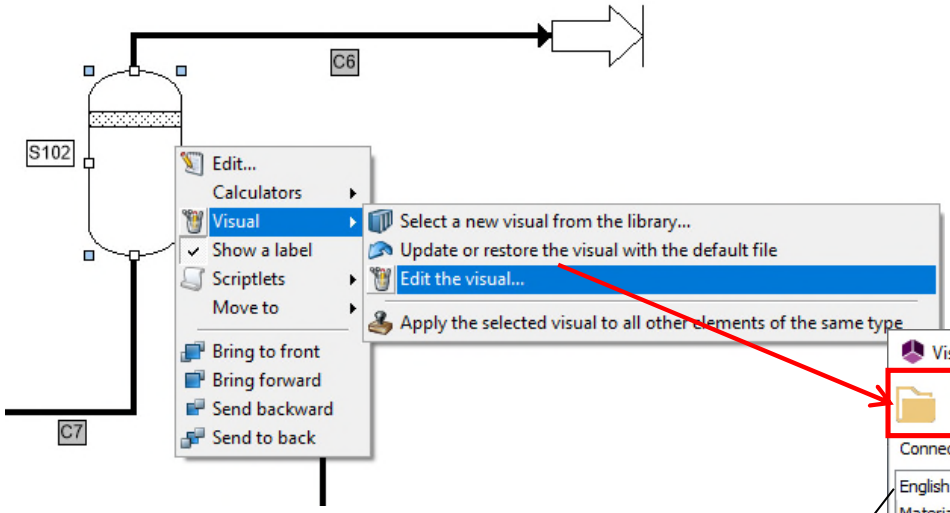


2. You can:

- **Select a new visual from the library: possibility to choose a visual in the library of visuals**
- **Update or restore the visual with the default file**
- **Edit the visual: possibility to create your visuals and to save them in the library to be able to reuse them**
- **Apply the selected visual to all other elements of the same type**

STEP #7: Graphical User Interface

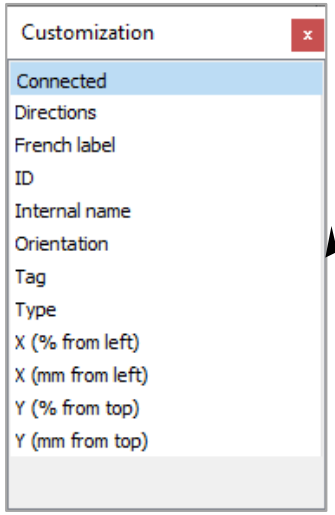
Visual: you can modify the connection ports (right click on the module)

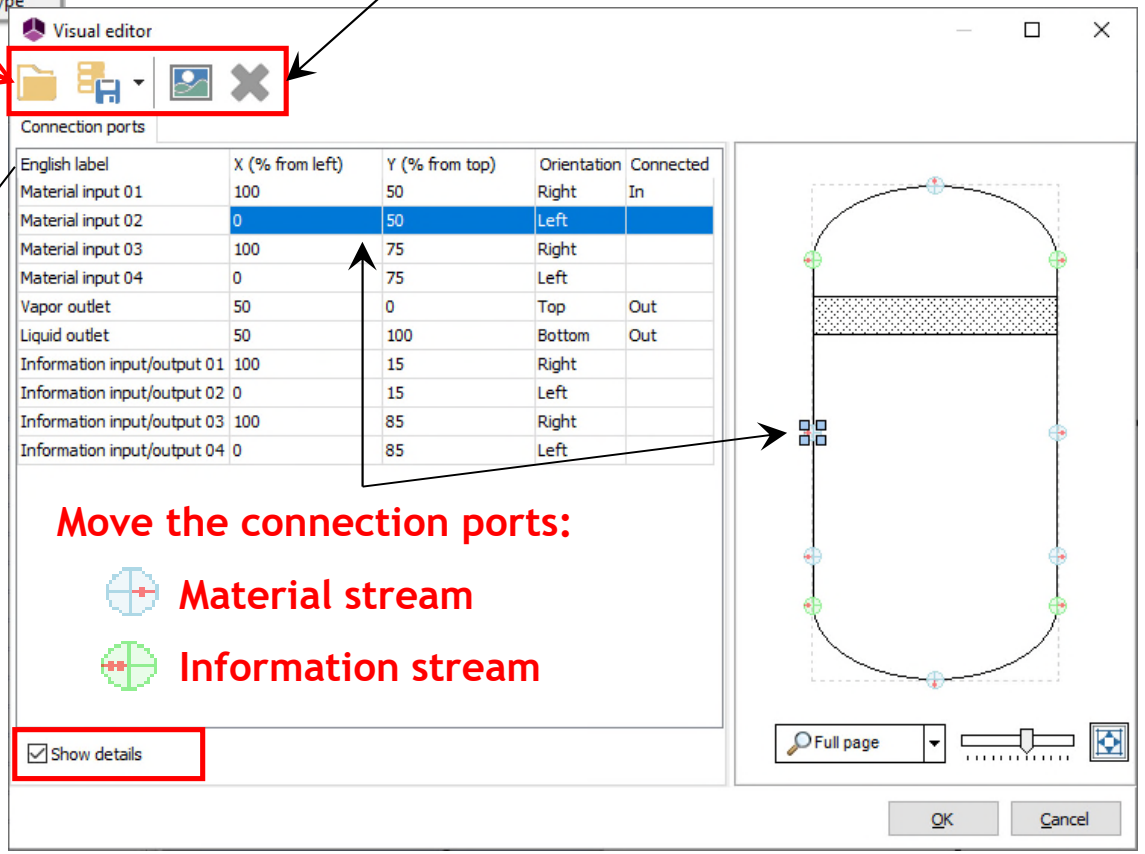


You can:

- Load a visual (*.puox file)
- Save a visual
- Load a picture
- Retore the original visual

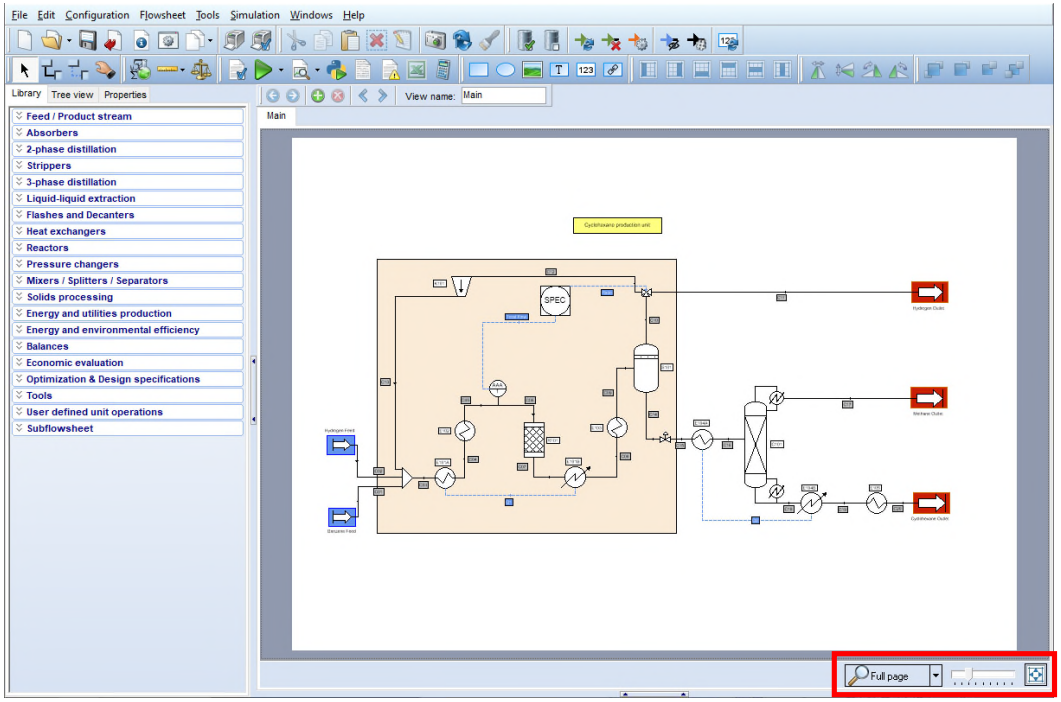
Right-click

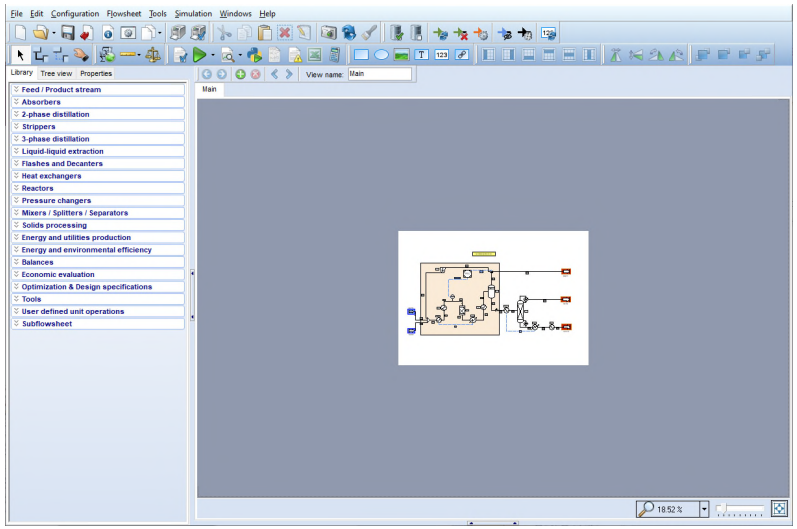




STEP #7: Graphical User Interface

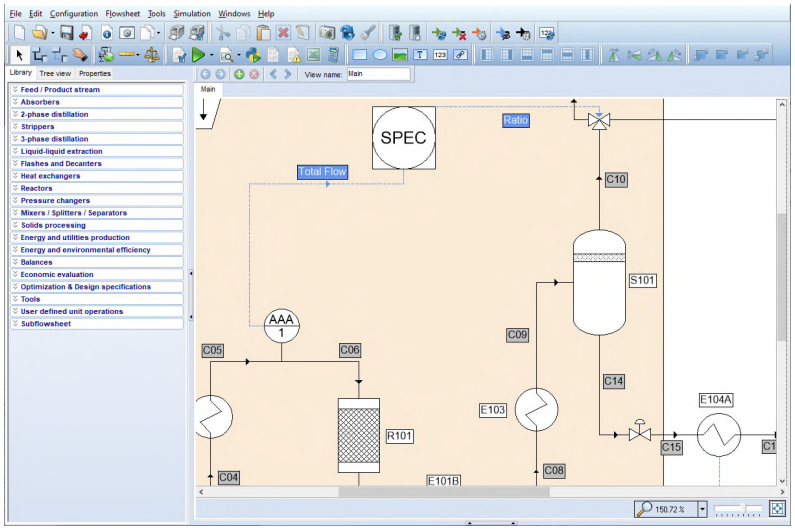
You can create zoom a flowsheet





CTRL + mouse wheel

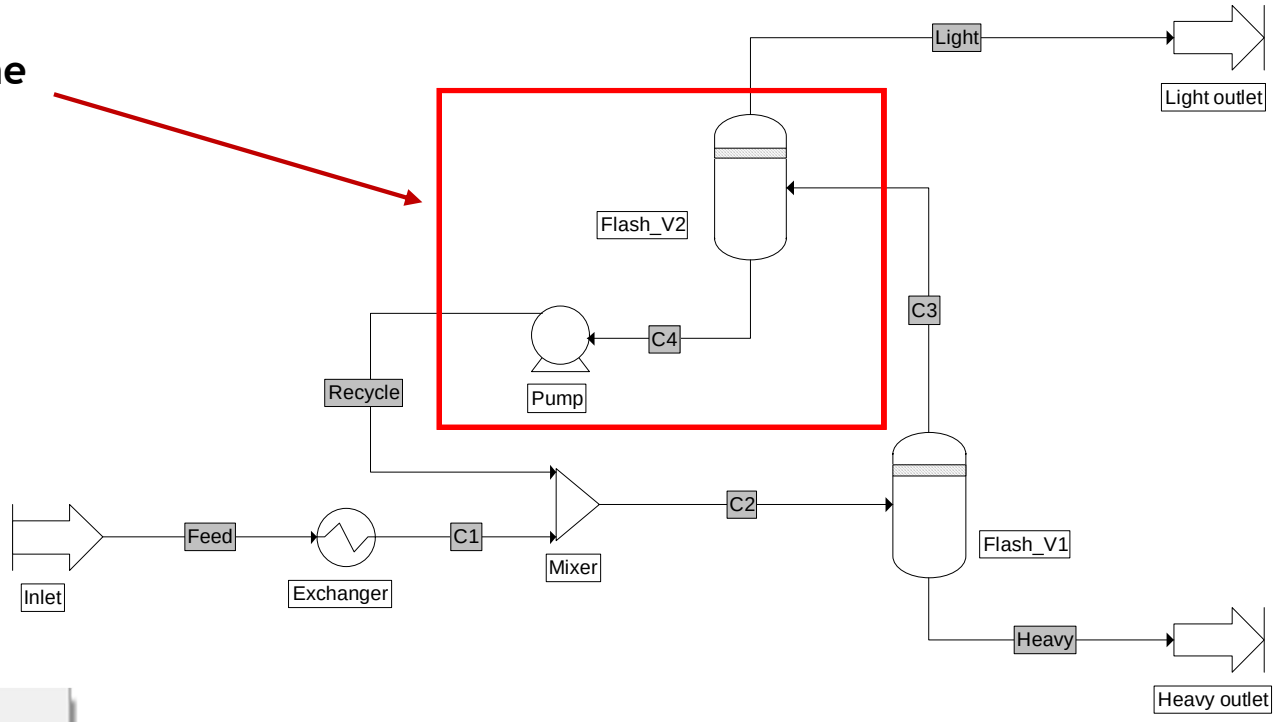
\- +

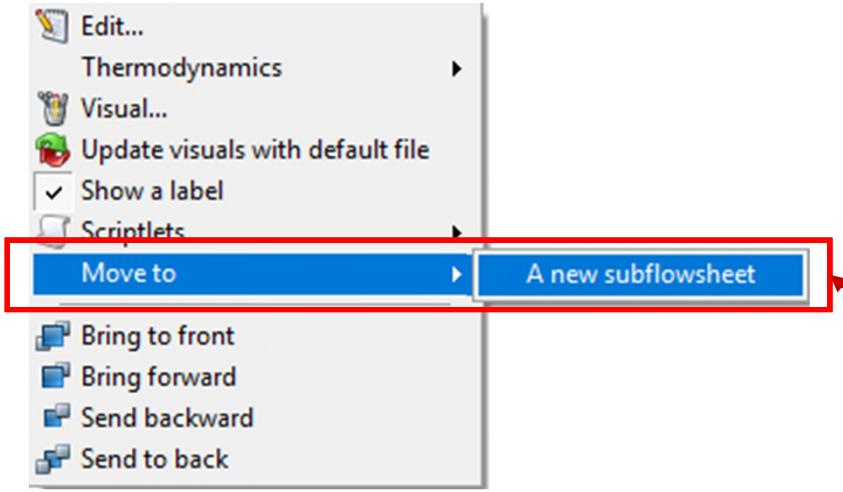


STEP #7: Graphical User Interface

Organization of the PFD: You can create subflowsheets

1. Select the modules to include in the subflowsheet

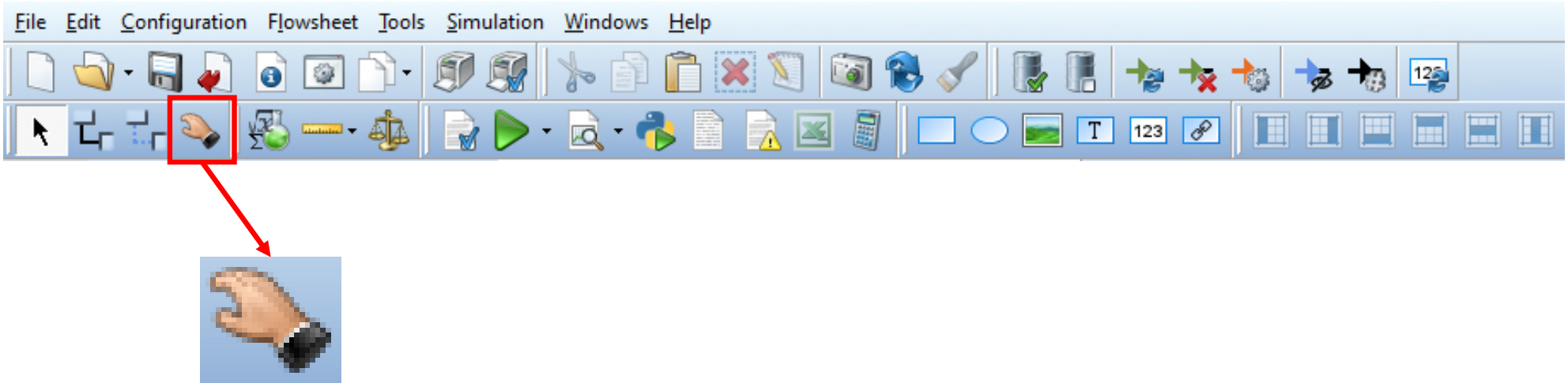




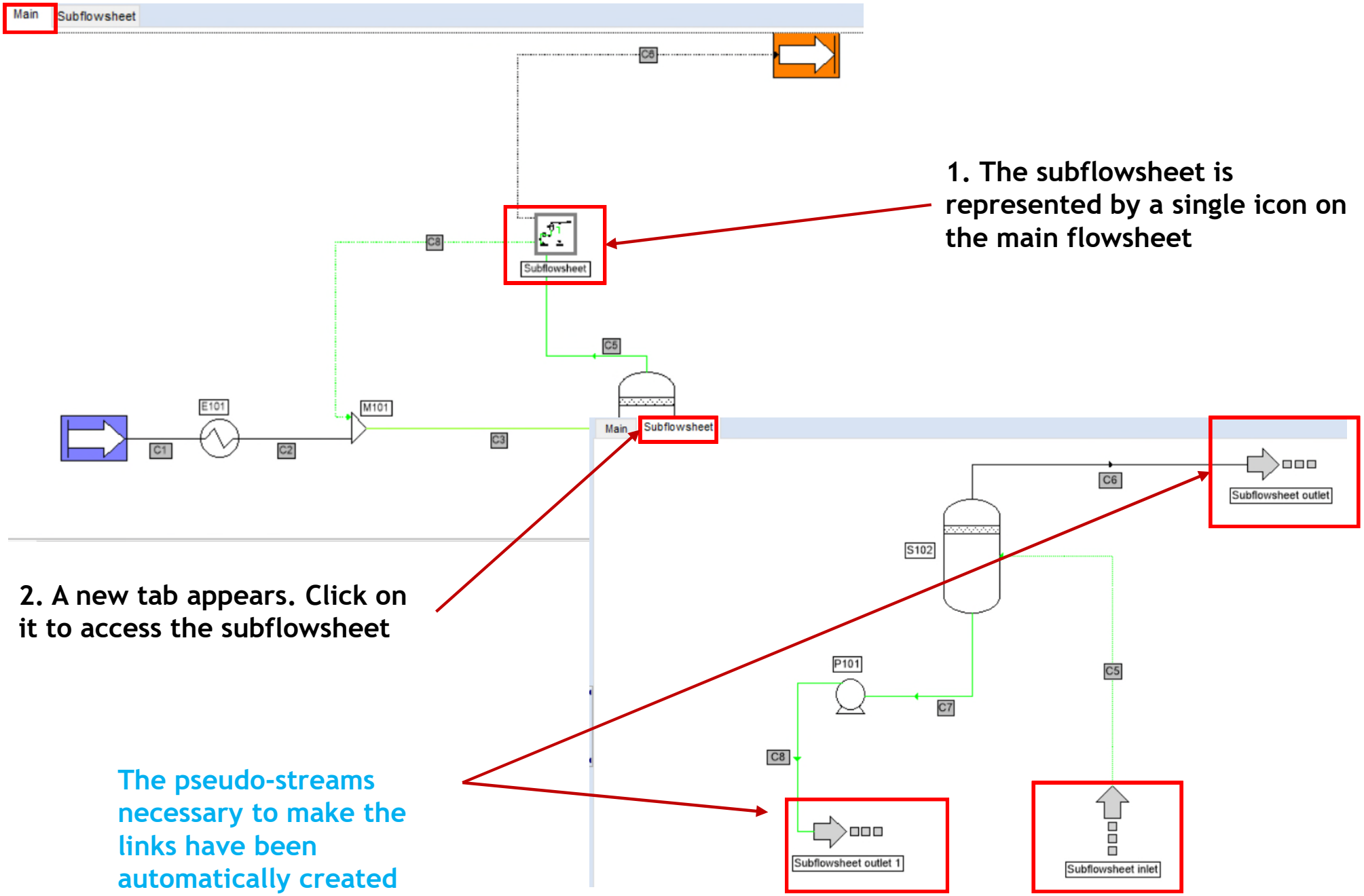
2. Do a Right click and select "Move to", then select "A new subflowsheet"

STEP #7: Graphical User Interface

Once zoomed you can move a flowsheet



STEP #7: Graphical User Interface



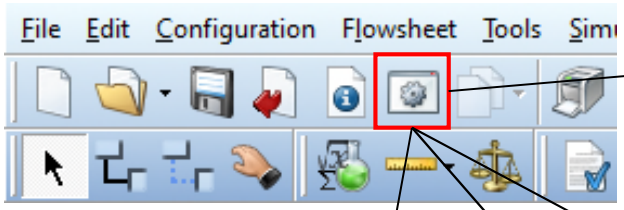
1. The subflowsheet is represented by a single icon on the main flowsheet

2. A new tab appears. Click on it to access the subflowsheet

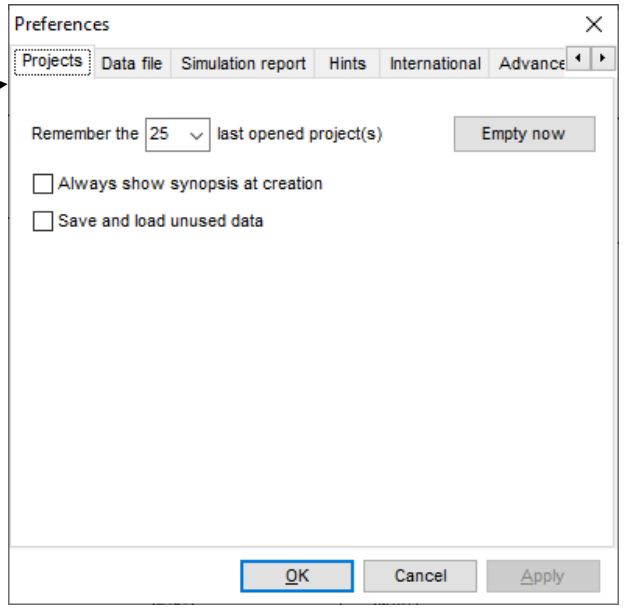
The pseudo-streams necessary to make the links have been automatically created

STEP #7: Graphical User Interface

Preferences

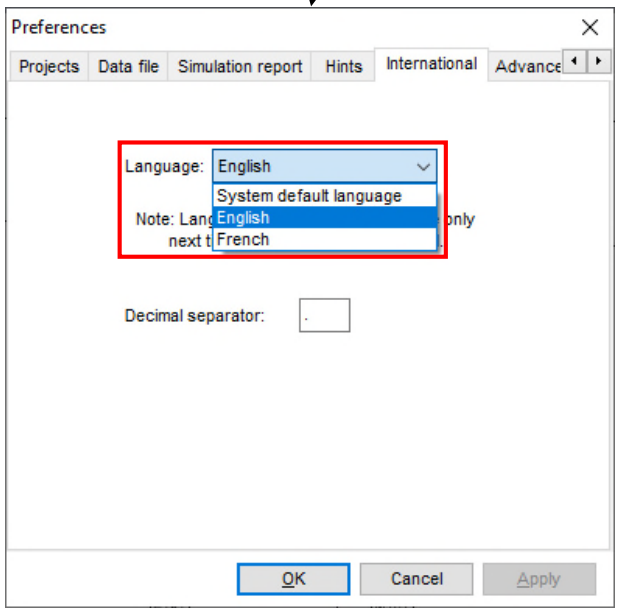


Projects

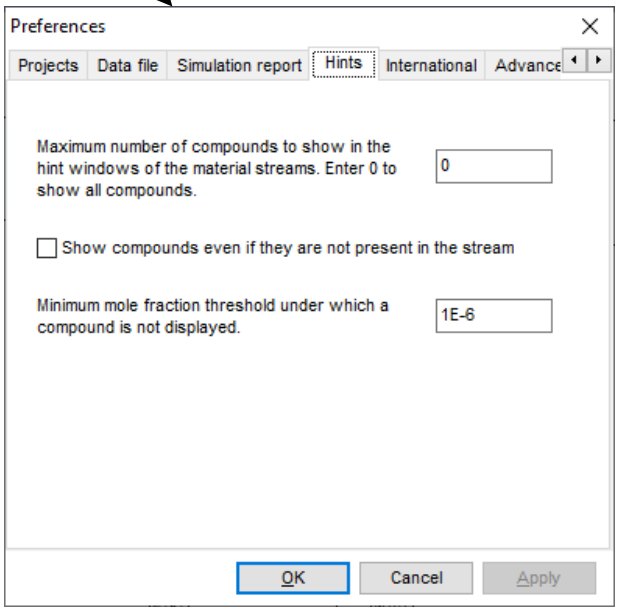


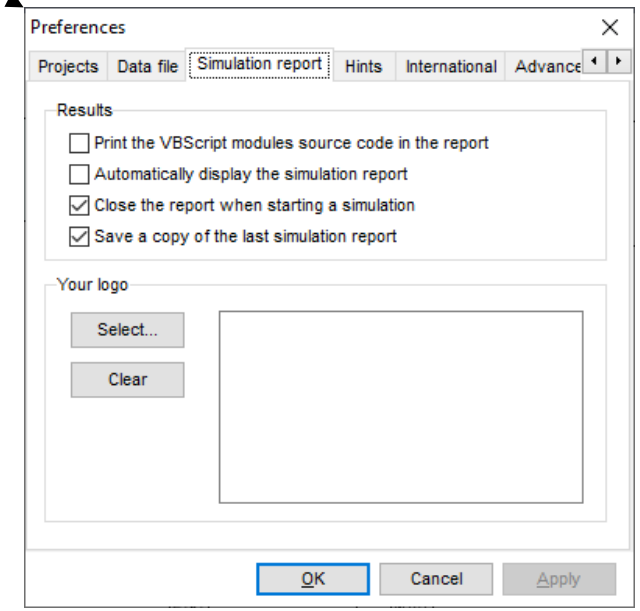
- Automatically display the simulation report
- Add your logo in the simulation report

Language

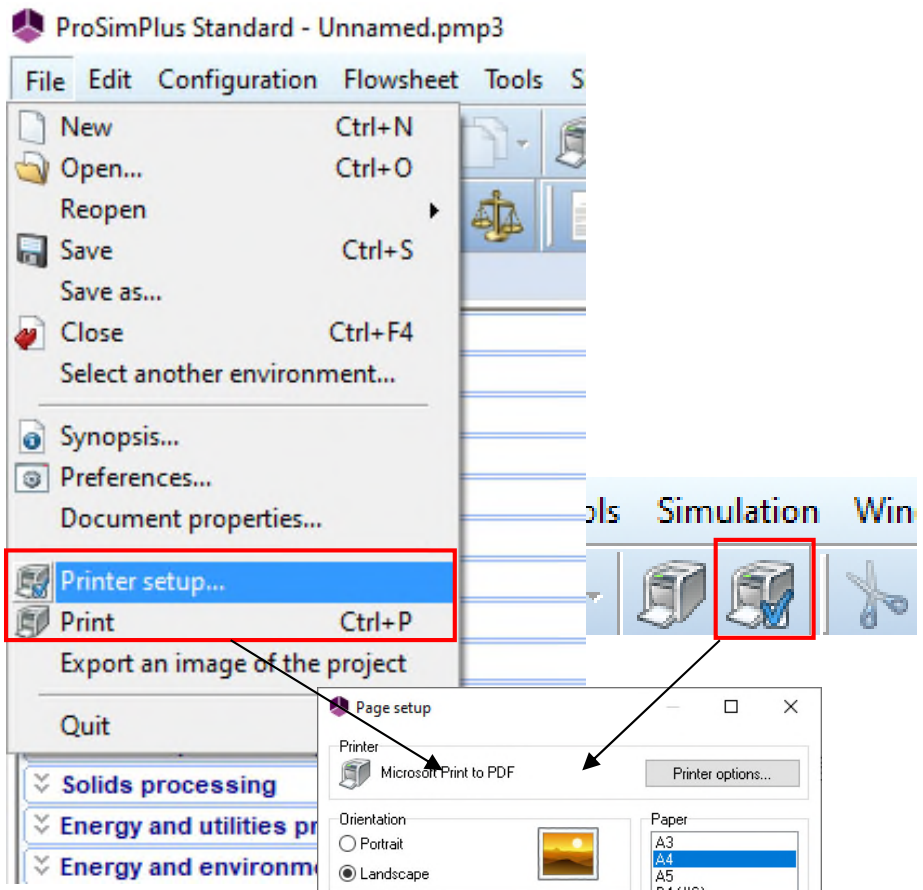


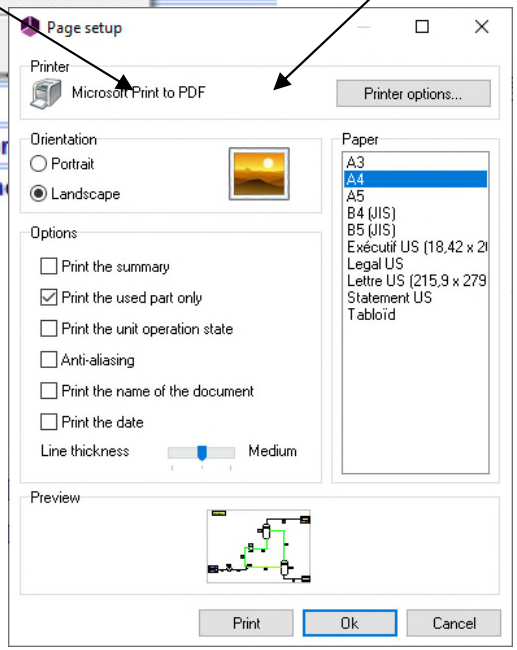
Hints



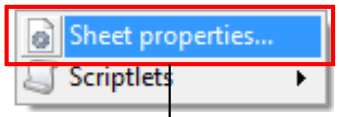


STEP #7: Graphical User Interface

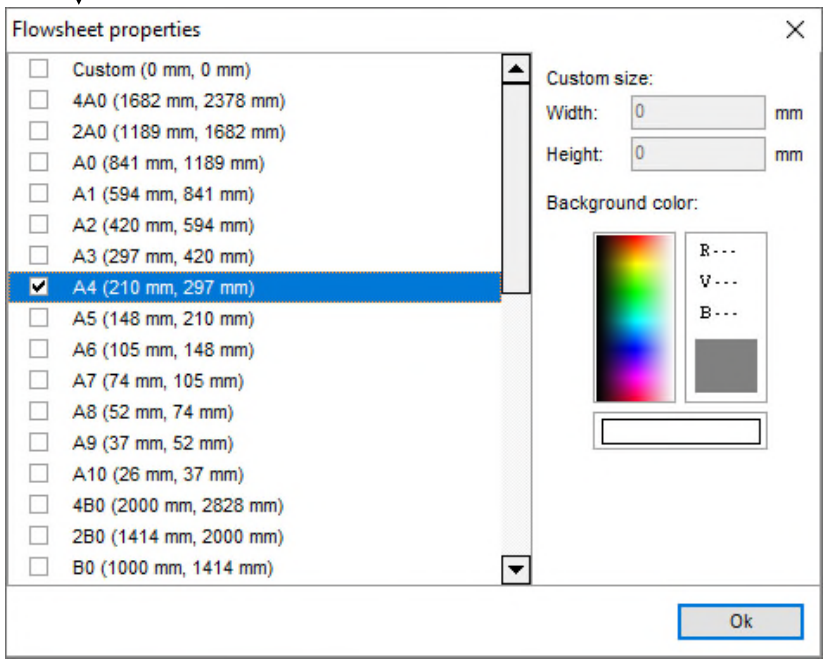




Right-click on the flowsheet



Size and background color of the PFD



Flowsheet printing

ProSim

Software & Services In Process Simulation

ProSim SA
51, rue Ampère
Immeuble Stratège A
F-31670 Labège
France

☎: +33 (0) 5 62 88 24 30

www.prosim.net
info@prosim.net

ProSim, Inc.
325 Chestnut Street, Suite 800
Philadelphia, PA 19106
U.S.A.

☎: +1 215 600 3759