Sumo Quick Tutorial

BY

Dynamita, www.dynamita.com, Sigale, France

Sumo *is a third generation wastewater process simulation software. It was put together by a dedicated team with professionalism and thousands of days of tender loving care. Enjoy, and please let us know if you find somewhere it is coming short of your expectations. We will do our best to improve it.*

Imre Takács, on behalf of the whole Dynamita team

Sigale-Toulouse-Budapest-Toronto-Innsbruck

info@dynamita.com

support@dynamita.com

Table of Contents

1 Contact

Dynamita SARL
2015 route d'Aiglun
Sigale, 06910
France

www.dynamita.com
info@dynamita.com
Mobile in France: +33.6.42.82.76.81
Landline: +33.4.93.03.34.06

Released: May, 2021

2 Install Sumo

The following steps will help to guide you through the installation of Sumo.

2.1 Prepare Microsoft Windows for Sumo

Operating system

Make sure your computer is operating Microsoft Windows 7 or later. Sumo supports touch screen computers running Windows 8/8.1 and Windows 10.

Sumo can be used on a Mac with Windows installed or through an emulator like “Parallels”.

Microsoft Office

Sumo requires Microsoft Excel 2007 or later installed.

.NET

Please make sure that your computer is running the Microsoft .NET 4.7.2 framework. You can check it in the list of installed applications (Control Panel / Programs / Programs and Features). If the .NET framework’s 4.7.2 version is not installed, please download and install it from the following location:

https://dotnet.microsoft.com/download/dotnet-framework/net472

Windows 10 comes with the required .NET environment. On other operating systems, if you have it already installed on your computer, the downloaded installer will exit or inform you. Proceed to the next step.

2.2 Installation of Sumo

You have received a link to the Sumo installer (or in rare cases the install file on a USB key). This is one file and contains everything necessary to install Sumo once Windows is prepared. Download it through the link provided to your computer. From the USB key you can install directly if desired, it is not required to copy the file to the computer.

Install Sumo by starting the install package and follow the instructions. Administrator rights may be required during installation. For workstations, we recommend choosing the “Install for myself only” option.

2.3 Obtaining a license

Obtaining a license is a two-step process.

1. After install, start Sumo from the Windows Start menu. Sumo will display a message providing multiple options. If you don’t have the license file yet, Select “I need a new license”. Sumo will display two additional buttons to show information about your hardware (Machine Identifier Code) and to copy this information directly to the Windows clipboard. Please paste this code into an email and send it to support@dynamita.com. Dynamita will provide a license file for you according to our agreement.
2. Copy the license file Dynamita provided to a folder on your computer, start Sumo, choose the “I have a license” option, click “Select license file” and navigate to load the license file. As long as the license file is not deleted or moved and it is valid, this validation does not have to be repeated. Other options, like using a license server or a hardlock, are also available.

3 How to build a plant

Start Sumo from the Desktop icon (pic). Sumo will start with showing the Welcome Screen, as illustrated in Figure 1. Below the Menu Bar, you will see the Task Bar that guides you through the project workflow, all the way from configuration to simulation. The main window is split into four panels with various functionalities.

Figure 1 – Sumo21 startup screen

3.1 Configure

Click on the *Configure* tab. In this introduction, we will build a simple AO configuration using the *Flow elements*, *Bioreactors* and *Separators* categories from the element list of the top left screen panel. Select the desired process unit by opening the category and dragging the process unit to the drawing board. To drop the selected unit, just release the mouse button (Figure 2). The units can be connected with pipes by simply positioning their outflow connection (port) on top of an input port of another process unit (Figure 3), or by positioning the mouse on an output port of a process unit, pressing the left mouse button, then moving the mouse – and this way drag the pipe – to an input port of another process unit (Figure 4). Existing pipes can be removed by right-clicking on them and selecting *Disconnect pipe* from the pop-up menu.

Build the plant configuration and connect the pipes as shown in Figure 5.

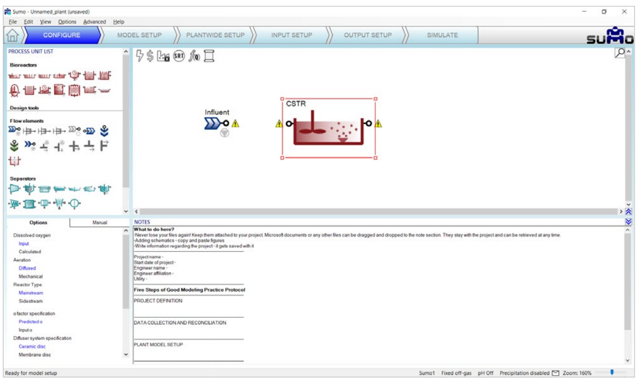

Figure 2 - Building the plant layout

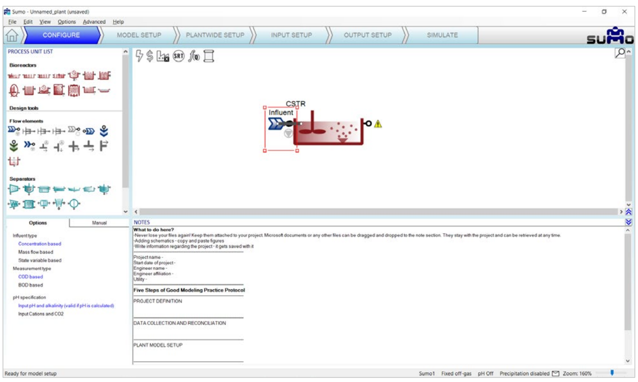

Figure 3 - Pipe created by touching process units

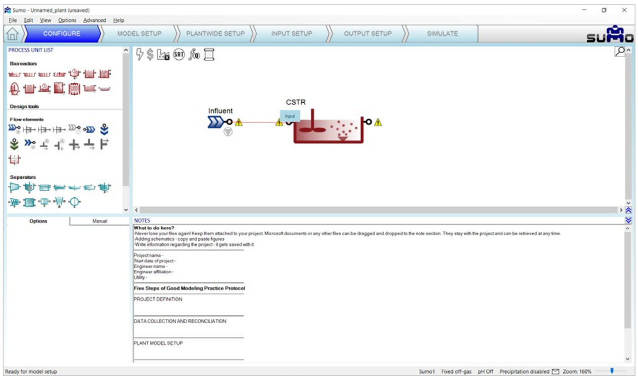

Figure 4 - Pipe created the conventional way

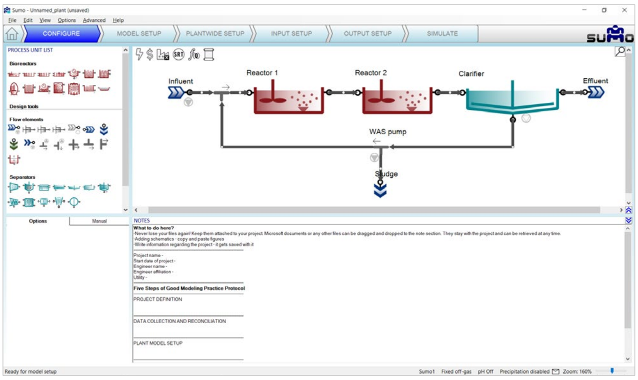

Figure 5 - The example layout

Rename or transform (rotate, mirror) the process units using the right-click pop-up menu. The pipes can be renamed as well, but this is usually not important – the pipe names are hidden by default (this can be changed in the *View* menu on the top). The visibility of process unit names can be controlled on a one-by-one basis.

Note: the above settings can only be modified in *Configuration* mode, but you can return and perform them at any point during your work. Changing the process unit names will not result in recompilation of the project model.

3.2 Model setup

In this tutorial, the model parameters will be left at the default values.

3.3 Plantwide setup

Proceeding to the *Plantwide Setup* tab, complex plantwide calculations can be defined.

“Plantwide” means that the calculations are based on variables contained in several process units around the plant. Various calculations can be added to the simulation, ranging from SRT calculation to different types of controllers. In this example, a plant SRT calculation will be added by clicking on the *Sludge Retention Time* button in the top left screen panel.

The SRT calculation is set up the following way: drag reactors which contain sludge mass to the numerator and drag the wastage pump – and if desired, the effluent – in the denominator, as shown by Figure 6.

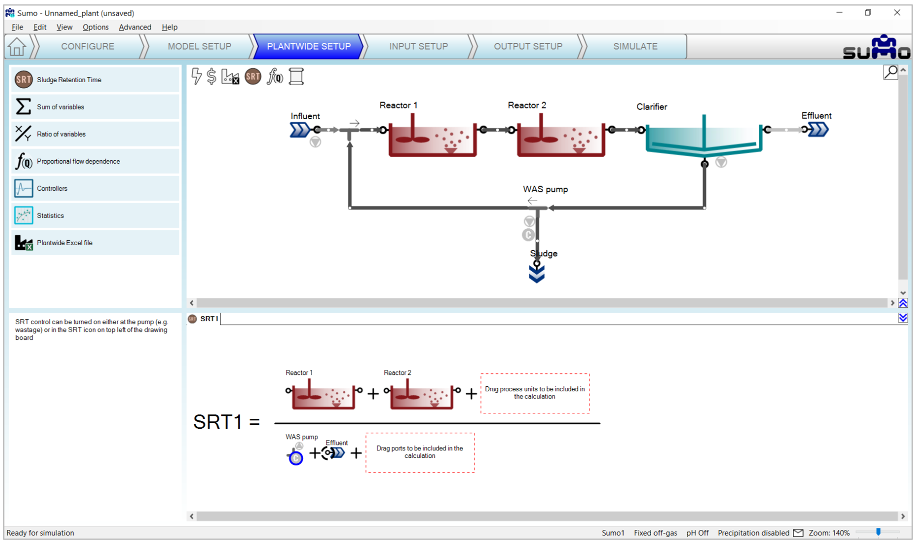

Figure 6 - SRT calculation setup

Target SRT can be defined as well, by assigning a proper controlled port. These ports, such as the WAS pump in our example, are indicated by a “C” sign within a blue circle. A certain pump can only be used to control one SRT.

3.4 Input setup

Choosing the *Input Setup* task, the blue workflow tab above the drawing board automatically splits into *Constants* and *Dynamics*. Meanwhile, the model starts to be built. For the initial plant setup, we will only need the *Constants*, as shown on Figure 7.

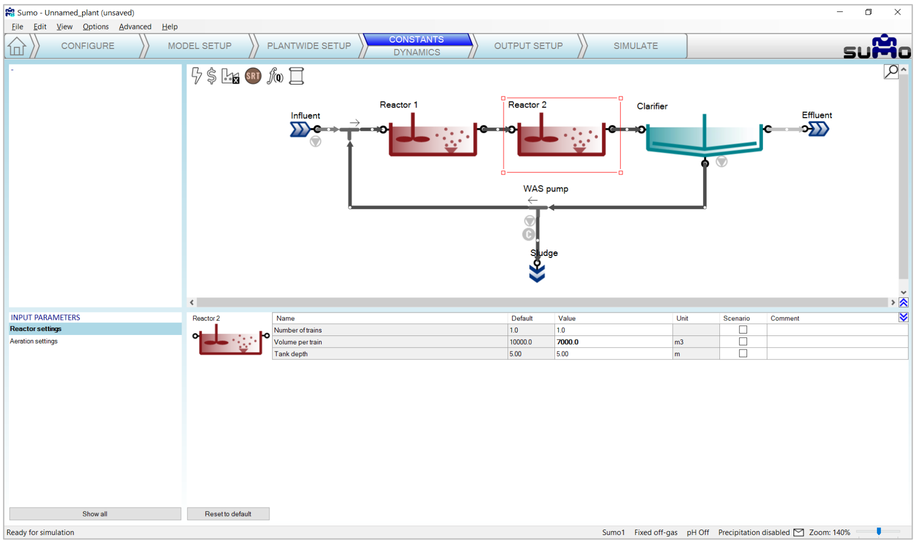

Figure 7 - Constant Input setup example

In this simple configuration, we only need to set a few values to build a realistic plant model, first for typical dry weather operation. Use the values from Table 1 for this example (we will only modify the reactor volumes, the influent, clarifier and wastage pump settings will be left at default values).

Table 1 - Input setup for simple AS plant

Process Unit	Parameter group	Parameter	Value	Unit
Reactor 1	Reactor settings	Volume per train	3000	m^3
Reactor 2	Reactor settings	Volume per train	7000	m^3

Enter these values by selecting the respective process unit on the drawing board and the parameter group in the *Input parameters* menu (bottom left panel) – the values can be edited in the bottom right panel (Figure 7). Each value that is different from the default will be highlighted with bold letter type. There is also a similar indication in the *Input parameters* menu for parameter groups that contain non-default values.

The input setup of this wastewater treatment plant model is now ready. Meanwhile the model has compiled as well (status bar message: “Ready for simulation”).

3.5 Output setup

This task can be used to specify which variables and in what format should be displayed and/or saved during the simulation phase.

For this example, add a **table** with the *Frequently used variables* of the Influent, Reactors, Effluent and Sludge (Figure 8); an MLSS **timechart** (Figure 9), Effluent N **timechart** (Figure 10) and a COD/BOD **barchart** (Figure 11). Select process units on the drawing board, then drag and drop variables (or variable groups) from the bottom left panel. You can also drag and drop process units to add new columns to an already existing table/barchart.

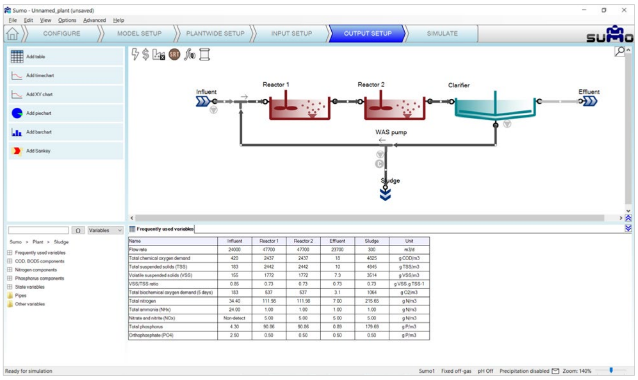

Figure 8 - Summary table setup

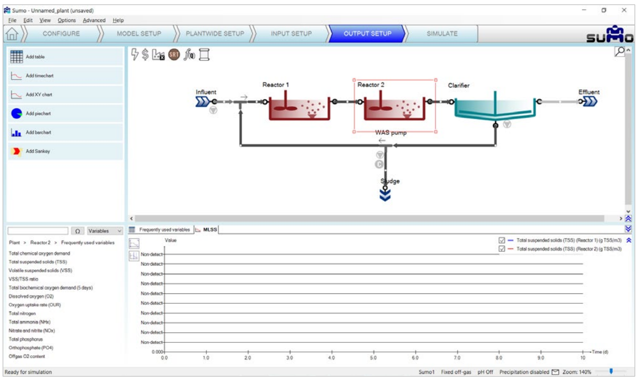

Figure 9 - MLSS timechart setup

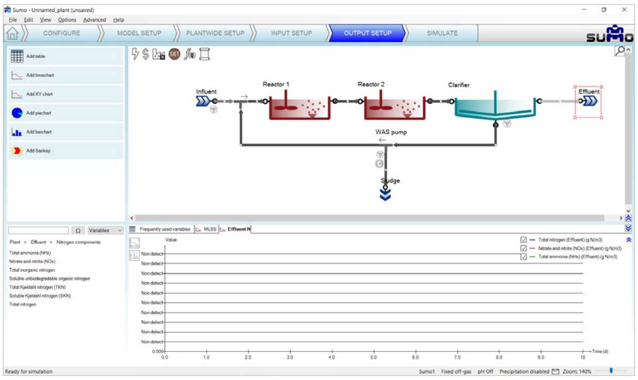

Figure 10 - Effluent N timechart setup

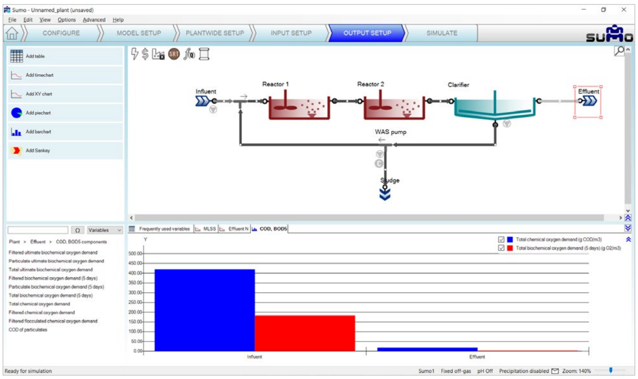

Figure 11 - COD/BOD5 removal setup

4 Simulate

The last item on the Task Bar can be used to run simulations and observe the results (Figure 12). *Steady* mode can be used to calculate directly the steady-state condition of the variables, whereas using *Dynamic* mode will show the variations with time (switch between the two modes using the tabs on the upper left panel).

Dynamic simulation can be started by pressing the *Start* button. The duration (*Stop time*) and the reporting frequency (*Data interval*) of the simulation can be set before the simulation. Upon first start, the simulation will be run from the initial conditions (defaults specified in the model file and in the process units) and every subsequent run will be initiated with the last system state. A 100-day graph gives a good indication whether the system has settled into stable condition and the results are meaningful for typical dry weather summer operation. (Figure 13 and Figure 14). The COD/BOD$_5$ removal is shown on the barchart (Figure 15).

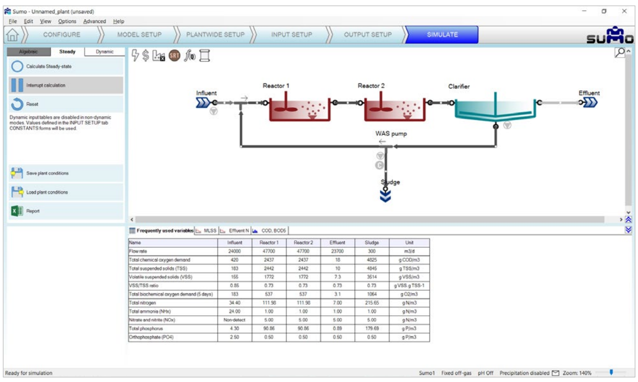

Figure 12 - Simulation tab, Ready for simulation

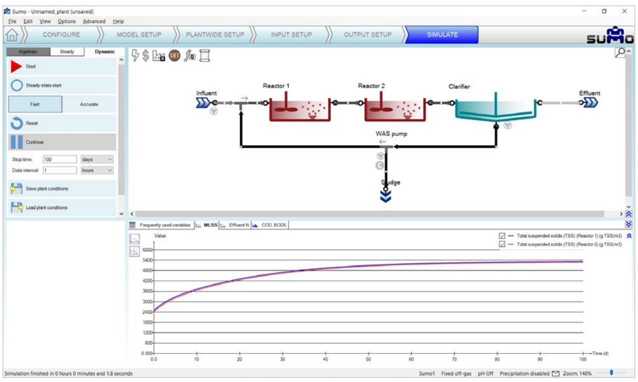

Figure 13 - MLSS timechart results

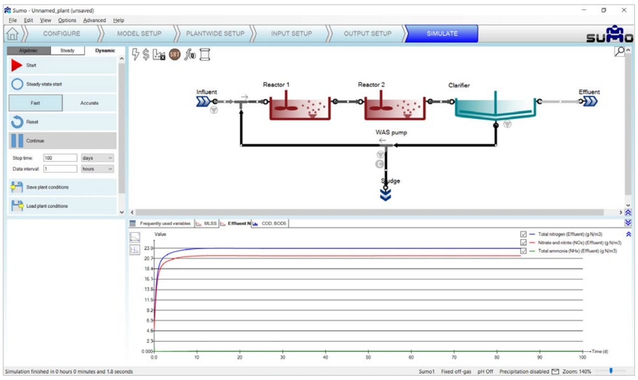

Figure 14 - Effluent N timechart results

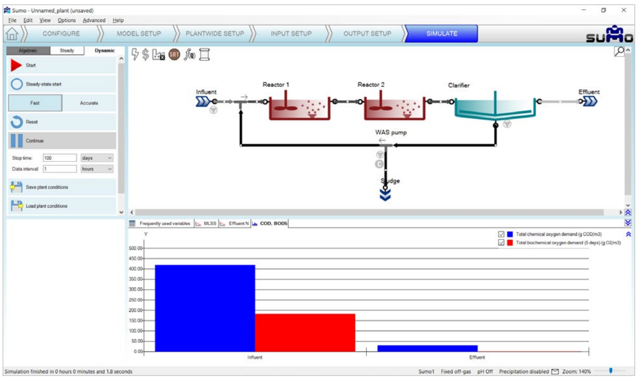

Figure 15 - COD/BOD5 removal results

The steady-state simulation employs different solvers to look for the steady-state condition of the system. In this mode, all dynamic inputs are disabled and the controllers are turned into integrated controllers. The steady-state simulation will find the concentrations in the plant with constant influent and operating conditions, such as monthly average conditions etc. (Figure 16). Please note that steady-state simulation cannot be performed on inherently dynamic processes such as the SBR.

In the *Dynamic* simulation mode, clicking on the *Steady-state start* button will initialize a steady-state run, followed by a dynamic run with the selected *Stop time* and *Data interval* (Figure 17).

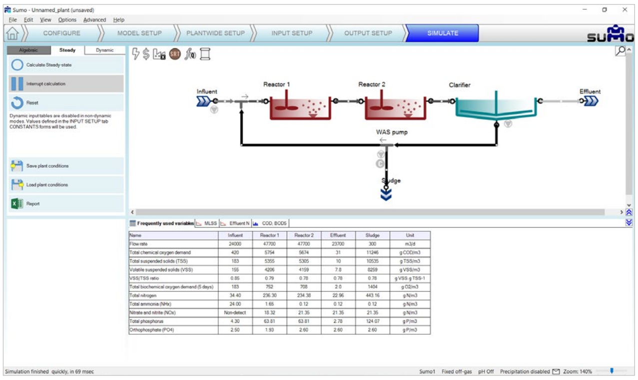

Figure 16 - Steady-state calculation results

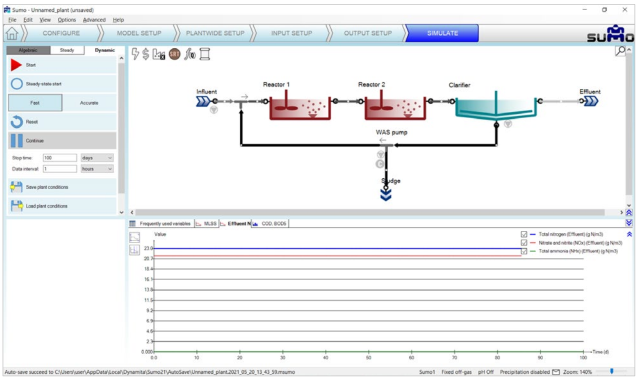

Figure 17 - Steady-state start results

Choosing between *Fast* and *Accurate* modes on the simulation control panel translates to using different pre-configured solver settings. Usually the former is fast and accurate enough for most cases, and therefore its use is generally recommended. However, in certain situations (e.g. with biofilm models), the *Accurate* mode might provide faster simulation. Note that the mode selection will have effect on the steady-state solver as well.

The *Reset* button, which is available in both simulation modes, reinitiates the next simulation with the default concentrations defined in the model and process units. It should be employed thoughtfully, especially when working with complex plants, because reaching steady-state again might be time-consuming in these cases.

Saving plant conditions can be useful to restore system state in case the simulation is driven into an unwanted condition (e.g. accidentally having been reset).

4.1.1 Dynamic Inputs

This ***optional*** task can be used to enter dynamically changing input information, e.g. diurnal flows, DO schedules and more. When choosing the *Input Setup* tab, the blue workflow tab automatically splits into *Constants* and *Dynamics*. (Figure 18). According to this, different types of settings become available: we can set either constant inputs (e.g. fixed influent composition, reactor DO setpoints or volumes as we just did) or dynamically changing inputs (e.g. variable influent flow, composition, shifting reactor DO setpoints etc.).

Being on the *Input setup/Dynamics* tab, we can copy and paste prepared data tables from an Excel file (i.e. Table 2) into Sumo, simply by clicking on the *Paste table from clipboard* button when the desired process unit is selected. This will apply the selected data ranges to the variables or parameters corresponding to the table headers (Figure 18). The latter have to comply with certain rules (for details, please see the User Manual).

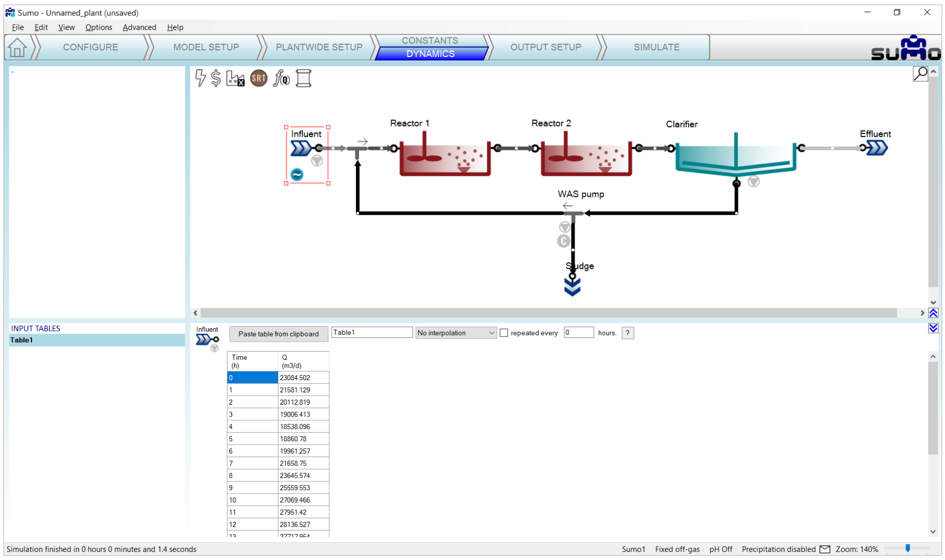

Figure 18 - Dynamic input

Table 2 – Dynamic input of influent flow pattern

Time	Q
h	m3/d
0.0	23084.502
1.0	21581.129
2.0	20112.819
3.0	19006.413
4.0	18538.096
5.0	18860.780
6.0	19961.257
7.0	21658.750
8.0	23645.574
9.0	25559.553
10.0	27069.466
11.0	27951.420
12.0	28136.527
13.0	27717.954
14.0	26916.230
15.0	26012.768
16.0	25269.925
17.0	24859.318
18.0	24817.715
19.0	25042.167
20.0	25325.377
21.0	25421.266
22.0	25122.513
23.0	24328.482

4.1.2 Adding measured data to the charts

Plants do collect information and one important task in process simulation is to compare measured data with the simulation results (and potentially using the information to calibrate the model). Let's assume this plant has Wastage flow data logged every 2 hours. The collected data is shown in Table 3.

Copy this table to the clipboard, then in Sumo, switch to the *Output setup* tab, add a timechart (renaming it to "Wastage") and drop the *Flow rate* for the Wastage pump on the chart (available from *Mass flows in pumped pipe* group). Then right click on the new timechart tab and select *Import data*. The data import is carried out by simply pasting the data from the clipboard (see Figure 19).

In order to have variation in the wastage flow of our plant due to the newly added diurnal flow, let's activate the SRT control. You can do this by switching to the *Input setup/Constants* tab, selecting the brown SRT icon on top of the drawing board, setting a 19-day SRT and turning on the control, as shown on Figure 20.

Table 3 – Measured Wastage flow data for the example plant

Time	Flow (W. pump)
h	m3/d
0	320
3	340
6	360
9	310
12	290
15	300
18	310
21	310

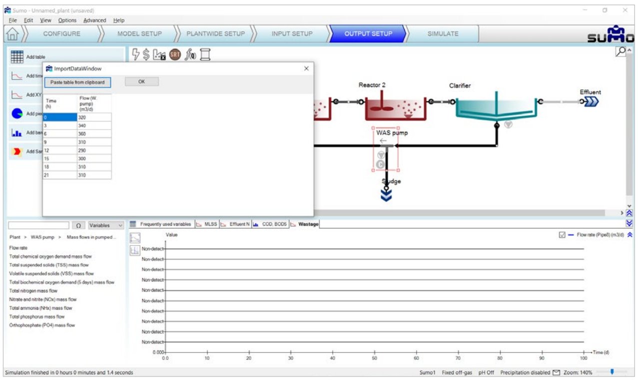

Figure 19 - The data import dialog

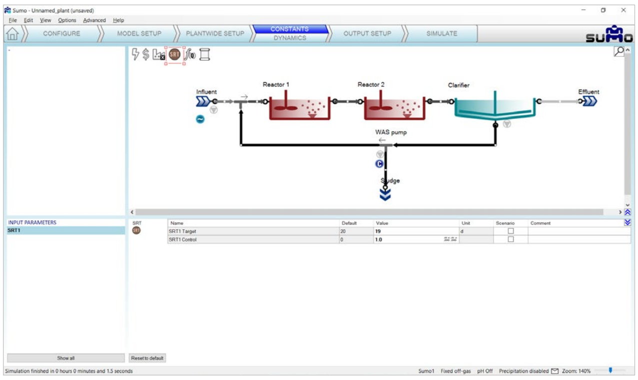

Figure 20 – Activating SRT control

Switch to the *Simulate* tab, set the stop time to 1 day and run a dynamic simulation by clicking on *Start*. Follow the measured and calculated data coherence on Figure 21.

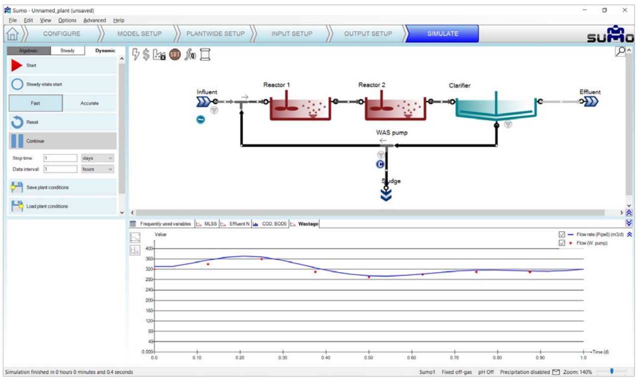

Figure 21 - Wastage timechart with measured data

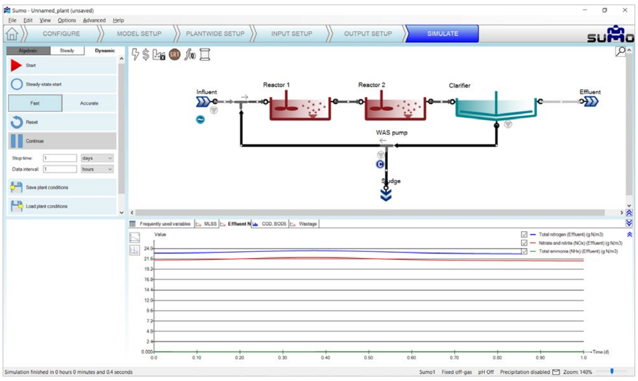

Figure 22 - Effluent N timechart with dynamic input

4.1.3 Writing a report

On the control panel of the *Simulate* tab, a *Report* button is available (you may have to scroll down within the panel in order to reveal it). When clicked, results will be written to an Excel file (saved with the same name as the sumo project file by default). This function creates an Excel file with the following sheets:

- Project overview: contains the configuration layout and basic project information
- Notes: the contents of the *Notes* screen panel from the *Configuration* tab
- Modified parameters: values of all parameters in the project that were changed from default
- Simulation results: all results of the output tabs get saved to separate sheets
- Model: contains all parameter values for the used model (in this example: Sumo1)
- Plantwide calculations: Plantwide, Energy center, Cost center, SRT and Flow dependence settings
- Process Units: all settings and PU parameters are displayed on separate sheets for each unit
- Controllers: all settings and parameters of the controllers employed in the project (if any).

The project can be saved any time during the configuration and project development using the file menu, and reloaded at a later point in time.

Many other operational scenarios can be simulated with Sumo, including more complex reactor configurations and more elaborate operational scenarios. Please see the User Manual and the built-in example layouts.