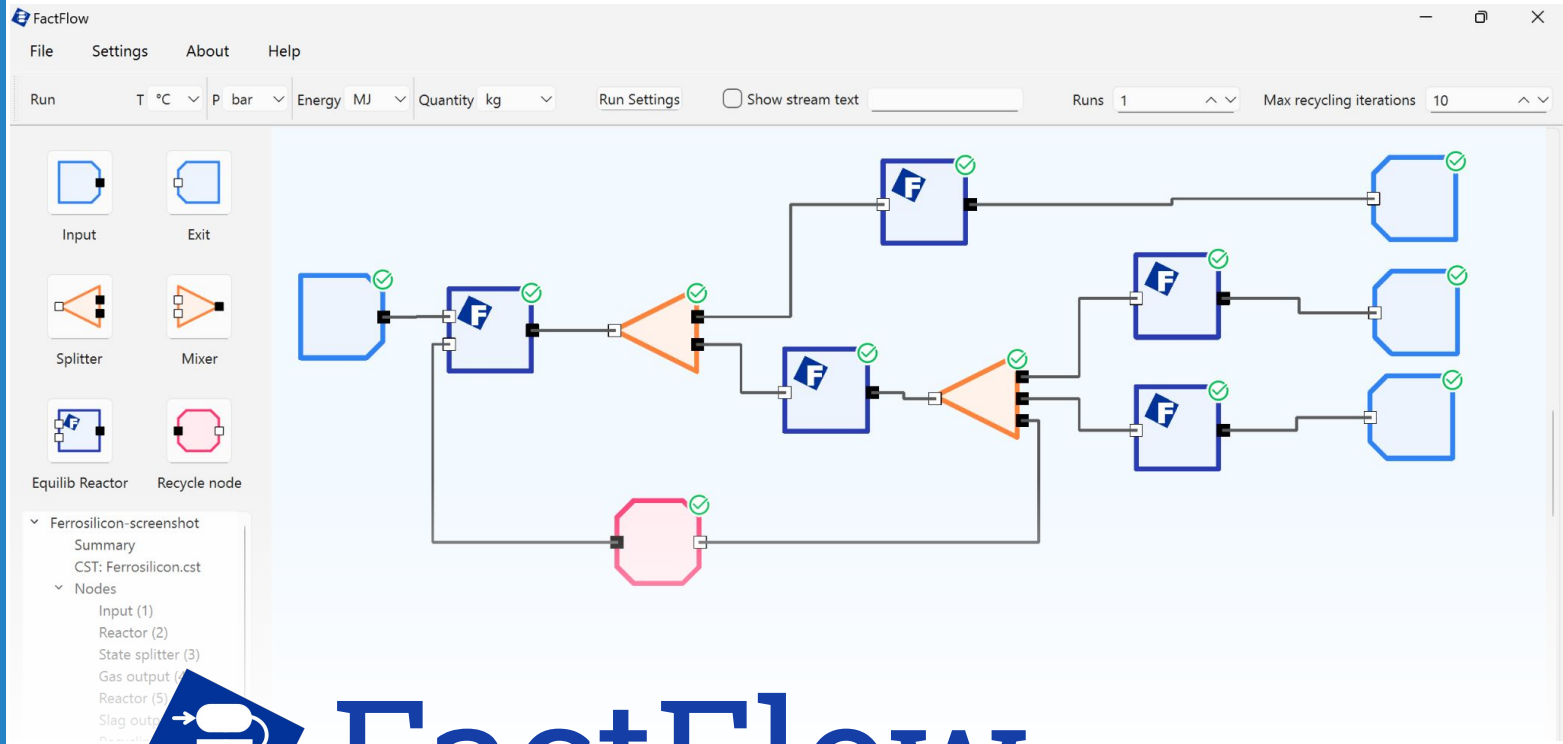
FactFlow

User manual

Powered by

FactSage™

ChemApp™



FactFlow User manual

What is FactFlow?

FactFlow is FactSage's new **process simulation and optimization** interface.

Key features include:

- **Powered by FactSage and ChemApp**: FactFlow employs the same thermodynamic equilibrium calculation algorithms and databases as FactSage and ChemApp, offering unparalleled accuracy.
- **Intuitive User Interface**: Designed to be user-friendly, FactFlow features a drag and drop flowsheet-based interface for process simulation.
- **Flexible Simulation Capabilities**: FactFlow supports a wide range of applications, from basic equilibrium calculations performed in series to complex process optimizations, steady-state simulations and parametric studies.
- **Comprehensive Analysis Tools**: FactFlow is equipped with extensive analysis and visualization tools, enabling users to experiment with different input parameters and allowing in-depth examination of results.
- **Long-term continuous development and support**: FactFlow is backed by a dedicated development team, ensuring regular updates, feature enhancements, and user support.

Powered by

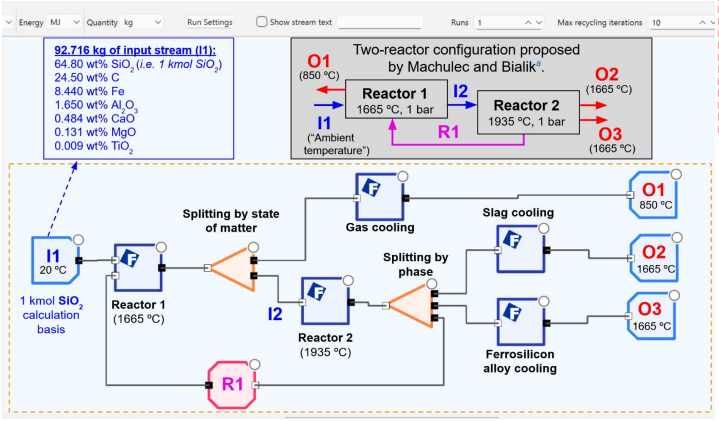
FactSage™

ChemApp™

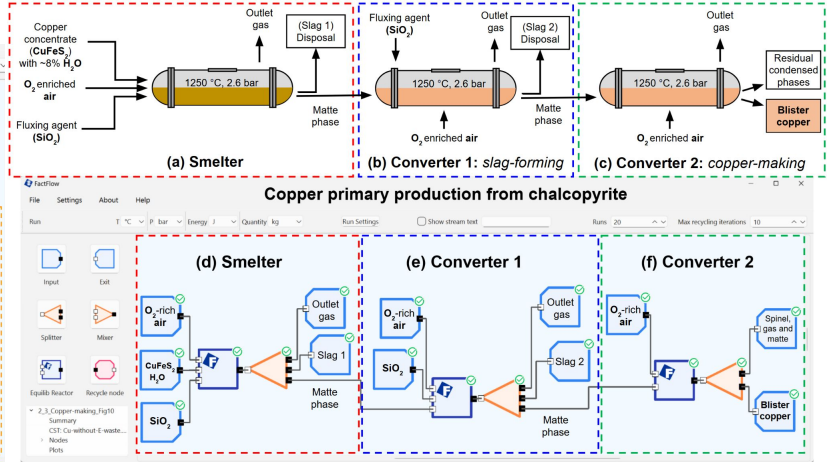
What is FactFlow?

FactFlow has already been employed to model various processes, and has been used by industry partners around the world.

We have published a paper titled **Pyrometallurgical process modeling using FactFlow** in the *Calphad* journal that demonstrates FactFlow's capabilities through 4 examples of pyrometallurgical processes:



Ferrosilicon alloy production



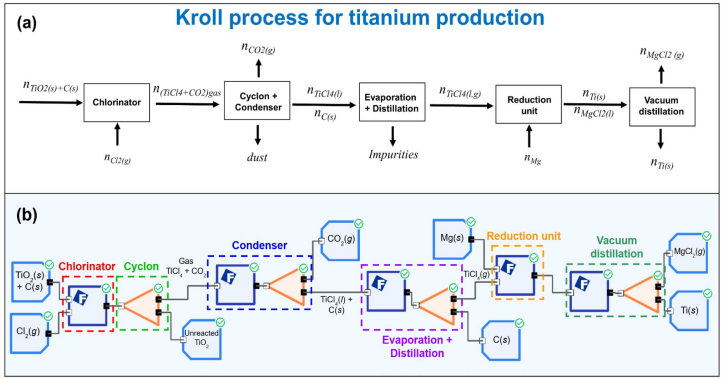
Copper primary production

https://doi.org/10.1016/j.calphad.2024.102772

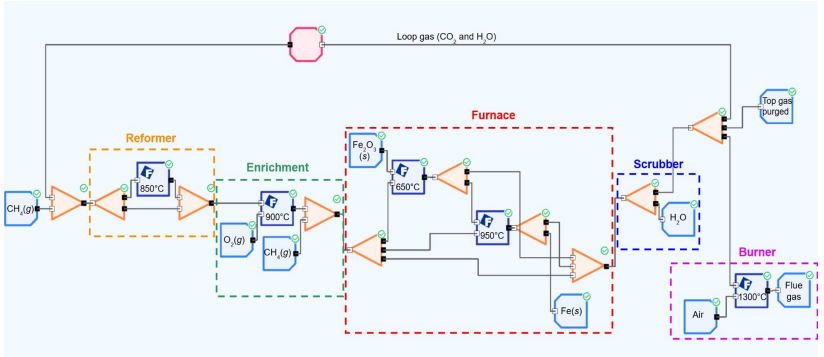
What is FactFlow?

FactFlow has already been employed to model various processes, and has been used by industry partners around the world.

We have published a paper titled **Pyrometallurgical process modeling using FactFlow** in the *Calphad* journal that demonstrates FactFlow's capabilities through 4 examples of pyrometallurgical processes:



Titanium primary production



MIDREX direct iron reduction process

https://doi.org/10.1016/j.calphad.2024.102772

FactFlow

User manual

Powered by

FactSage™

ChemApp™

About This Manual

This manual is designed to help you learn how to use FactFlow effectively, whether you're new or looking to explore more advanced features.

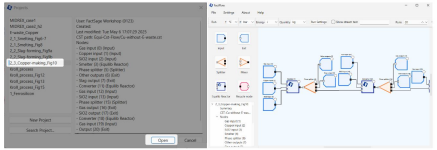
We strongly recommend beginning with the **Getting Started** section. It’s a guided walkthrough that shows you how to **reproduce a real figure** from the FactFlow paper, step by step.

You'll follow along directly in your own FactFlow installation and learn how to:

- Launch FactFlow
- Open an example project
- Navigate the interface
- Run a calculation
- View and plot simulation results

This hands-on approach is the fastest way to become familiar with FactFlow’s core workflow.

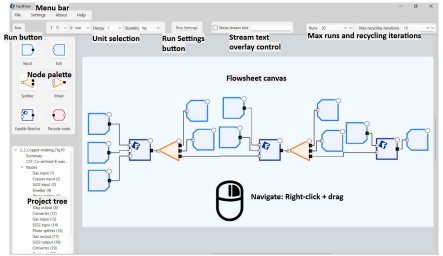
Getting Started - Open an example



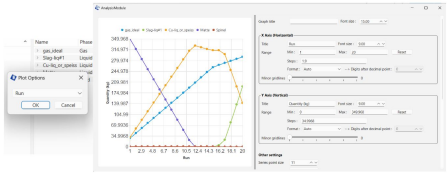
FactFlow examples are a great way to learn how the software works and explore its capabilities. To open an example, select it from the list and click **Open** in the bottom-right corner.

Try opening the **2_3_Copper-making_Fig10** example. This example reproduces Figure 10 from Section 3.2.3 of the FactFlow paper. In the following steps, we’ll walk through how to generate the plot shown in that figure.

Getting Started - Main interface



Getting Started - Plot the results



In the **Plot Options** dialog, simply **click OK**. This will launch the **Analysis Module**, FactFlow’s dedicated environment for advanced analysis and visualization. Plot settings and customization options will be covered later in this guide.

Powered by

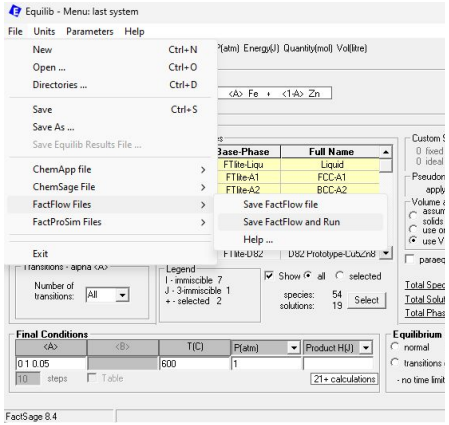
FactSage™

ChemApp™

About This Manual

What Comes Next - Creating your own Process Simulation

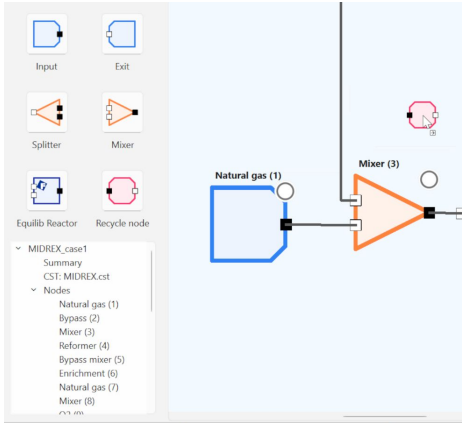
To create your own simulation, you'll follow a multi-step process.
Each of these steps is explained in detail in this manual.



Step 1

Prepare your Database

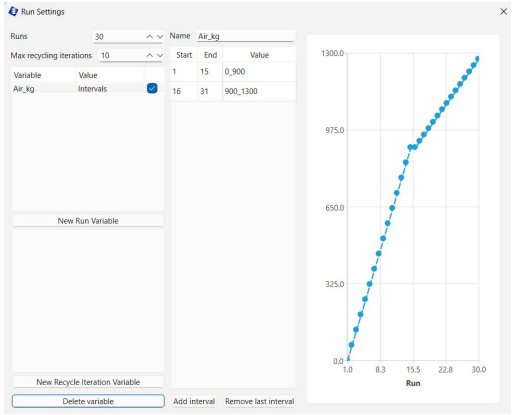
Use FactSage's Equilib module to perform the species and phase selection for your flowsheet's mini database.



Step 2

Build the Process Flowsheet

Drag and connect nodes to model your process: inputs, reactors, splitters, recycling loops, etc.



Step 3

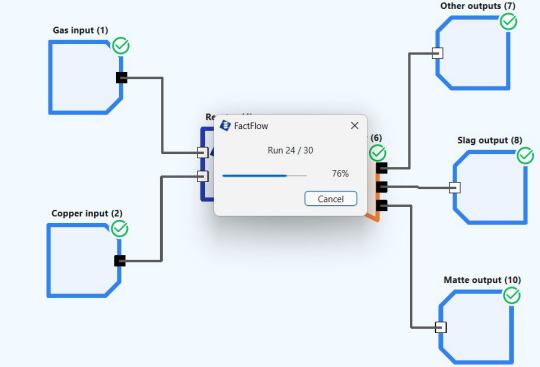
Configure Run Settings

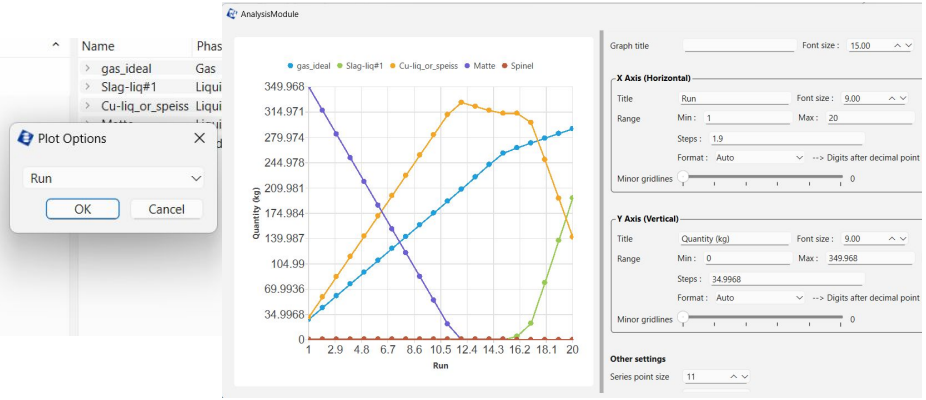
Define how many simulations to run and how input conditions should vary (e.g., temperature, quantity of reactants).

About This Manual

What Comes Next - Creating your own Process Simulation

To create your own simulation, you'll follow a multi-step process.
Each of these steps is explained in detail in this manual.





Step 4

Run the Simulation

Launch the simulation—FactFlow will execute all runs, including parametric variations and recycle loops, progressing node by node until convergence.

Step 5

Analyze the Results

Access calculation results from any node, then use the Analysis Module to plot trends, compare runs, or examine elemental compositions. You can also export data and plots for further use.

Getting Started

Now that you've seen the overall structure of the manual, it's time to get **hands-on**.

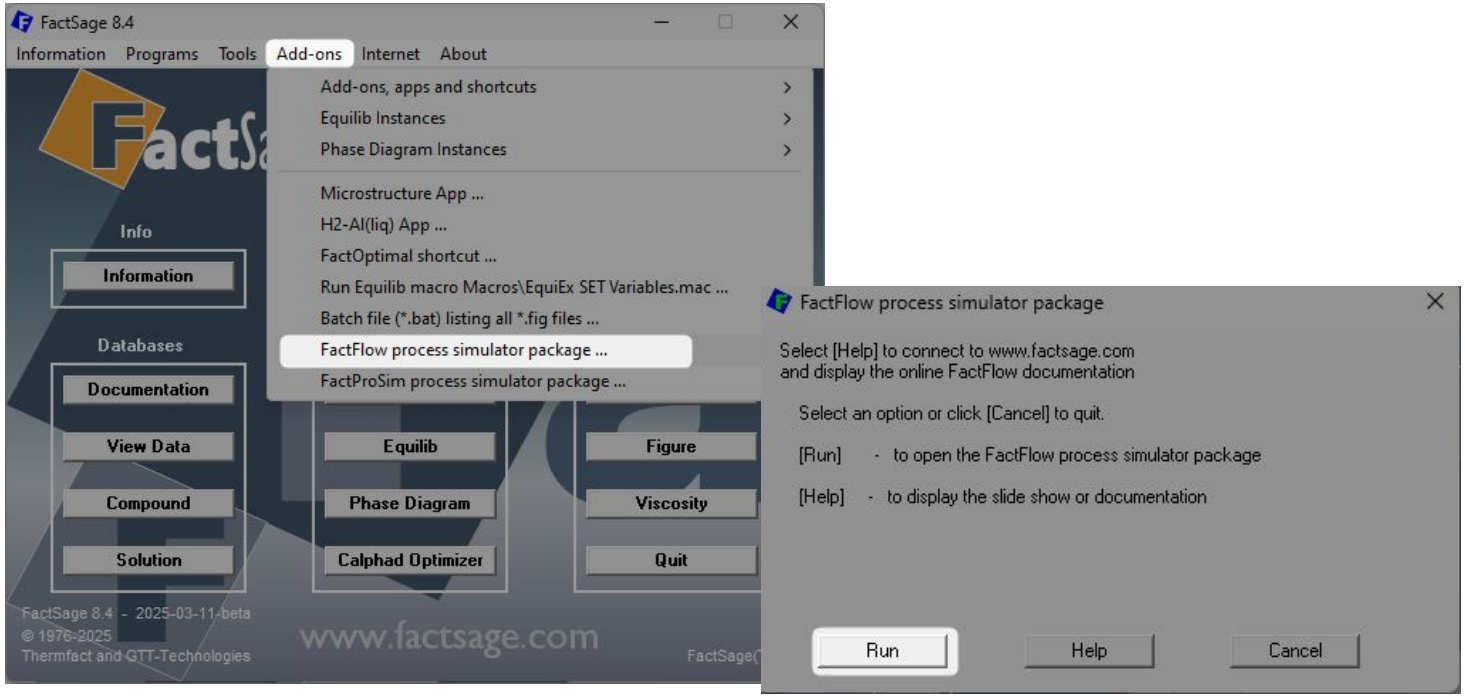
In this section, you'll reproduce a **real figure** from the FactFlow paper by following along with a **built-in example**.

Getting Started - System Requirements

Before launching FactFlow, make sure your system meets the following requirements:

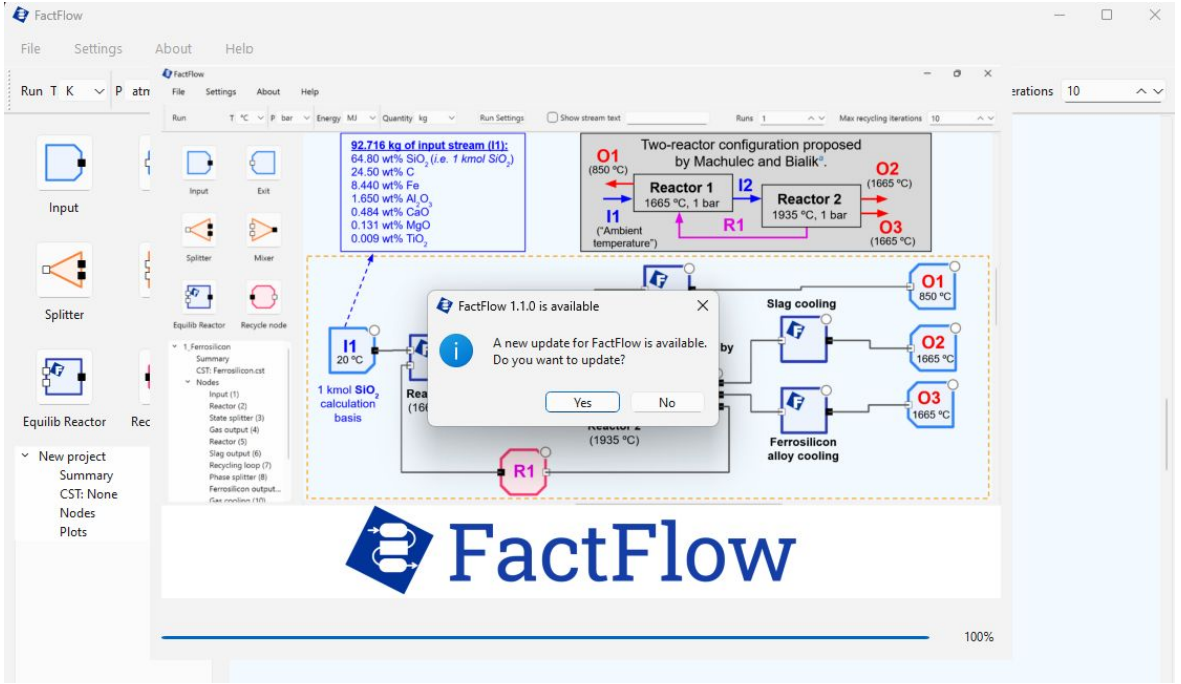
- **Operating System**: Windows 10/11 or later
- **RAM**: 8GB minimum, 16GB recommended
- **FactSage Requirement**: Valid license of FactSage 8.4 or later

Getting Started - Launching FactFlow



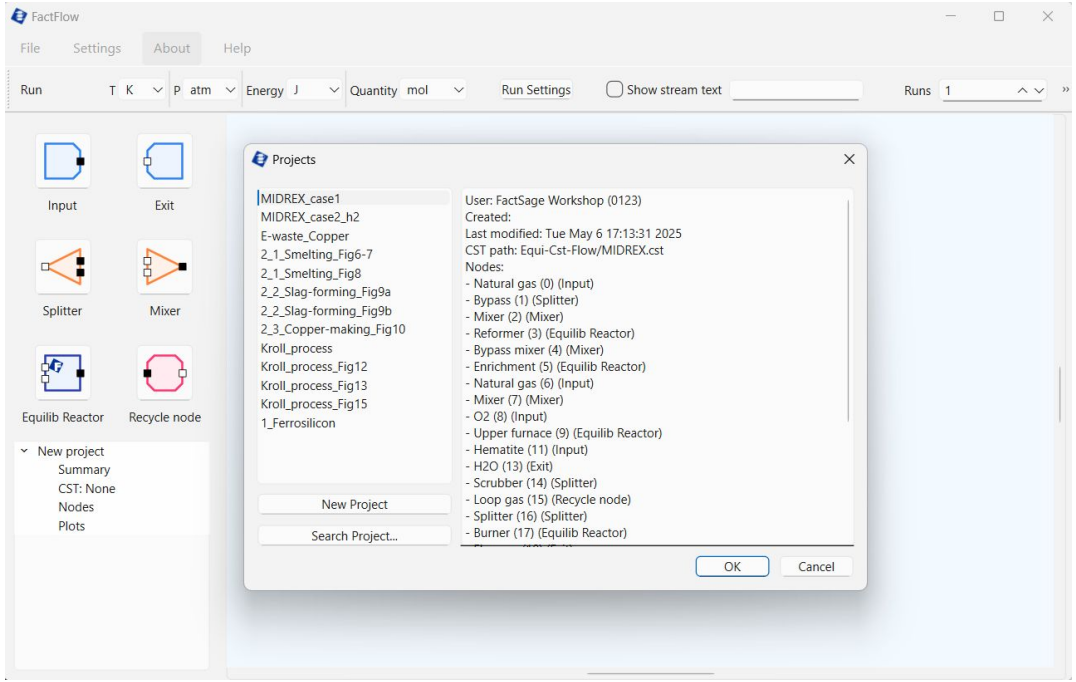
To launch FactFlow, open **FactSage**, then go to the **Add-ons** menu in the top toolbar and select **FactFlow process simulator package…** from the dropdown list. Click **Run** to launch FactFlow in the pop-up window.

Getting Started - Launching FactFlow



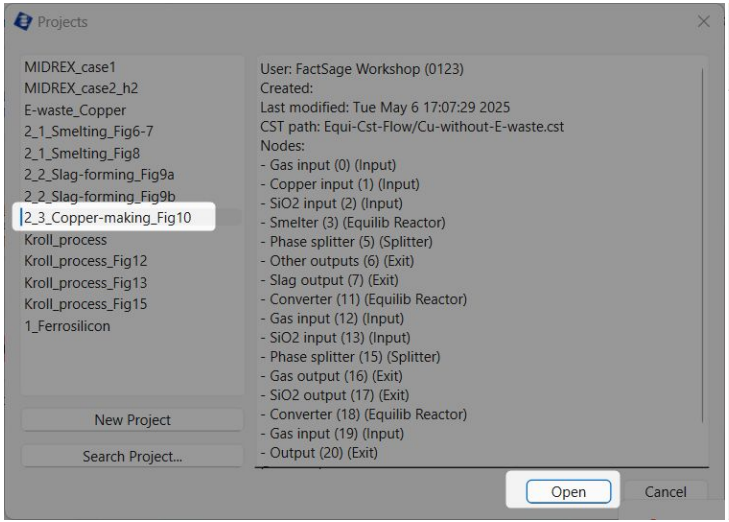
On launch, FactFlow may prompt you to install an available update. We **highly recommend** clicking **Yes**—the update will run automatically, and FactFlow will relaunch once it's complete.

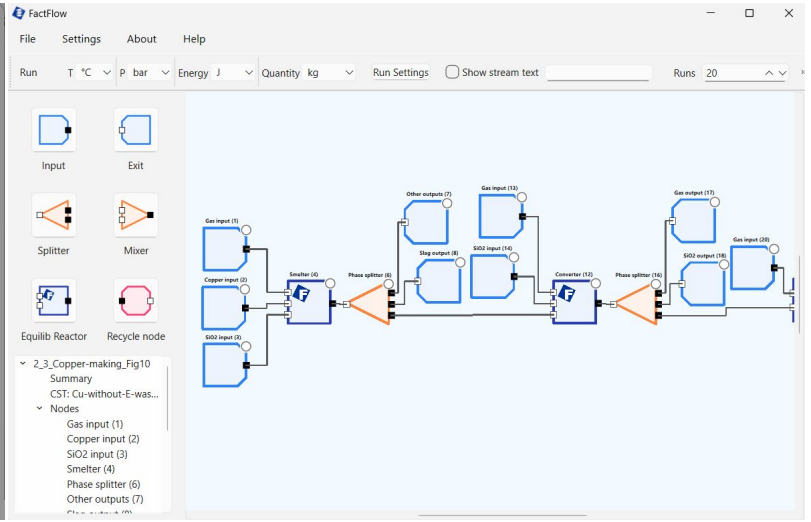
Getting Started - Projects window



Upon launch, FactFlow automatically opens the **Projects** window, where you can view, open, or create projects.

Getting Started - Open an example





FactFlow examples are a great way to learn how the software works and explore its capabilities. To open an example, select it from the list and click **Open** in the bottom-right corner.

Try opening the **2_3_Copper-making_Fig10** example. This example reproduces Figure 10 from Section 3.2.3 of the FactFlow paper. In the following steps, we'll walk through how to generate the plot shown in that figure.

FactFlow

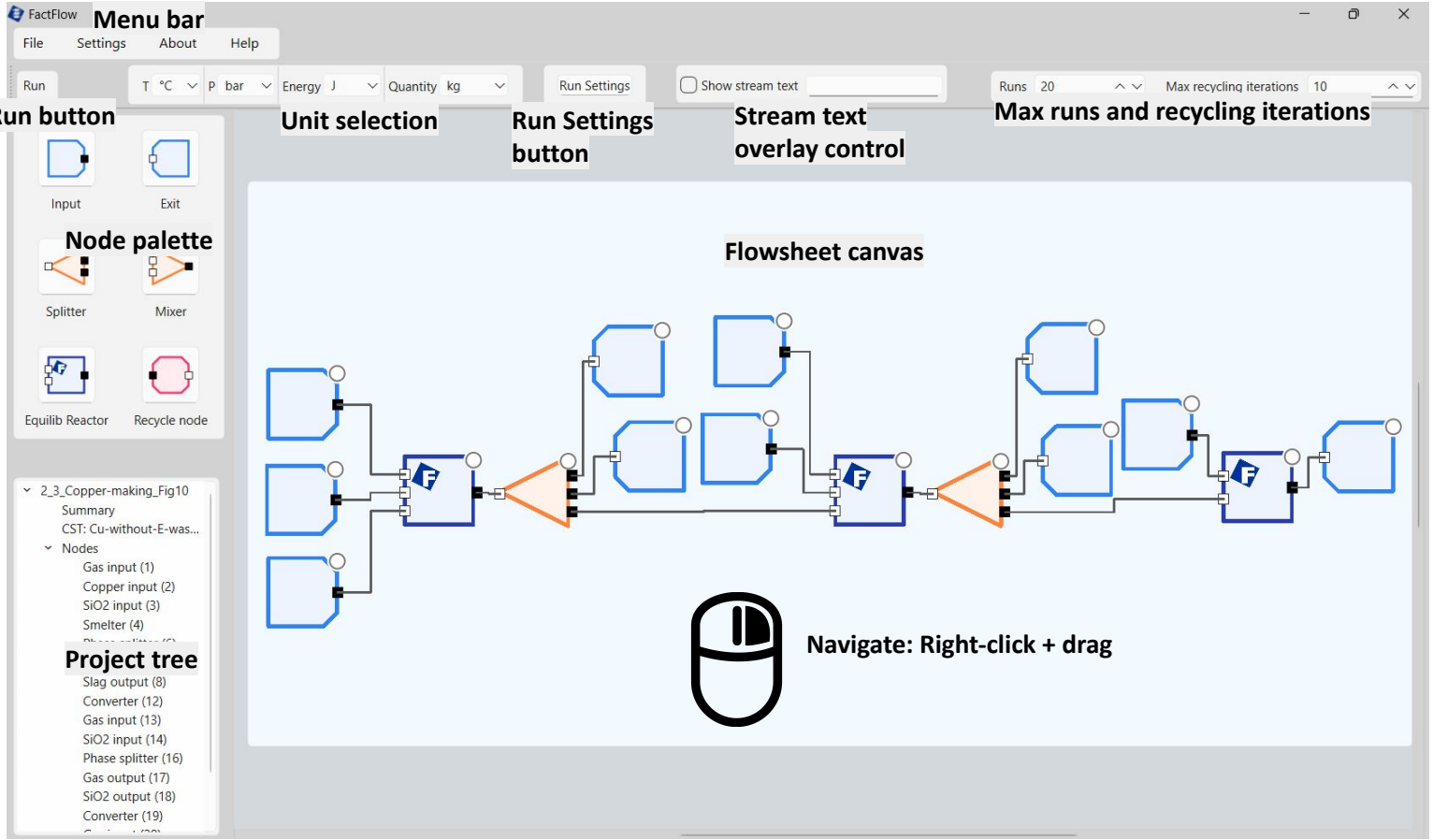
User manual

Powered by

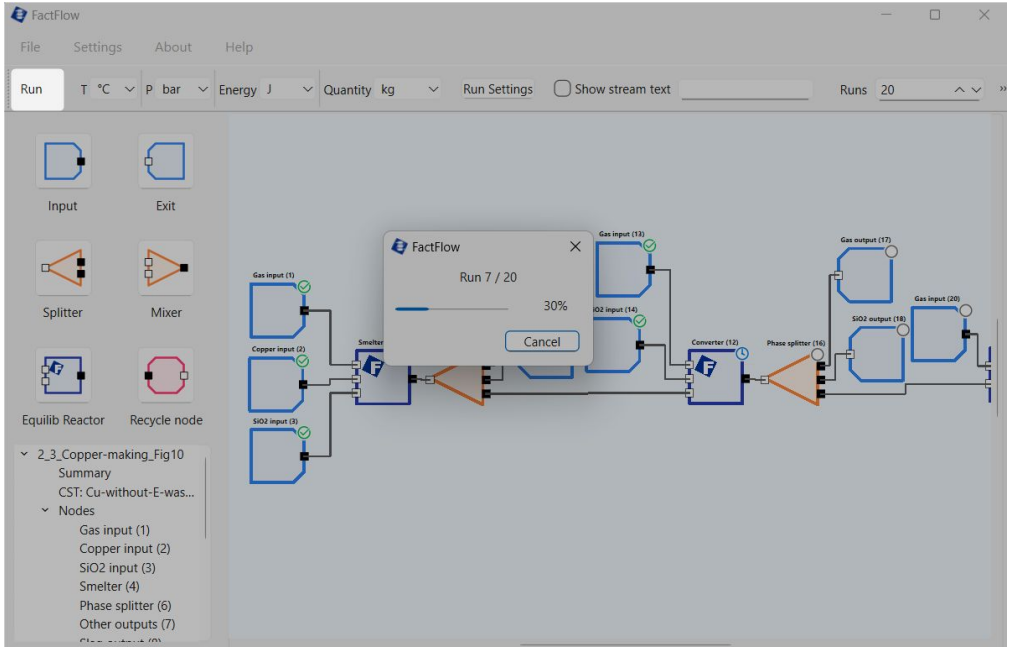
FactSage™

ChemApp™

Getting Started - Main interface

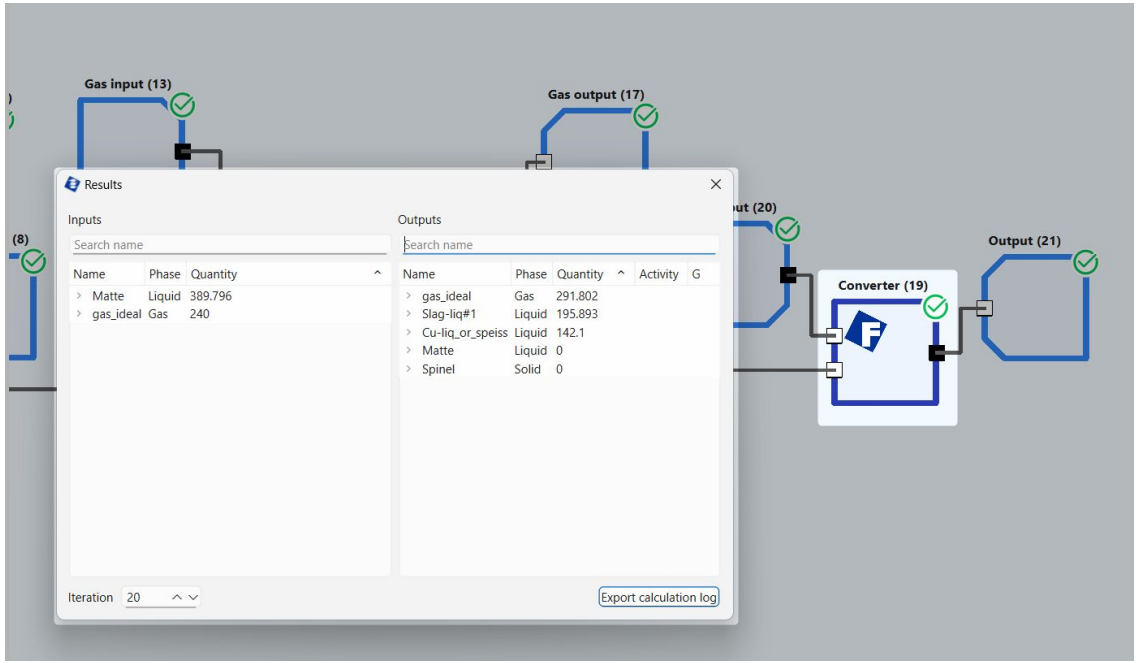


Getting Started - Run a calculation



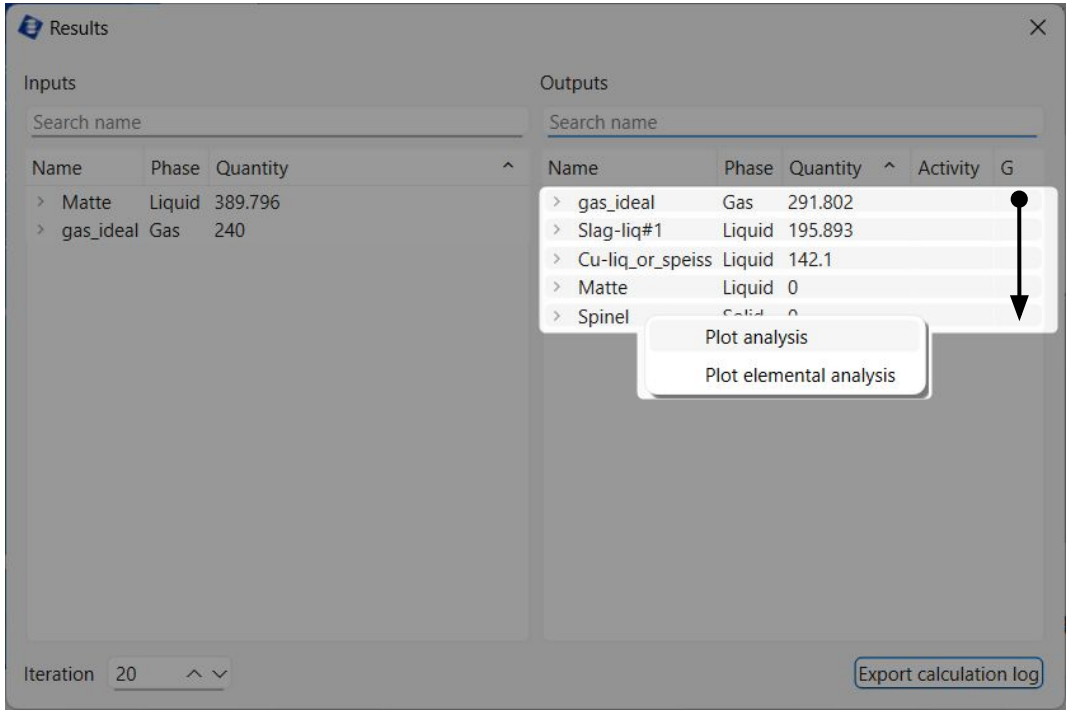
Let's run the example. Click the **Run** button at the top left and wait for the simulation to complete. This example is set up to perform **20 runs**, **varying the amount of O_2-enriched air input** in the second converter. You'll learn how to build your own flowsheet and configure runs later in the guide.

Getting Started - Plot the results



Once the calculation is complete, **right-click and drag to navigate** to the end of the flowsheet on the right. There, you'll find the **second converter**, modeled as an **Equilib Reactor node** labeled **Converter (19)**. **Double-click** this node to open its **Results** window.

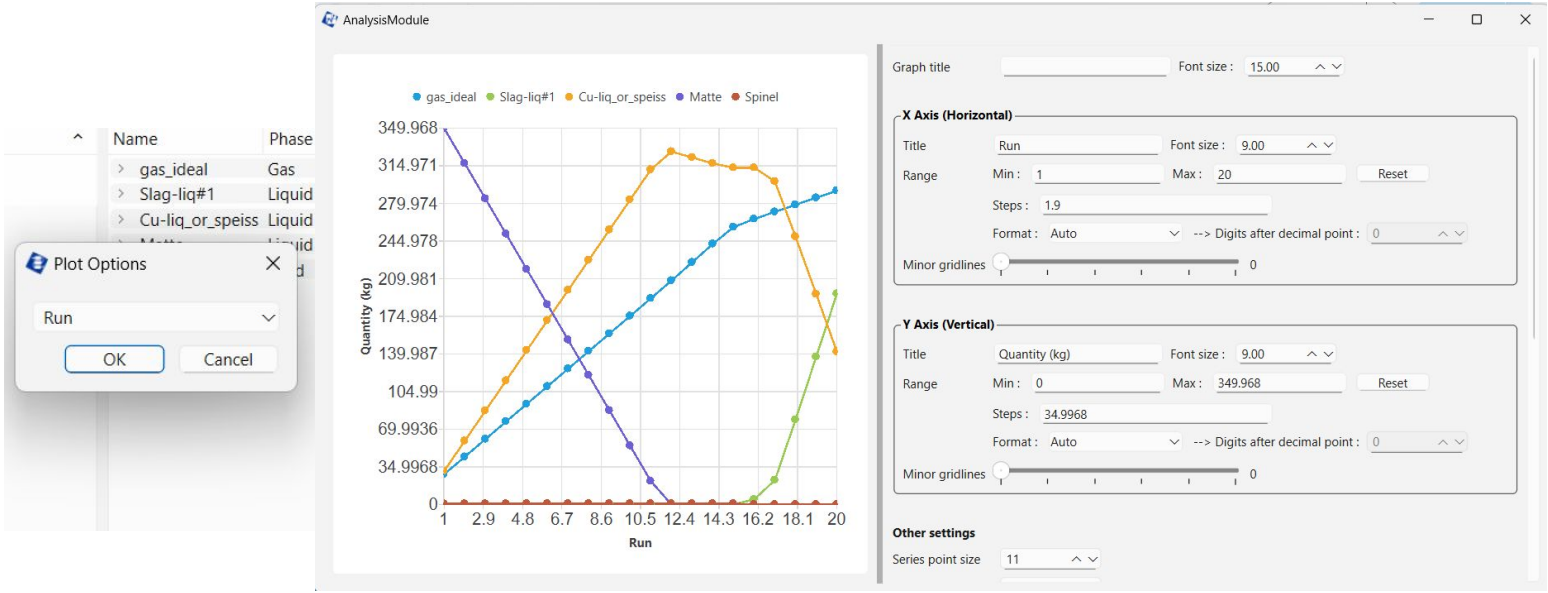
Getting Started - Plot the results



Multi-select: Left-click + drag

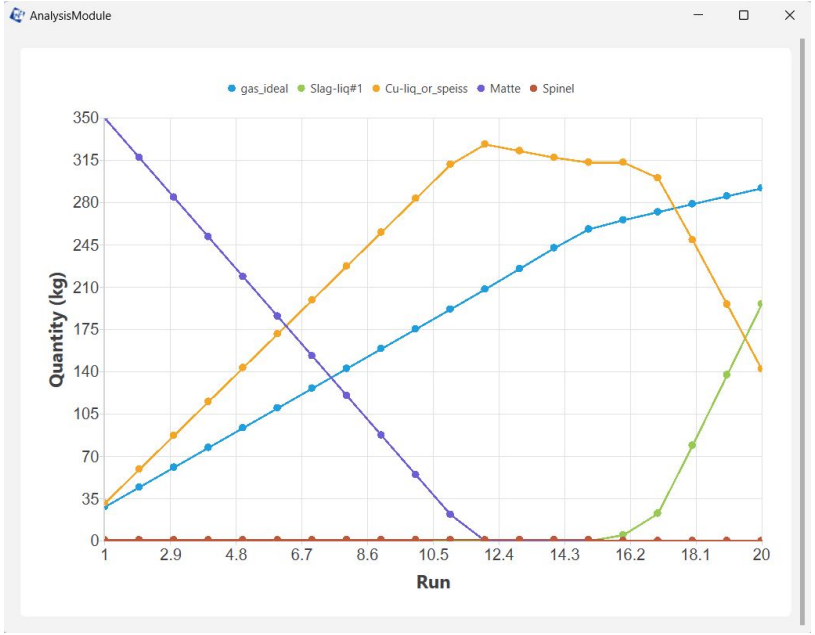
To select all phases in the **Outputs** section, click the first phase, then **hold left-click and drag down to the last one**. Alternatively, you can select individual phases by holding **Ctrl and left-clicking each**. Once all desired phases are selected, **right-click** and choose **Plot Analysis** to generate the plot.

Getting Started - Plot the results



In the **Plot Options** dialog, simply **click OK**. This will launch the **Analysis Module**, FactFlow's dedicated environment for advanced analysis and visualization. Plot settings and customization options will be covered later in this guide.

Getting Started - Plot the results



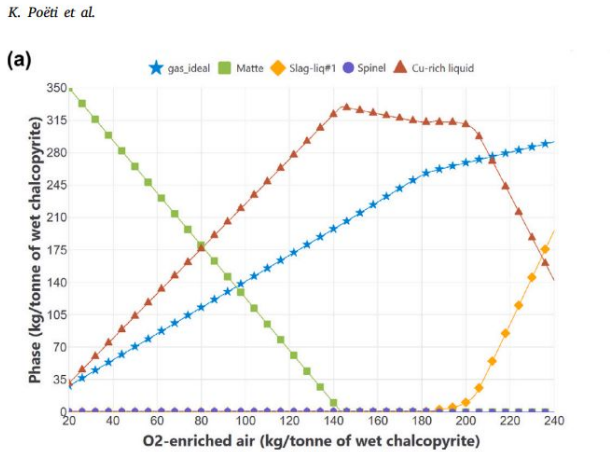


Fig. 10. (a) Phase assemblage evolution during the copper-making step (converter 2) as a a function of the air injection in converter 2.

🎉 Congratulations! You've successfully reproduced **Figure 10a** from the **FactFlow paper**. Most figures in the paper were generated directly using the **Analysis Module**, and you'll find a corresponding **FactFlow example** for nearly every one.
Feel free to explore them to deepen your understanding.
https://doi.org/10.1016/j.calphad.2024.102772

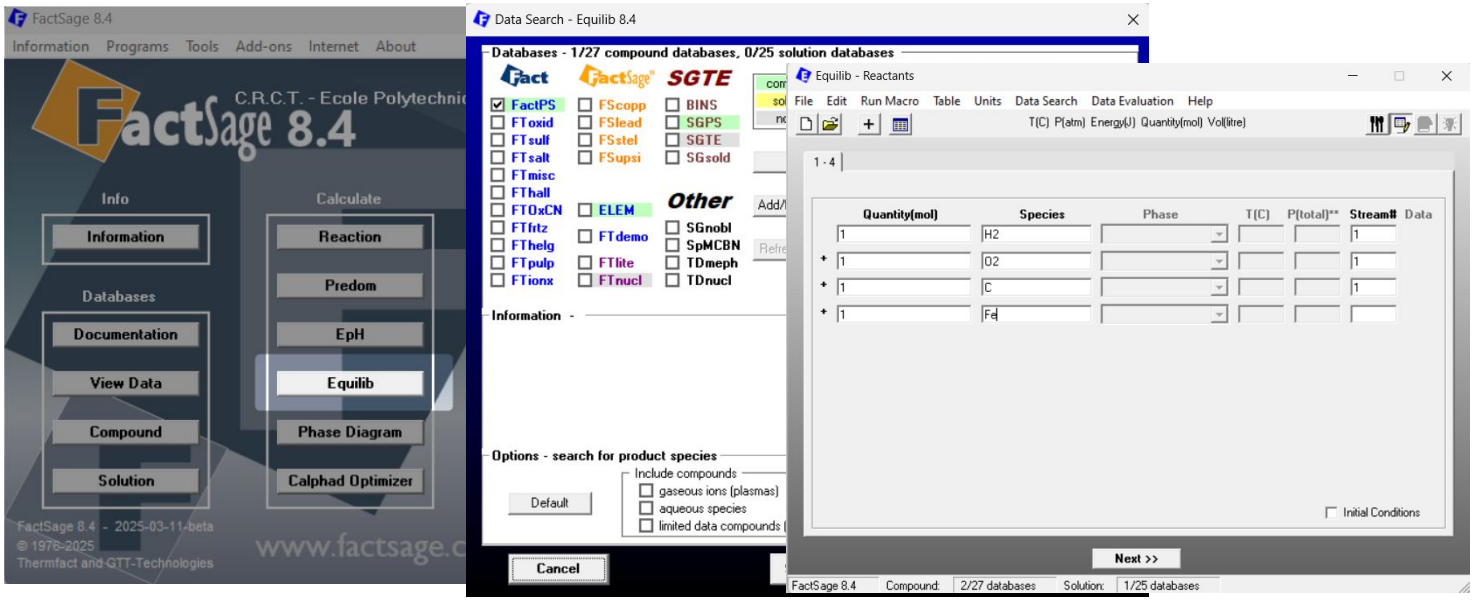
Prepare your Database

Now that you've completed the guided example, it's time to **build your own simulation from scratch**.

The first step is to define the **chemical system** your process will use.

You'll do this using **FactSage's Equilib module**, which lets you select the relevant elements, species, and phases — and send them directly to FactFlow with a single click.

Prepare your Database

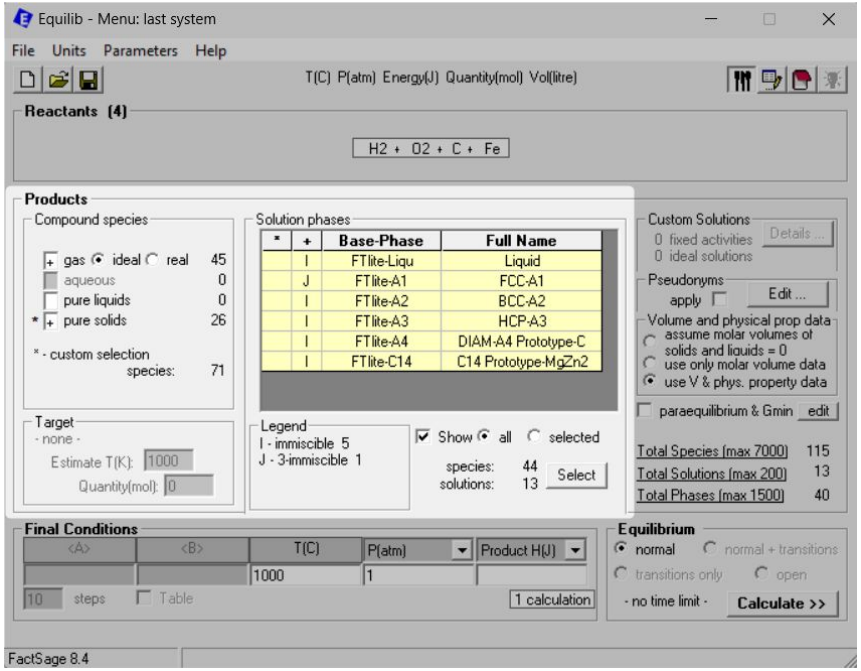


1. Launch the **Equilib module** in FactSage.

2. **Select the appropriate databases** in the Data Search menu.

3. **Enter the input species involved in your system.** The quantities can be arbitrary — they do not affect the mini-database used by FactFlow.

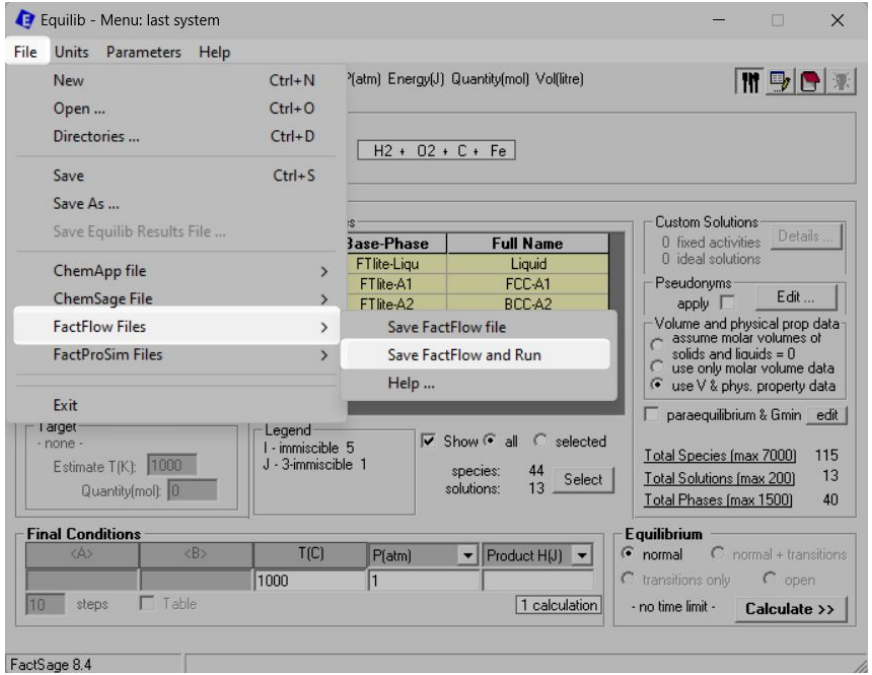
Prepare your Database



Perform the phase selection by choosing all phases that may form in your system. This ensures the generated database includes all relevant phase data.

Temperature, pressure, and other calculation conditions do not affect the database generation and can be left unchanged.

Prepare your Database



Send your system to FactFlow by selecting **File** → **FactFlow Files** → **Save FactFlow and Run**.

This generates the mini database in the background and automatically launches FactFlow with your selected system.

Prepare your Database – Behind the scenes

When you select **Save FactFlow and Run** in the **Equilib** module, FactFlow prepares and stores the files needed for your simulation.

Here's what happens behind the scenes:

- All project files are saved directly inside the **Equi-Cst-Flow** folder in the FactFlow directory. This is the central location where all FactFlow projects are kept.
- Three files are created for your project:
  - **.equi** file – Describes the equilibrium system prepared in Equilib
  - **.cst** file – A compiled binary file representing the mini database (This is what FactFlow actually uses during calculations)
  - **.flow** file – Stores the flowsheet, node configurations, and simulation settings

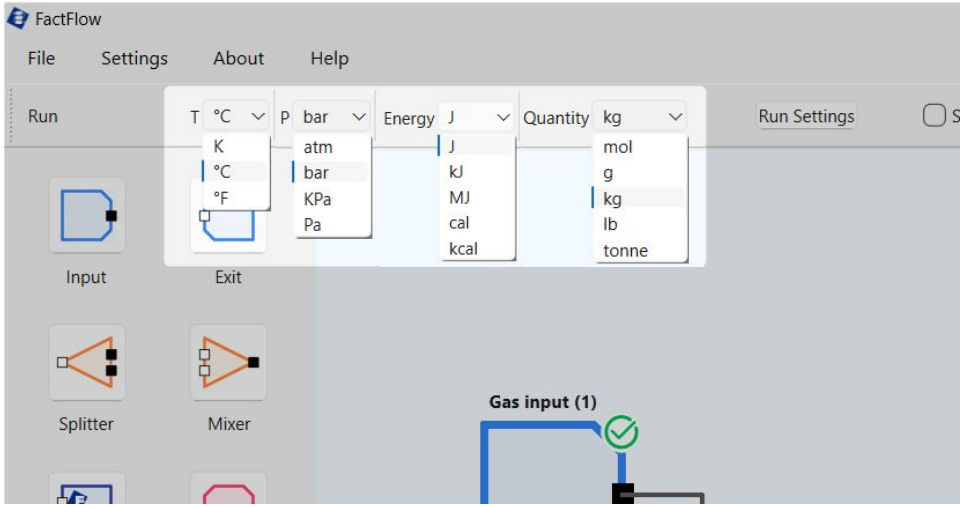
These files work together to define and run your simulation — there's no need to manage them manually, but it's helpful to understand how they relate.

Build the Flowsheet

Now that FactFlow is launched with your system, it's time to **build your process flowsheet**.

In this section, you'll learn how to model your process using **nodes** — the building blocks of a simulation. You'll place, connect, and configure nodes to represent **inputs, reactors, splitters, outputs, and recycling loops**, forming the structure of your simulation.

Build the Flowsheet - Units selection

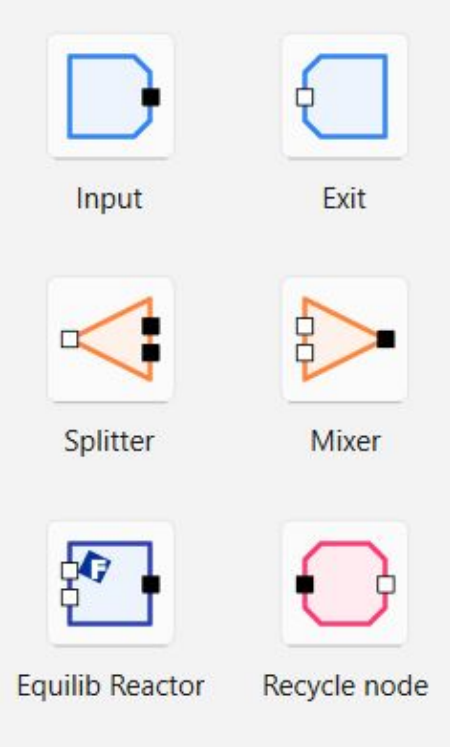


Before building your flowsheet, **choose the units you'd like to work with**. Use the dropdown menus at the top of the FactFlow window to select units for **temperature, pressure, energy, and quantity**.

You can change these units at any time — they only affect how values are displayed and entered in the interface.

Build the Flowsheet - Node Types

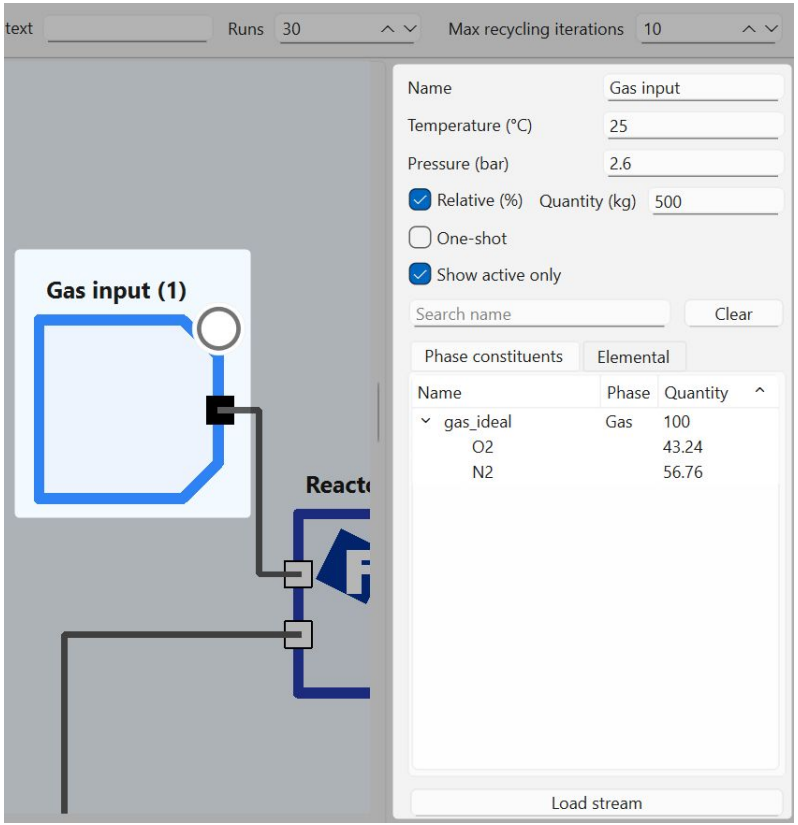
FactFlow simulations are built using interconnected **nodes**, each representing a specific role in the process.



- **Input**: Defines an input stream, including phase or constituent quantities, as well as its temperature and pressure.
- **Exit**: Retrieves and displays the results of a given output stream for analysis.
- **Equilib Reactor**: Performs an equilibrium calculation using the same algorithm as FactSage.
- **Splitter**: Divides a stream based on flow rate, phase, or state.
- **Mixer**: Combines multiple input streams without performing an equilibrium calculation.
- **Recycle**: Loops part of an output back into the system for iterative convergence.

Build the Flowsheet - Input node

All nodes in FactFlow have an **Inspector**, which you can open by clicking on the node.



For the **Input** node, the Inspector allows you to:

- Set the **temperature** and **pressure** of the input stream
- Define the **stream composition** by entering phase or constituent quantities

You can use the **search box** to quickly locate species and the **"Show active only"** option to filter for non-zero entries.

Input nodes introduce material into the process and serve as entry points to the flowsheet.

Build the Flowsheet - Input node

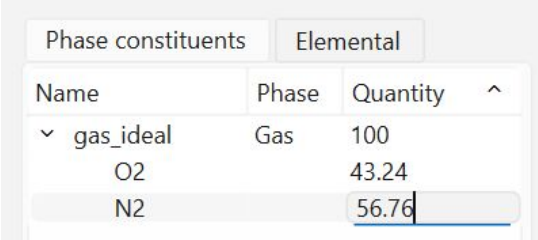
You can define stream composition in two ways: using **absolute quantities** or **relative percentages**. Toggle between modes using the **"Relative (%)"** checkbox in the Input node Inspector.

Absolute entry ("Relative (%)" is off, default)

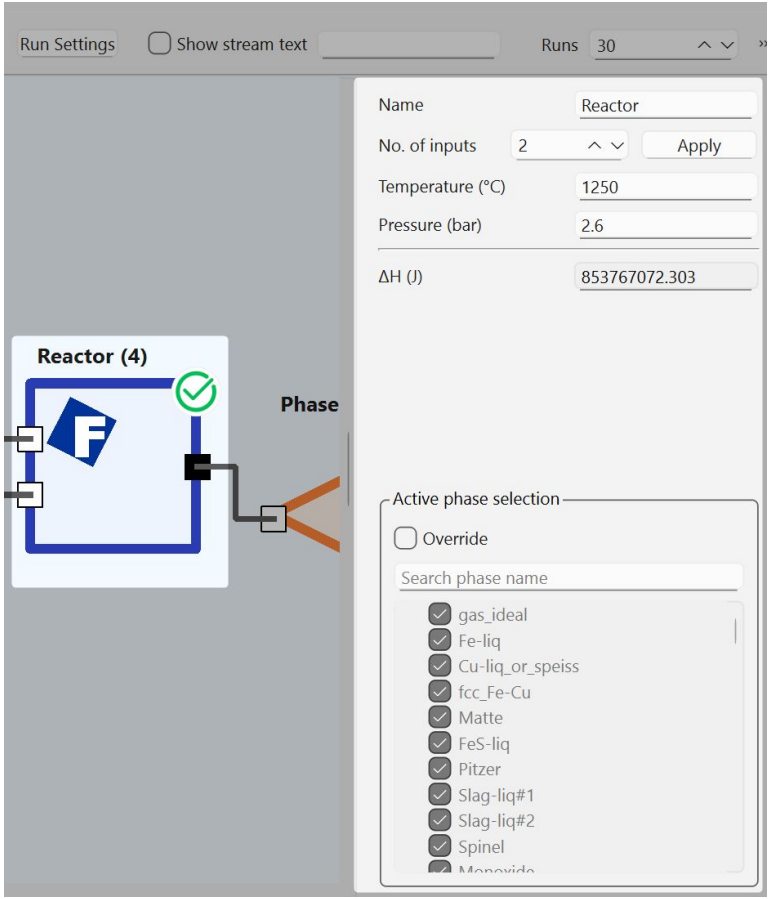
- Double-click a cell in the **Quantity** column to enter a value
- Total amount is determined by what you input

Relative entry ("Relative (%)" is checked)

- Enter percentage values in the Quantity column
- Total must equal 100



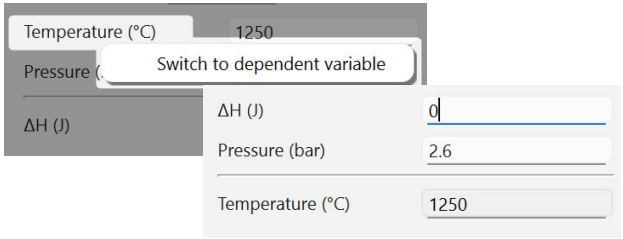
Build the Flowsheet - Equilib Reactor



For the **Equilib Reactor** node, the Inspector allows you to:

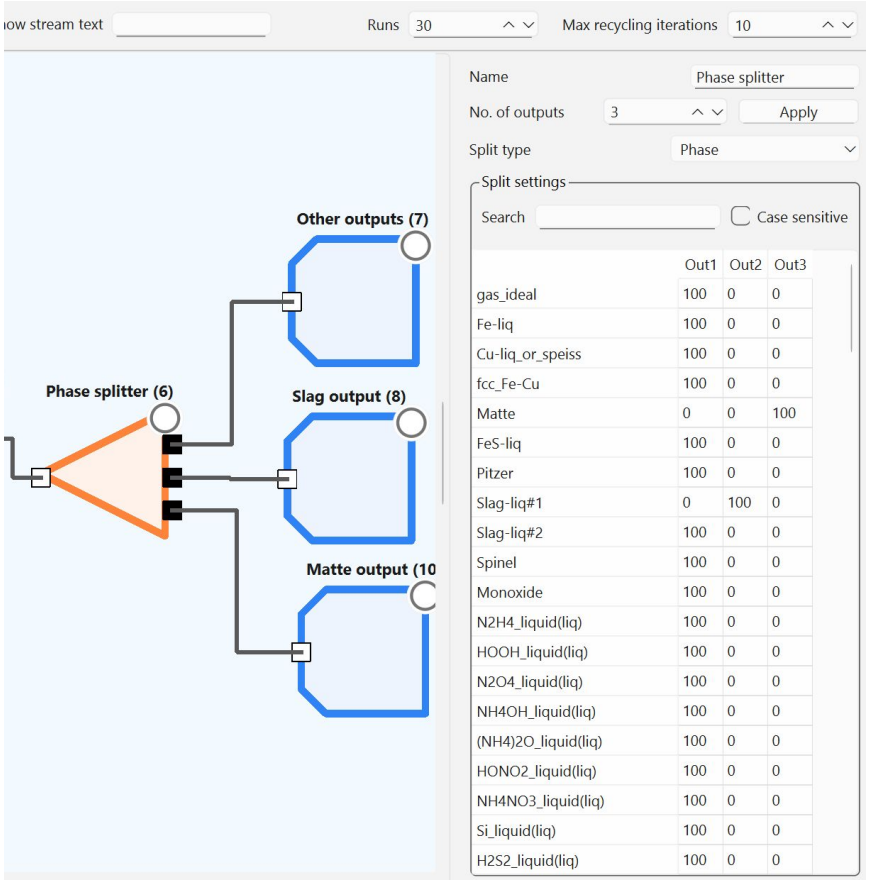
- **Set any 2 of the following:** Temperature, Pressure, ΔH.
- **Number of inputs**: Set how many streams feed into this reactor.

To switch which variable is calculated, right-click on it and select “Set as dependent variable.”



To simulate an adiabatic reaction, set Temperature as the dependent variable and enter ΔH = 0.

Build the Flowsheet - Splitter node



	Out1	Out2	Out3
gas_ideal	100	0	0
Fe-liq	100	0	0
Cu-liq_or_speiss	100	0	0
fcc_Fe-Cu	100	0	0
Matte	0	0	100
FeS-liq	100	0	0
Pitzer	100	0	0
Slag-liq#1	0	100	0
Slag-liq#2	100	0	0
Spinel	100	0	0
Monoxide	100	0	0
N2H4_liquid(liq)	100	0	0
HOOH_liquid(liq)	100	0	0
N2O4_liquid(liq)	100	0	0
NH4OH_liquid(liq)	100	0	0
(NH4)2O_liquid(liq)	100	0	0
HONO2_liquid(liq)	100	0	0
NH4NO3_liquid(liq)	100	0	0
Si_liquid(liq)	100	0	0
H2S2_liquid(liq)	100	0	0

The **Splitter** node is used to divide an input stream into multiple output streams.

Split types:

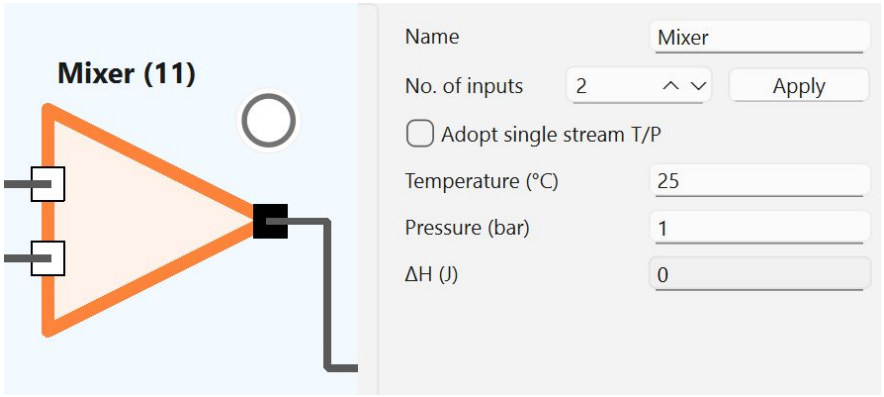
- **Flow**: Splits the entire stream by total flow percentage.
- **Phase**: Splits the stream by phase (e.g., metal, slag, matte).
- **State**: Splits by state of matter (solid, liquid, gas).

In all modes, percentages determine how much of each part of the stream is routed to each output.

Use Splitter nodes when directing specific parts of a stream to different process paths.

Build the Flowsheet - Mixer node

The **Mixer** node combines multiple input streams into a single output without performing an equilibrium calculation.



Configuration

- **Number of inputs**: Set how many streams enter the Mixer.
- **Temperature & Pressure**: These values are imposed on the mixed stream.
- **ΔH**: Displays the energy required to bring all inputs to the target T/P.

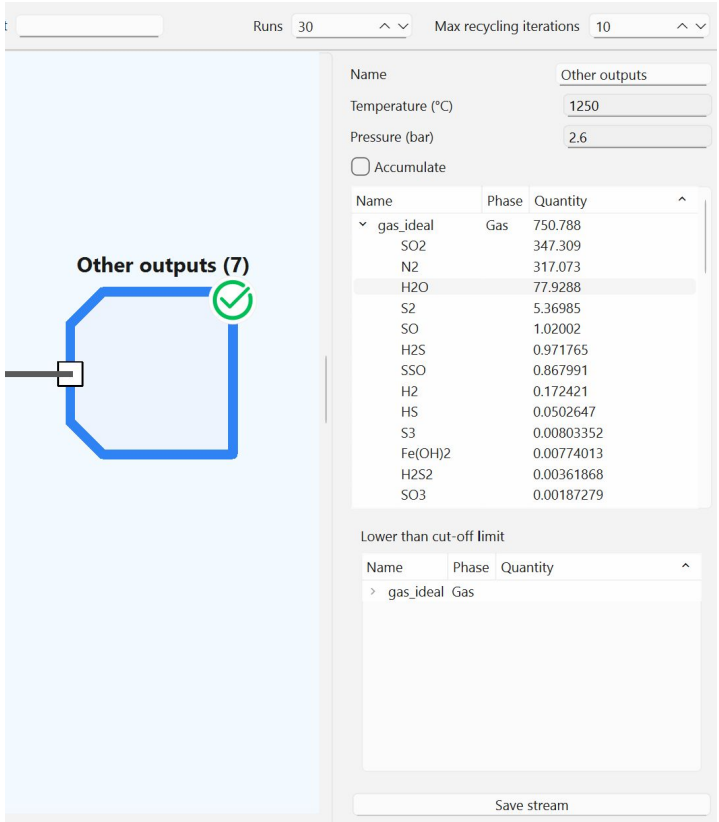
Adopt single stream T/P: If this box is checked and **only one non-empty stream** enters the Mixer, the Mixer will:

- Ignore the user-specified T/P
- Adopt the temperature and pressure of that single stream

This is useful when using the Mixer as a **pass-through node** — nothing is altered unless multiple streams are present. When multiple streams are non-empty, the Mixer reverts to its standard behavior: mixing them and applying the defined T/P.

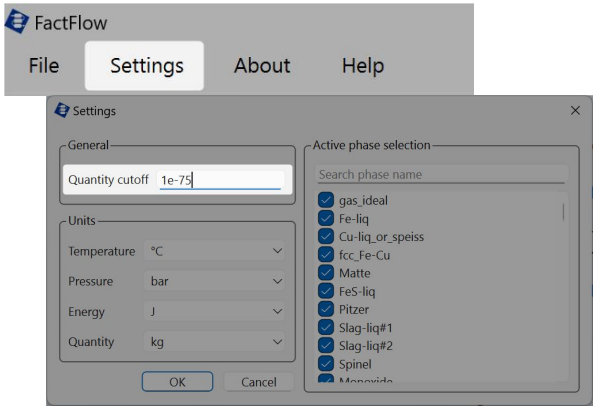
Build the Flowsheet - Exit node

The **Exit** node is used to view the contents of an output stream in your flowsheet.



The Inspector lists all **phases and constituents**, along with their quantities.

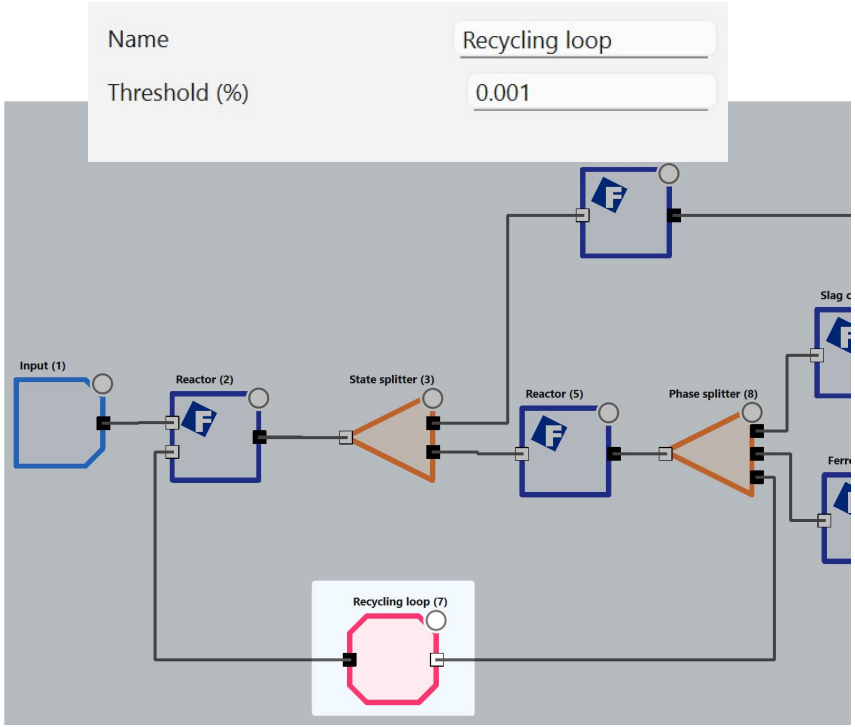
Species below the cut-off limit appear in a separate section.
You can adjust the cut-off limit in Settings:



Accumulate: Sums output values across multiple recycling iterations.

Build the Flowsheet - Recycle node

The **Recycle** node allows part of a stream to be sent back to an earlier point in the flowsheet, enabling **Recycling Iterations**.



The **Threshold (%)** setting controls when the recycling loop should stop.

For each phase or constituent, FactFlow checks whether the **percentage change** in quantity between two consecutive iterations is below the threshold.

The loop stops once **all values** meet this condition, or when the **maximum number of recycling iterations** is reached.

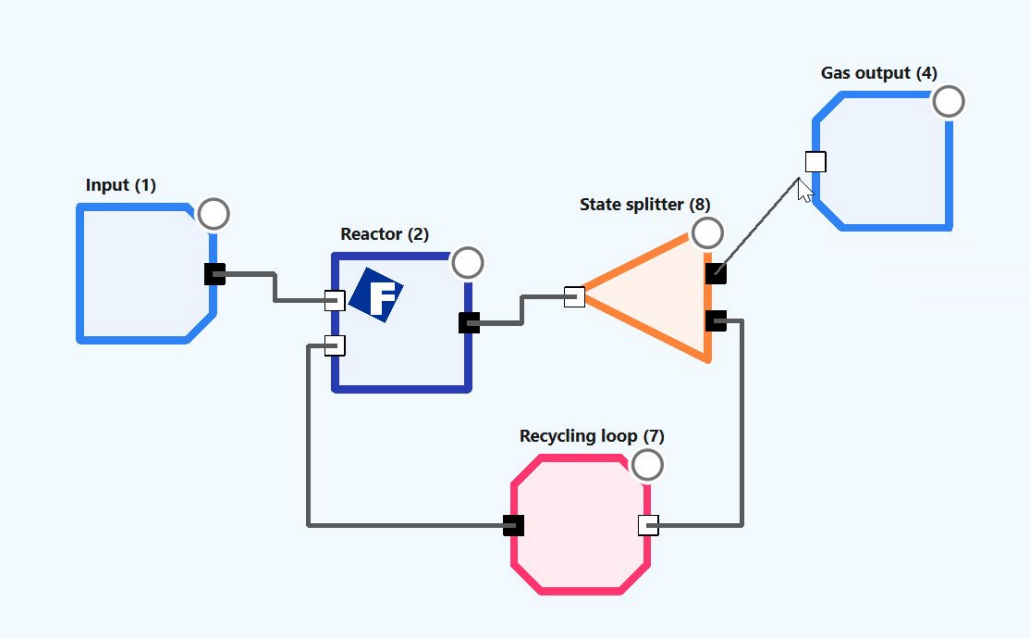
$$\text{Threshold (\%)} > \left| \frac{m_{i,n} - m_{i,n-1}}{m_{i,n-1}} \right| \times 100$$

Build the Flowsheet - Connecting nodes

To connect nodes:

- **Click and drag** from the **output port (black square)** of one node
- **Drop the line** onto the **input port (white square)** of the next node

You can combine nodes freely to represent your process flow.



Configure Run Settings

Now that your flowsheet is built, it's time to configure how the simulation will run.

In this section, you'll learn how to:

- Set how many **runs** to perform (e.g., for parametric studies)
- Set how many **recycling iterations** are allowed (if your flowsheet includes Recycle nodes)
- Vary **input conditions** across both runs and recycling iterations
- Use the **Run Settings panel** to assign variables and control simulation behavior

Configure Run Settings

Understanding Runs vs. Recycling Iterations

When you run a simulation in FactFlow, there are **two layers** of calculation:

1. **Runs** (Parametric Runs)

- Each **Run** represents a **separate simulation** with a **unique set of input values**.
- You can vary inputs like temperature, reactant amount, or pressure across runs.
- Used to perform **parametric studies** and see how outputs change with different conditions.

2. **Recycling Iterations** (Inside each Run)

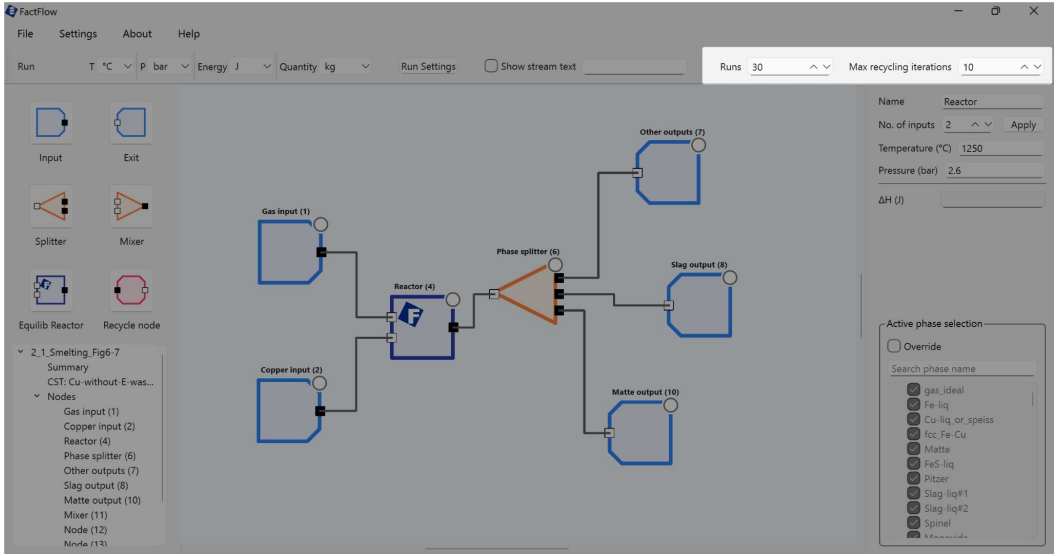
- If your flowsheet includes a **Recycle node**, FactFlow will perform **multiple internal recycling iterations** *within each Run*.
- These iterations continue until:
  - All recycle loops converge (based on the threshold you set), or
  - The **Max recycling iterations** limit is reached.

Run 1
- Iteration 1
- Iteration 2
- Iteration 3 (converged)

Run 2
- Iteration 1
- Iteration 2 (converged)

Run 3
- Iteration 1
- Iteration 2
- Iteration 3
- Iteration 4 (max reached)

Configure Run Settings

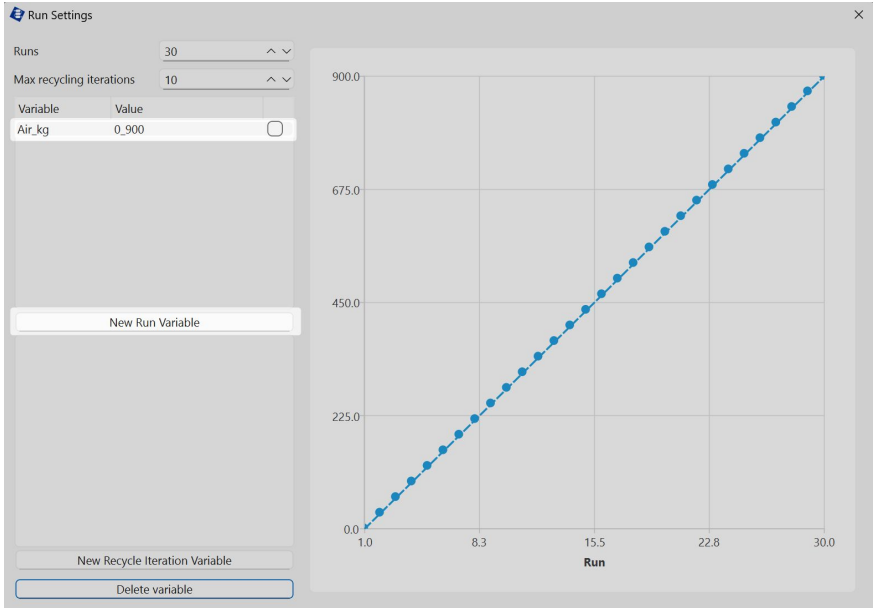


Set Runs and Max Recycling Iterations

- **Runs** field: Controls how many parametric runs will be executed.
- **Max recycling iterations** field: Controls the maximum number of iterations per run *if* recycling is active.

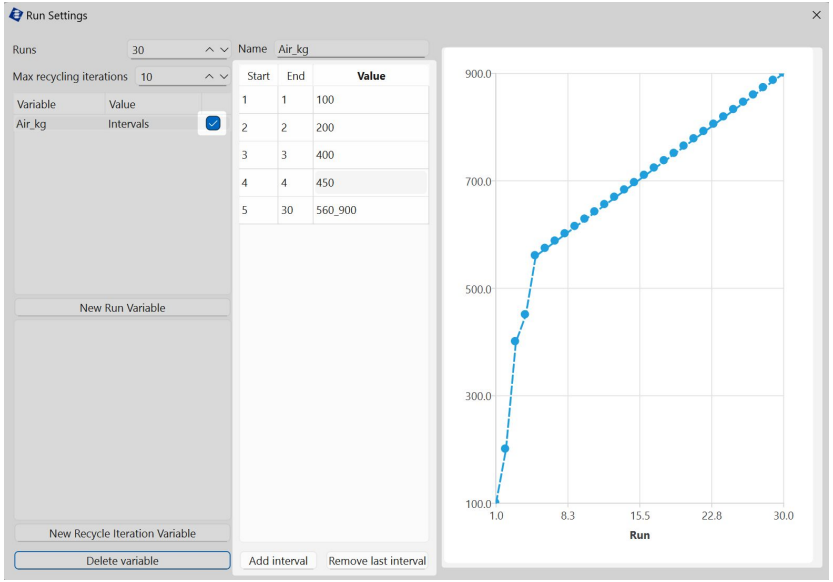
Note: In the example shown, there are no Recycle nodes in the flowsheet. Even though the Max recycling iterations is set to 10, no recycling will occur — only one calculation is performed per run.

Configure Run Settings - Creating a Run Variable



- Click **New Run Variable** to add a variable.
- Enter the variable's name in the field that appears.
- Rename a variable at any time by double-clicking its name in the variable list.
- In the **Value** column, enter a range using the format: **MIN_MAX**
  - Example: 0_900 will vary the value from 0 to 900 across all runs.
  - Steps are automatically divided evenly.

Configure Run Settings - Advanced Intervals

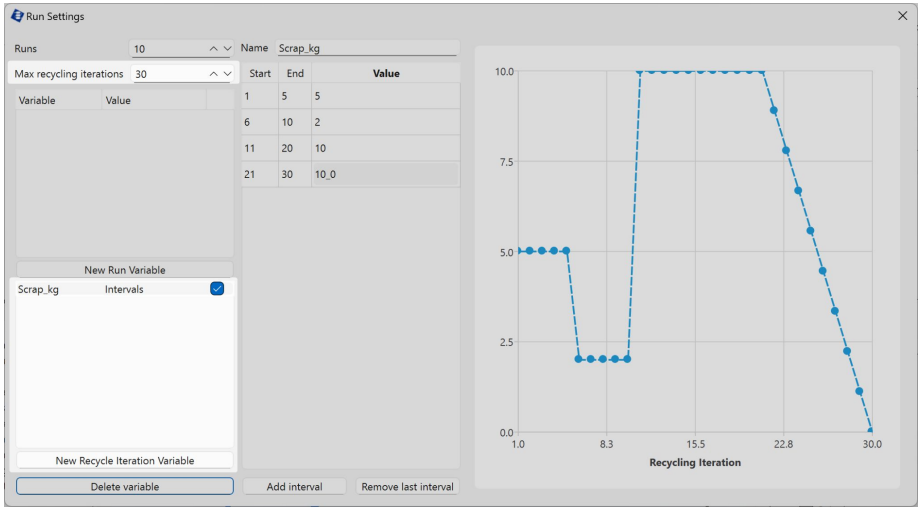


Tip: You can mix fixed values and ranges in the same variable. This is especially useful when working with multiple run variables, allowing you to keep one value constant while varying another.

- Check the box next to a variable to enable **Interval Mode**.
- This opens a table where you can define multiple **intervals**, each with:
  - A **Start** and **End** run index
  - A **Value**, which can be:
    - A **fixed value** (e.g., 400)
    - A **range** (e.g., 560_900), which will be linearly interpolated across the interval
  - Use the **Add interval** and **Remove last interval** buttons to manage rows.

Configure Run Settings - Recycle Iteration Variables (Advanced)

Most simulations vary input values across runs, but you can also vary inputs **within each run**, across **recycling iterations**. This feature is especially useful for modeling **parameterized behaviour as a function of the recycling iteration**.



How to Define One:

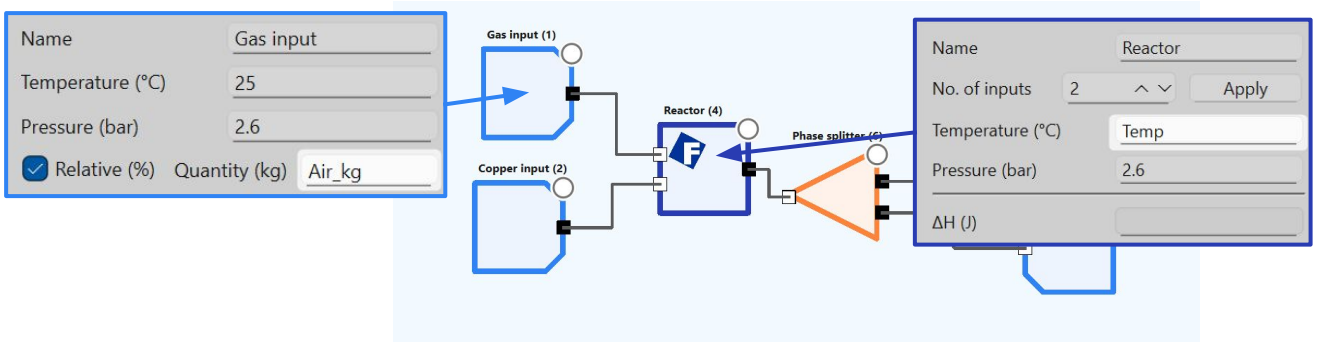
- Click **New Recycle Iteration Variable** in the Run Settings panel.
- Enter the variable name and range (MIN_MAX) just like a Run Variable.
- Optionally, enable **Interval Mode** for finer control across iterations.

Configure Run Settings - Using Variables in the Flowsheet

To use a **Run Variable** or **Recycle Iteration Variable**, simply **type the variable's name into any editable input field** in your flowsheet.

You can use variables in:

- **Temperature or Pressure** fields of Equilib Reactors
- **Quantities** in Input nodes
- **Percentage splits** in Splitter nodes *(more advanced — covered later in the Function Builder section)*



FactFlow will automatically substitute the correct value for each run or iteration based on your definitions in the **Run Settings** panel.

Tip: You can use multiple variables in a single simulation to explore combinations of process conditions.

Configure Run Settings - Importing Variables from Excel

You can define **Run Variables** and **Recycle Iteration Variables** by importing them from an Excel file.

- The **first sheet** in the Excel file is used to define variables that vary across **runs**
- The **second sheet**, if present, is used to define variables that vary across **recycling iterations**
- If only one sheet is included, FactFlow will assume it defines **Run Variables**

Each column defines a **separate variable**, and each row provides the value for a specific run or iteration (starting from 1).

	A	B
1	Temp	
2	100	
3	200	
4	300	
5	400	
6	500	
7	600	
8	700	
9	800	
10	900	
11	1000	
12	1100	
13	1200	
14	1300	
15	1400	
16	1500	
17		

Runs | Recycles

	A	B
1	O2_mol	Purity
2	1	50
3	2	50
4	3	50
5	4	50
6	5	50
7	6	80
8	7	80
9	8	80
10	9	80
11	10	80
12	11	80
13	12	80
14	13	80
15	14	80
16	15	80
17		

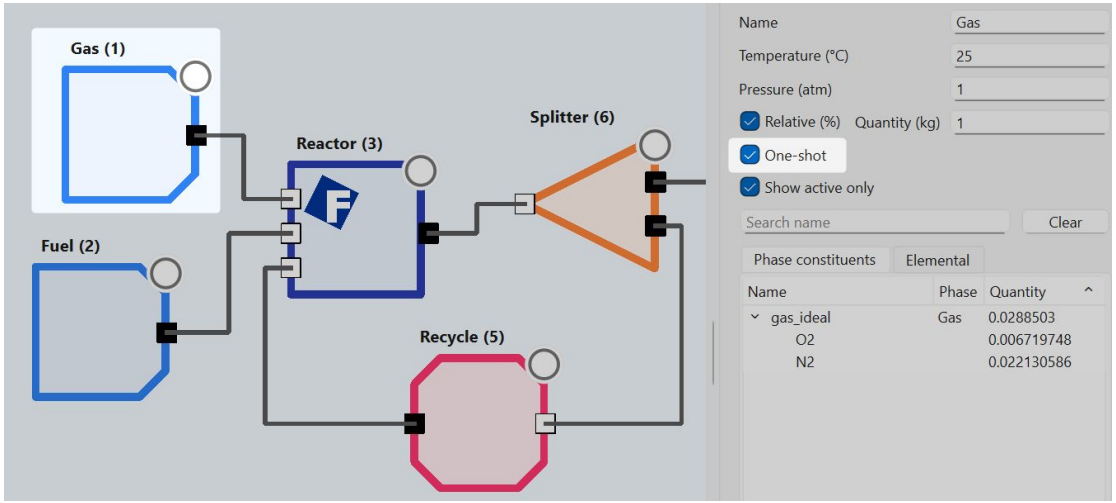
Runs | Recycles | +

Configure Run Settings - One-shot Input nodes

In flowsheets with recycling, some inputs may only be intended to act **once**, such as an initial dose of fuel, gas, or a reactant.

The **One-shot** checkbox in an Input node allows you to model this behavior:

- When **enabled**, the input node will only provide material during the **first recycling iteration** of each run.
- For all subsequent iterations in that run, the node is effectively **empty** — it doesn't inject additional material.



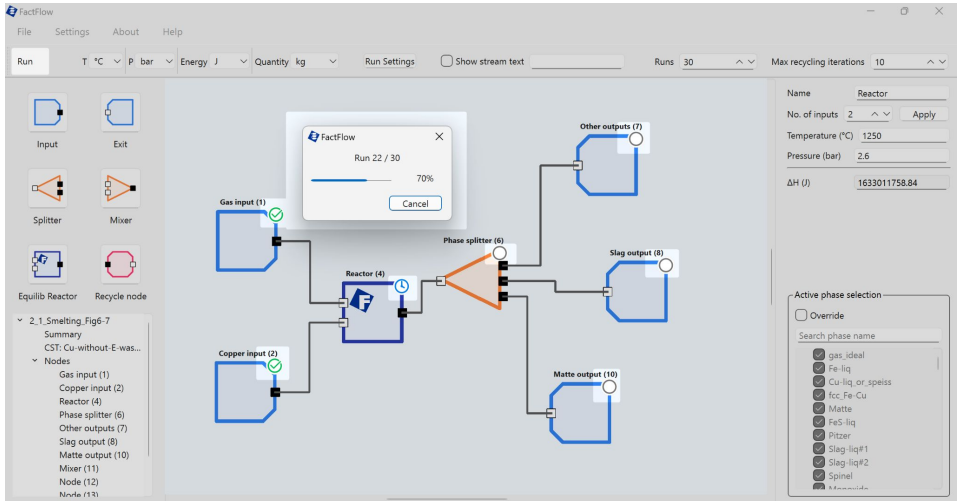
Run the Simulation

With your flowsheet built and variables configured, you're ready to execute the simulation.

In this section, you'll learn how to:

- Launch the simulation using the **Run** button
- Interpret **node status indicators** during execution

Run the Simulation



Once everything is set up, you can **launch your simulation** by clicking the **Run** button at the top left of the FactFlow window.

While the simulation is running, each node displays a **status icon** to show its current state:

- **Idle** – The node is waiting to be calculated.
- **Calculating** – The node is actively processing.
- **Success** – The node completed its calculation without errors.
- **Error** – The node encountered an issue.

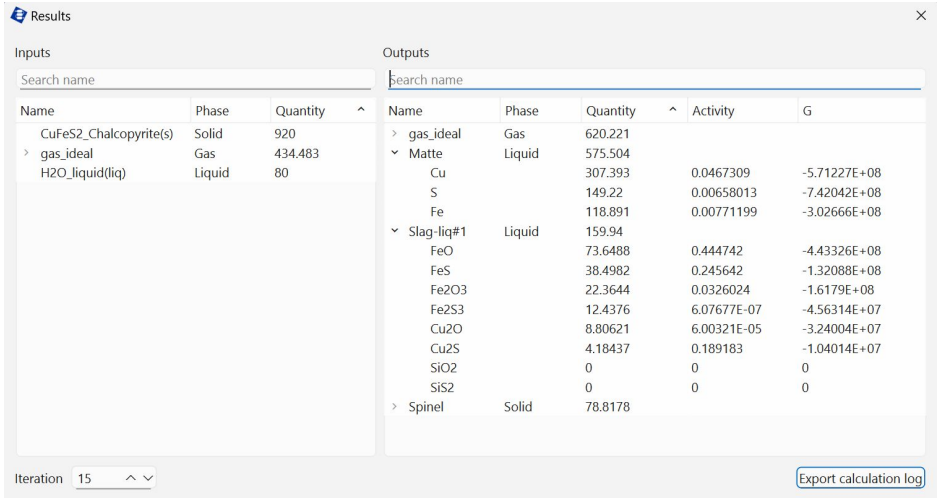
Analyzing Results

Once your simulation has completed, it’s time to explore and interpret the results.

In this section, you’ll learn how to:

- View **stream results** directly from any node
- Use the **Analysis Module** to generate plots and compare results across runs

Analyzing Results



After the simulation finishes, you can view the results from **any node** in the flowsheet.

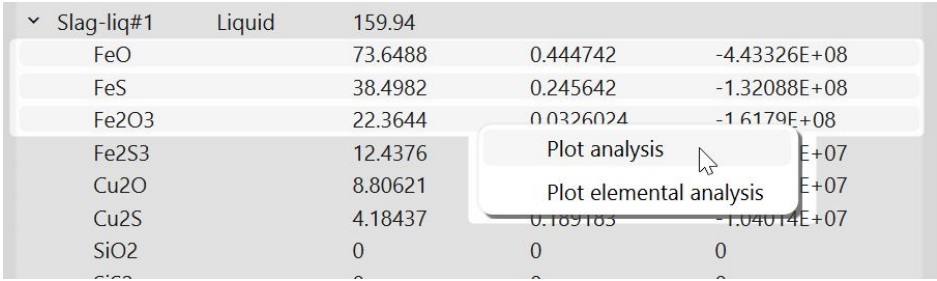
- **Double-click a node** to open its **Results** window
- For **Exit** nodes, results are accessible via the Inspector

The **Results** window displays:

- The **phases and constituents** present at that point in the process
- Their corresponding **quantities**, **activities**, and **Gibbs energy (G)** (when applicable)

For **Equilib Reactor** nodes, you can also click **Export calculation log** to save a detailed record of the equilibrium calculation. This log is the same as what would be generated by **FactSage** for that iteration.

Analyzing Results - Plotting



To create a plot, **right-click** any row in the Results window.

You'll see two options:

- **Plot analysis** – Plot quantities for the selected phases and species
- **Plot elemental analysis** – Plot total elemental amounts

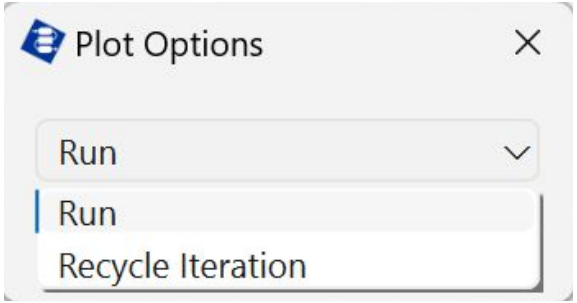
Selecting What to Plot

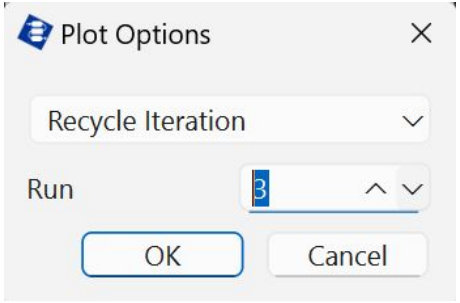
You can select one or more species using standard multi-selection methods:

- **Click and drag** to select a group
- **Shift-click** to select a range
- **Ctrl-click** to select or deselect individual rows

This allows you to customize exactly what gets sent to the **Analysis Module** for plotting.

Analyzing Results - Plotting





Before generating your plot, the **Plot Options** dialog will appear.

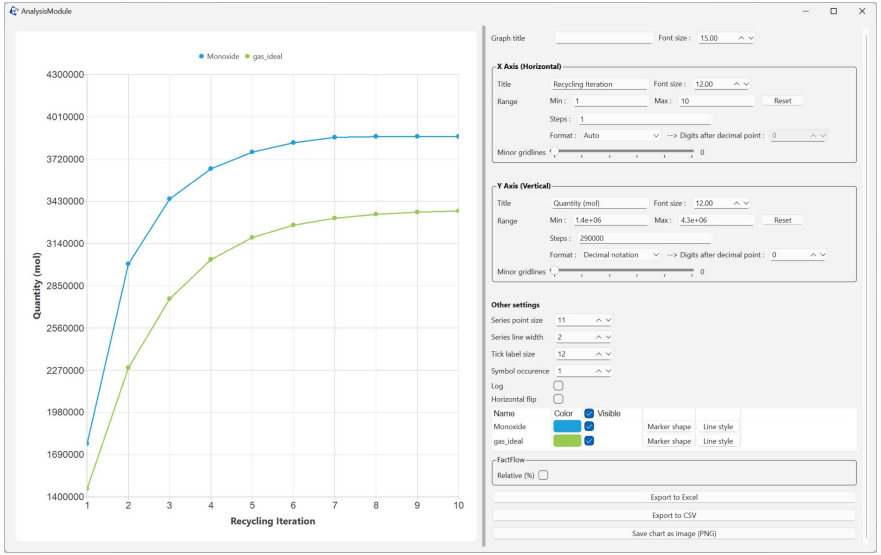
This lets you choose what the x-axis of the plot should represent:

- **Run** – Plot how values change across parametric runs
- **Recycle Iteration** – Plot how values evolve within a single run, across recycling iterations

If you select **Recycle Iteration**, an additional field will appear allowing you to specify **which Run** to analyze. Each run has its own internal recycle history, so this selection determines which run's iterations will be plotted.

Click **OK** to proceed to the Analysis Module with your selected view.

Analyzing Results - Plotting



The plot will open in the **Analysis Module** — a dedicated space for exploring and customizing your simulation results.

Here, you can:

- Visualize how selected values change across **runs** or **recycling iterations**
- Switch between **absolute values** and **relative percentages** (e.g., weight % within a phase)
- Enable log scale, adjust axes, labels, and visual styles
- **Export** the plot as an image or download the data as CSV or Excel

Advanced Section – Function Builder

You've made it!

By now, you've learned how to prepare your system, build a flowsheet, configure variables, and analyze results.

This next section introduces the **Function Builder** — a powerful tool that lets you define values using **custom formulas**, **mathematical functions**, and even simple **programming logic** like if-else, for, and while.

If you're ready to go beyond basic variable inputs, this tool gives you full control.

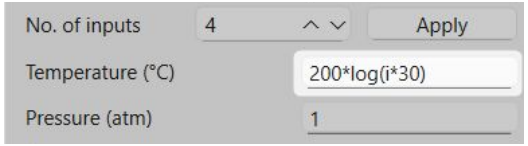
Function Builder - Custom formulas

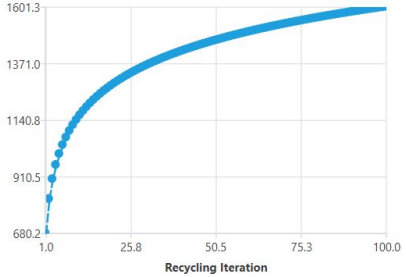
In any numeric input field, you can enter not just a number or variable name — but a **mathematical expression** using the Function Builder.

You can use:

- **Basic arithmetic**: +, -, *, /, ^
- **Parentheses** for grouping: (A + B) / 2
- **Math functions**: sqrt(x), log(x), exp(x), abs(x), min(a, b), max(a, b)
- **Constants**: pi, e

You can also reference any **Run Variable** or **Recycle Iteration Variable** by name.





Built-in variables

- **run** ⟶ the current run index (starting from 1)
- **i** ⟶ the current recycling iteration index (starting from 1)

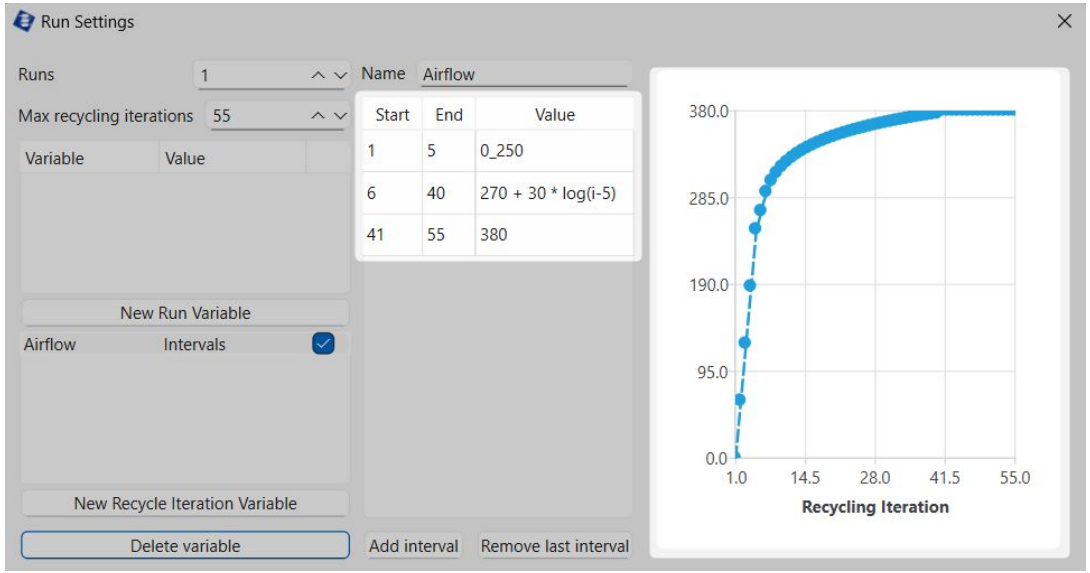
Expressions are evaluated automatically for each run or iteration, allowing fully dynamic behavior.

Function Builder - Custom formulas

In addition to input fields, mathematical expressions can also be used in the **Value** column of the Run Settings panel — both in **basic** and **interval** modes.

This is especially powerful when using **Recycle Iteration Variables**, which can model **parameterized behaviour** as a function of the recycling iteration.

Example: Suppose you're simulating an oxidation step in a converter. Air flow can be increased with each iteration depending on user setting:



This allows you to model:

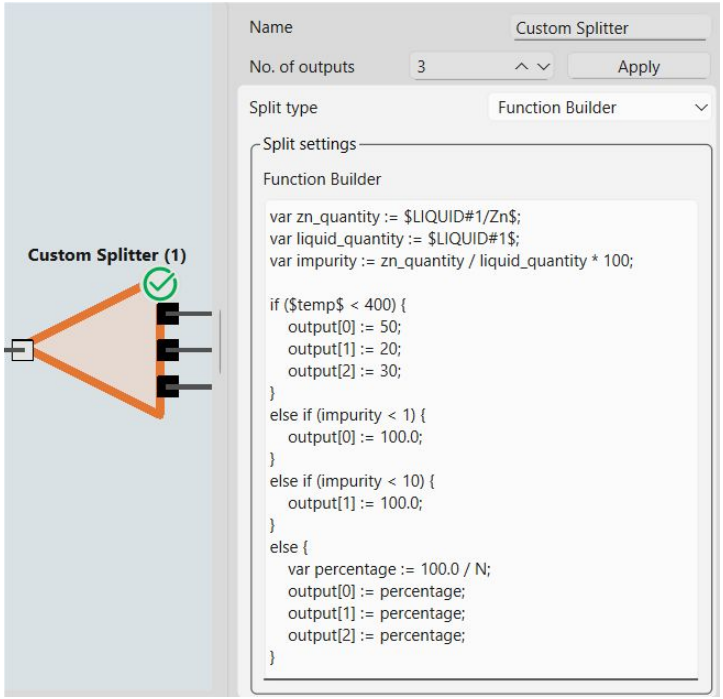
- A **diffusion-limited stage**
- A **steady-state plateau**

All within a single simulation run.

Function Builder - Splitter

In addition to using the Function Builder in input fields and variables, you can also use it directly in the **Splitter** node to control how material is routed between outputs. To enable this, set the **Split type** to **Function Builder** in the node's Inspector.

Instead of assigning fixed split percentages, the Function Builder lets you use **formulas** and **conditional logic** (if, else if, else) to dynamically control the split based on the input stream's composition or conditions.



In this example, the output is split differently depending on the **temperature** and the **Zn impurity** in a liquid phase.

Built-in Variables for Splitters

- **$Phase$ or $Phase/Constituent$** – quantity in the input stream
- **$temp$, $pressure$** – input stream conditions
- **output[index]** – array for assigning output split percentages
- **N** – total number of outputs

Support

We hope this guide has helped you get the most out of FactFlow and explore its full potential.

Should you run into any issues, have suggestions, or want to share feedback, we'd love to hear from you. Please don't hesitate to reach out at factflow@polymtl.ca.

Our team is here to help and always looking for ways to improve your experience with FactFlow.