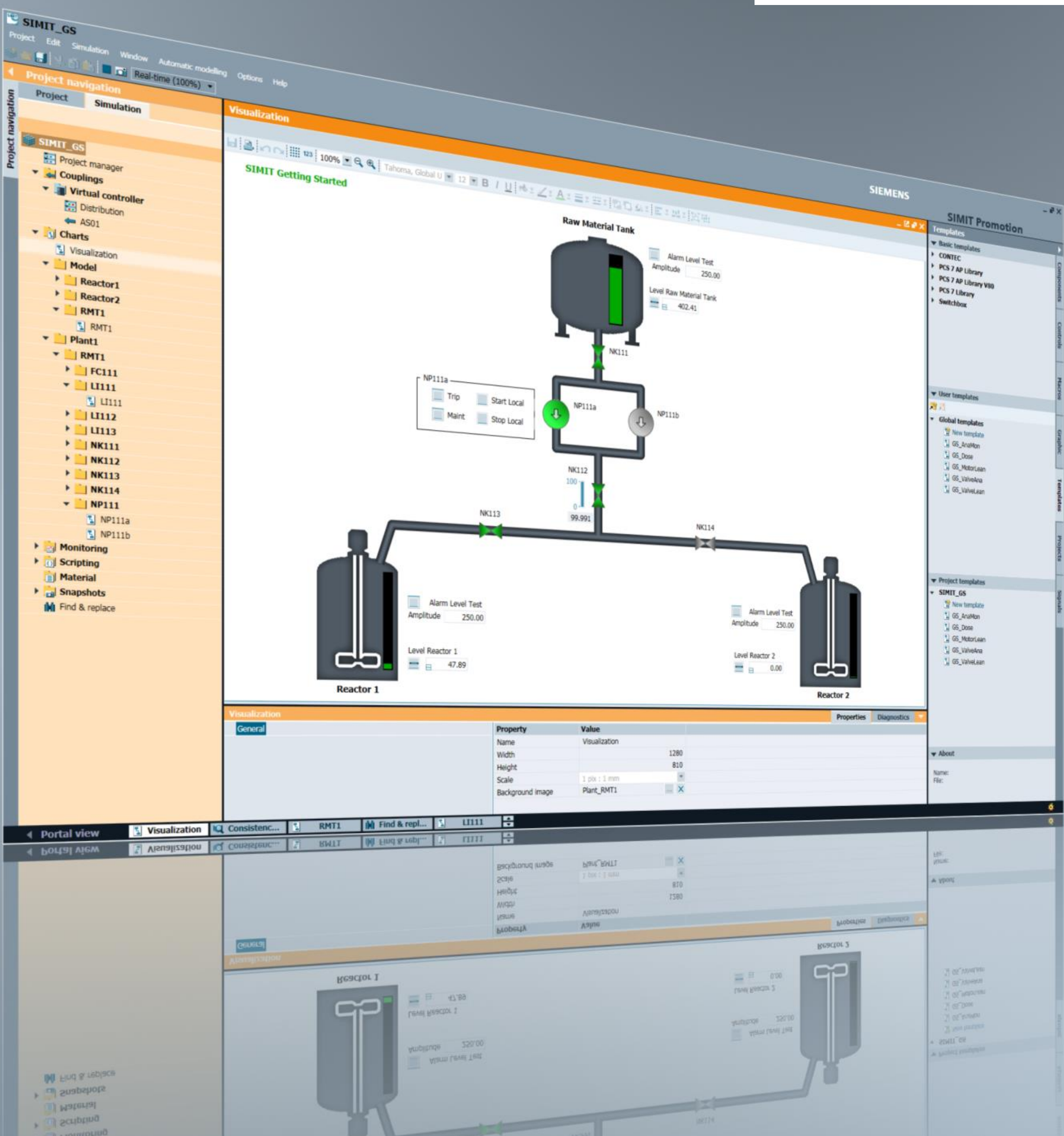
SIEMENS
Ingenuity for life



SIMIT Simulation V9.0 Getting Started

SIMATIC PCS 7 V8.2, SIMIT Simulation V9.0 SP1

https://support.industry.siemens.com/cs/ww/en/view/77362399

Siemens Industry Online Support

Warranty and Liability

Note

The Application Examples are not binding and do not claim to be complete regarding the circuits shown, equipping and any eventuality. The Application Examples do not represent customer-specific solutions. They are only intended to provide support for typical applications. You are responsible for ensuring that the described products are used correctly. These Application Examples do not relieve you of the responsibility to use safe practices in application, installation, operation and maintenance. When using these Application Examples, you recognize that we cannot be made liable for any damage/claims beyond the liability clause described. We reserve the right to make changes to these Application Examples at any time without prior notice.
If there are any deviations between the recommendations provided in these Application Examples and other Siemens publications − e.g. Catalogs − the contents of the other documents have priority.

We do not accept any liability for the information contained in this document.
Any claims against us – based on whatever legal reason – resulting from the use of the examples, information, programs, engineering and performance data etc., described in this Application Example shall be excluded. Such an exclusion shall not apply in the case of mandatory liability, e.g. under the German Product Liability Act ("Produkthaftungsgesetz"), in case of intent, gross negligence, or injury of life, body or health, guarantee for the quality of a product, fraudulent concealment of a deficiency or breach of a condition which goes to the root of the contract ("wesentliche Vertragspflichten"). The damages for a breach of a substantial contractual obligation are, however, limited to the foreseeable damage, typical for the type of contract, except in the event of intent or gross negligence or injury to life, body or health. The above provisions do not imply a change of the burden of proof to your detriment.
Any form of duplication or distribution of these Application Examples or excerpts hereof is prohibited without the expressed consent of the Siemens AG.

Security information

Siemens provides products and solutions with industrial security functions that support the secure operation of plants, systems, machines, and networks.
In order to protect plants, systems, machines and networks against cyber threats, it is necessary to implement – and continuously maintain – a holistic, state-of-the-art industrial security concept. Siemens' products and solutions only form one element of such a concept.
Customer is responsible for preventing unauthorized access to their plants, systems, machines, and networks. Systems, machines, and components should only be connected to the enterprise network or the Internet if and to the extent necessary and with appropriate security measures in place (e.g. use of firewalls and network segmentation).
Additionally, Siemens' guidance on appropriate security measures should be taken into account. For more information about industrial security, please visit http://www.siemens.com/industrialsecurity.

Siemens' products and solutions undergo continuous development to make them more secure. Siemens strongly recommends applying product updates as soon as they are available and to always use the latest product versions. Use of product versions that are no longer supported and failure to apply the latest updates may increase customer's exposure to cyber threats.
To stay informed about product updates, subscribe to the Siemens Industrial Security RSS Feed under http://www.siemens.com/industrialsecurity.

Table of contents

1 SIMIT at a glance

1.1 What is SIMIT Simulation?

Description

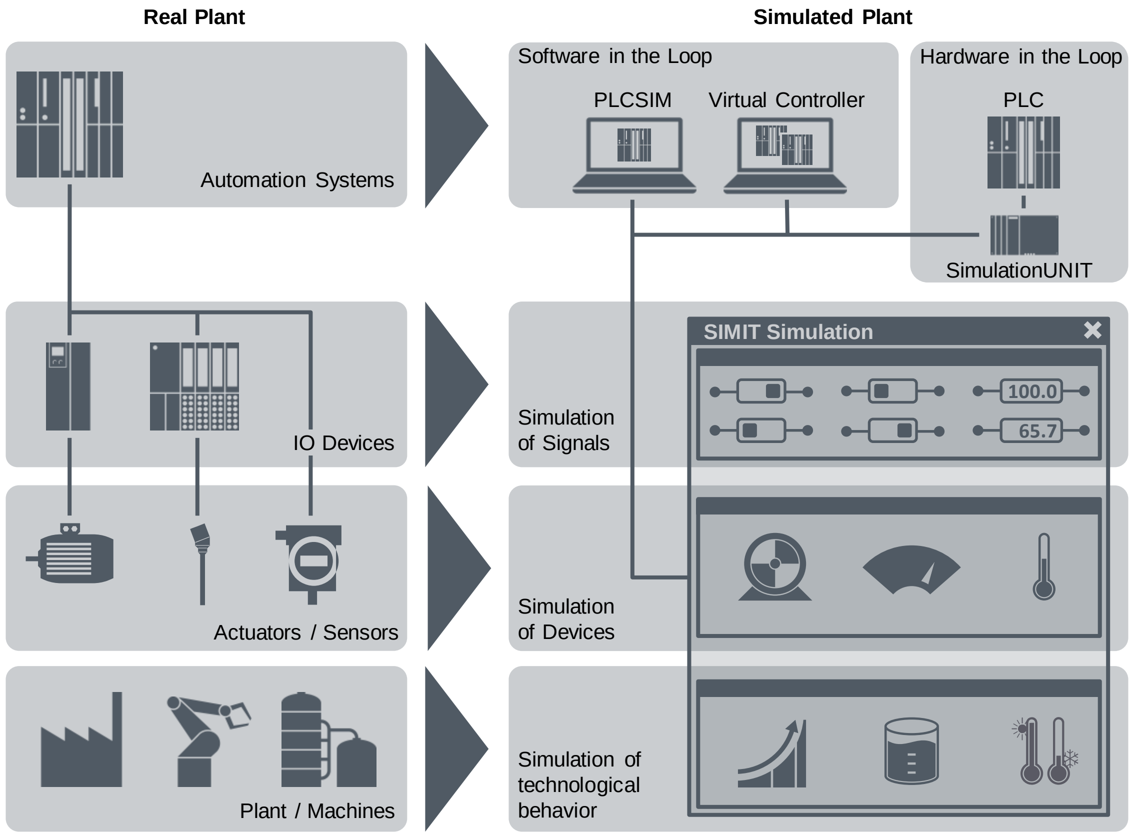
At the engineering stage in particular, the three factors below come to the fore:

- Quality
- Costs
- Time

System individuality and high availability play an ever more important role.
The engineering stage extends far into the operating stage of the system.
This results in projects that are much more complex, a reduction in the time frame for engineering, and increased pressure on each person involved in the project.

SIMIT Simulation (which will be referred to from now on as SIMIT) can contribute to increased cost efficiency. SIMIT is a simulation platform for virtual commissioning of the user software in automation systems. In doing so, SIMIT places a wide spectrum of the plant simulation at your disposal. Starting with simulation of input and output signals, through simulation of field equipment, to modeling of the overall process.

Figure 1-1: Comparison of a real plant and a simulated one



Simulation using SIMIT allows you to test the automation software using real hardware (Hardware in the Loop, or "HiL") or using emulated hardware (Software in the Loop, or "SiL"). The system simulation is created in SIMIT. This simulation mirrors the behavior of the system and is connected to a real controller via an interface module for PROFIBUS, PROFINET, or an emulator.

This means that simulation using SIMIT can bring cost savings and increase the quality of automation projects. Due to its wide spectrum, SIMIT supports you at every stage of your project. It is possible to detect possible sources of error at an early stage and to eliminate them economically during this period of the planning phase. In the same way, faults can be simulated without any risk to staff, the environment, and the plant.

Simulation in SIMIT is subdivided into three levels:

- Signal level
- Device level
- Process level

Signal level

Plant signals are simulated on the signal level. SIMIT offers supporting tools for defining and scaling signals.

You can carry out a functional test in SIMIT as soon as you have created the signals. PLCSIM makes available dialogs for this that you can use to monitor or change signals on the basis of the I/O address. The Virtual Controller does not offer this function; however, using the Virtual Controller, you can also carry out a signal test at an early stage using variable tables in the SIMATIC Manager, for example.

Device level

On the device level, the system maps the actuators and sensors in the system like valves or motors, for example. To do this, SIMIT comes with ready-made components to simulate the behavior of the devices. You can connect them to the signals from the signal level.

Apart from this, SIMIT offers templates and tools that you can use to create actuators and sensors by importing PCS 7 configuration data.

This makes it possible to test run-time behavior (e.g. opening a valve) and feedback (e.g. reaching limit switches). This increases the quality of the simulation and has the advantage that you can implement the automation software in the real plant with a high level of quality assurance.

Process level

The physical model of the plant is represented on the process level. With a variety of library components from the base library as well as the CHEM-BASIC, FLOWNET, and CONTEC libraries that can be purchased in addition to the base library, SIMIT provides you with a platform that allows you to create physical models.

The Component Type Editor (CTE) is an additional tool that gives you a high degree of flexibility. You can develop your own components or adapt existing ones to your own applications.

1.2 Aim of SIMIT Simulation Getting Started

In the course of this Getting Started document, you will get to know the following procedures:

- **Using SIMIT to create simulation models**
  You will learn how to configure and use existing components for your simulation using the SIMIT library. You will also learn how to use SIMIT to visualize your process. For this, SIMIT provides you with an editor that you can use to create the user interface. This means that you can monitor the simulation process and, if necessary, intervene and control it if the situation demands it.
- **Emulating automation systems using the Virtual Controller**
  The virtual controller simulates the behavior of a SIMATIC controller of type S7-300 or S7-400. The virtual controller is loaded with the original PLC user program.
- **Emulating automation systems using PCSIM**
  You will learn how to configure PLCSIM and how to establish communication between PLCSIM and SIMIT.

1.3 Structure of SIMIT Simulation

The SIMIT user interface is divided into the following areas:

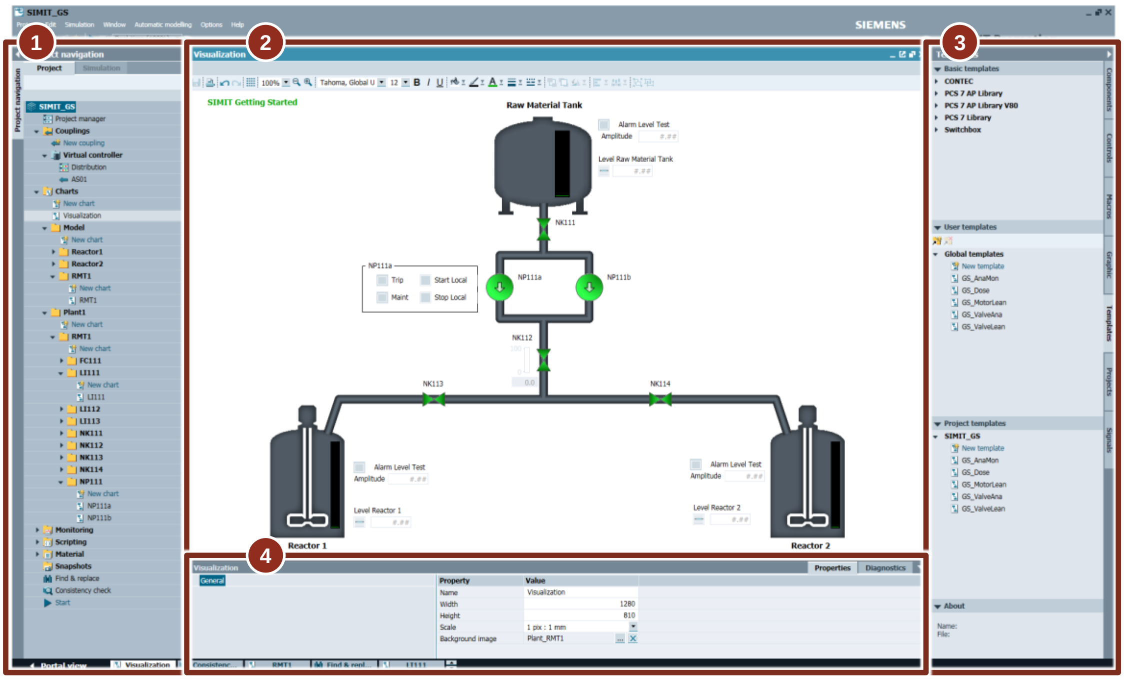
(1) Project navigation

(2) Work area

(3) Tools

(4) Properties

Figure 1-2 The user interface

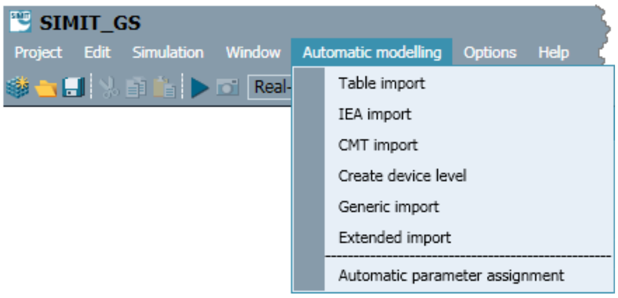


1.3.1 Menu and toolbar

Figure 1-3

In the menu and on the toolbar, you can carry out standard functions like saving and opening projects, for example. Apart from this, you can start simulation runtime.

Figure 1-4



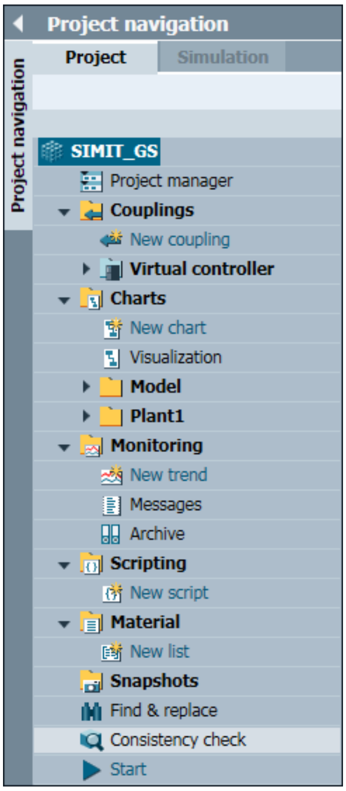
In the Automatic modelling menu item, you can find functions for carrying out efficient configuration of the simulation model.

1.3.2 Project navigation

The Project window displays the currently open project in the Windows Explorer structure that you are familiar with. At the top the system shows the name of the open SIMIT project.

The individual folders always contain a function, e.g. "New chart", and the objects that are created using this function.

When you double-click on an object in the project window, the system either runs the function or displays it in the work area.



In project navigation, you can carry out the following functions:

Table 1-1

Symbol	Name	Description
	Project manager	Opens project view in the work area. The view is the same as the one in project navigation.
	Couplings/ New coupling	In this folder, the system displays the functions for creating couplings as well as all the couplings that have already been created. The following couplings are available: • PROFIBUS DP • PROFINET IO • OPC SERVER • OPC CLIENT • SHM • PLCSIM • PRODAVE • VIRTUAL CONTROLLER
	Charts/ New chart	In this folder, all the charts are stored that contain your plant's simulation models or visualization. You create a new chart by double-clicking on the "New chart" icon.
	Monitoring/ New trend Messages Archive	In this folder, you can find different analysis tools: • Trend Visualization of signal paths • Messages (Displaying and exporting messages) • Archive (Archiving of signals via the simulation duration)
	Scripting/ New script	In this folder, you create scripts. This tool makes it possible for you to access process signals at deterministically set times during runtime. This allows you to intervene in the process on an automated basis and to output information during simulation. You can find additional information on this topic in the "SIMATIC SIMIT (V9 SP1) manual https://support.industry.siemens.com/cs/ww/en/view/109744640
	Snapshots	Using this function, you can take snapshots and save them in the folder of the same name. The saved state of the process can be then loaded at any time using the shortcut menu of the saved snapshot. The simulation is continued from this state afterward.
	Find and replace	Using this function, you can find and replace elements in the project. **Note** Before carrying out searching, save the project; otherwise, the result can be incomplete.
	Consistency check	using this function, you test your project for errors
	Start	You use this function to start the simulation.

1.3.3 Tools

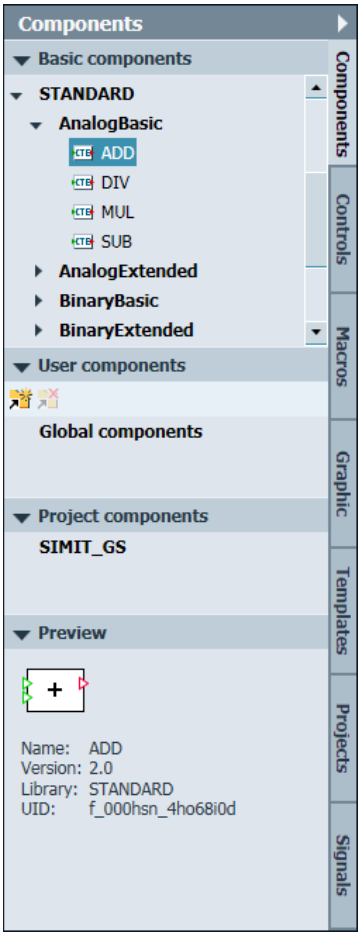
Depending on the object that is open in the working window, different tools are available: They are assigned to the following categories:

- Components
- Controls
- Macros
- Graphic
- Templates
- Projects
- Signals

Components

On the "Components" task card, SIMIT provides a set of libraries to design simulations. The display area is divided up as follows:

- Basic components
- User components
- Project components
- Preview



The basic component is similar to the PCS 7 block library and includes all the libraries provided by SIMIT. This Getting Started document focuses on the libraries: Communication, Connectors and Standard.

On the "User Components" pane, you can store copies of components from the base library. In the library, you can edit the copied component that is available for configuration of your simulation. If you have created your own components, they can also be found in the "User Components" pane.

In order for the created component to be assigned to the project, you must save them on the "Project Components" tab.

Once you have archived the project and then dearchived it, the component is available to you again.

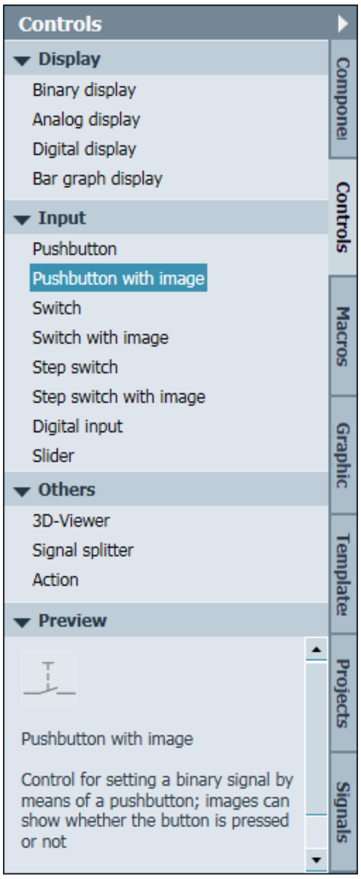
In the last "Preview" pane, the system displays detailed information about the selected components.

Controls

This tab contains all the components you need for input and output of process variables. Using the components from the "Inputs" pane, you can specify process variables at runtime. On the "Display" pane, you can find all the components that you need to display process values.

With SIMIT version 7.1 and above, you can also use 3D objects for visualization on the "Others" tab. To integrate a 3D object, you need an object in ".wrl" format.

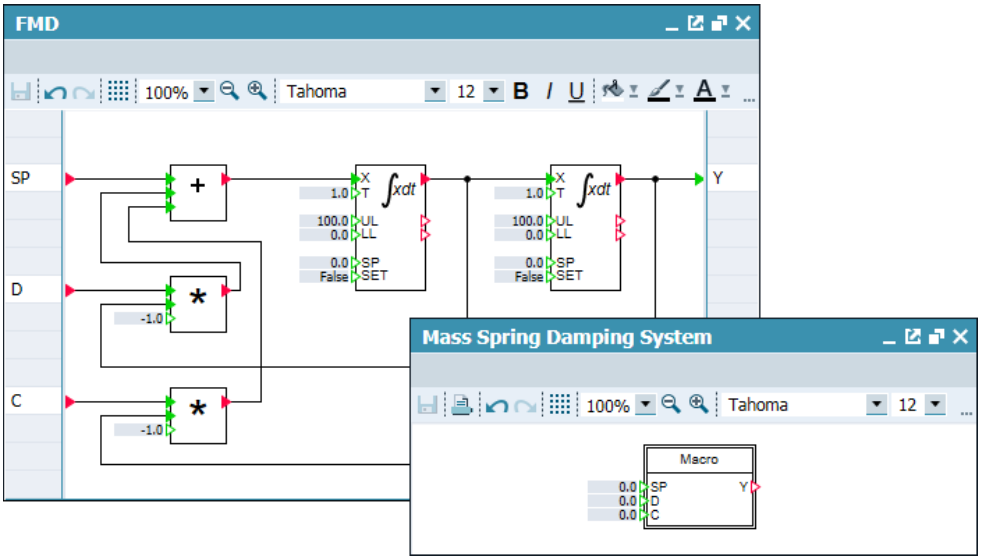
The signal isolator is another element on the "Others" tab. You can use it to disengage signals from processes during runtime. After this, the disengaged signal can be manipulated by means of an input component like the slide, for example.



Macros

Using the macro function, you can combine several sub-components into one entire component. The generated component can be then inserted into a chart. As the figure below shows, a second order differential equation:

Figure 1-5: Macro as a chart (on the left) and as a symbol in the character (on the right)



This makes it possible to group frequently used circuits. You generate inputs and outputs by dragging a line, e.g. from the Y output of the integrator to any cell on the sidebar.

For more information about the functions of the Macro Component Editor, refer to chapter 3.4 of the SIMATIC SIMIT (V9.0 SP1) manual.

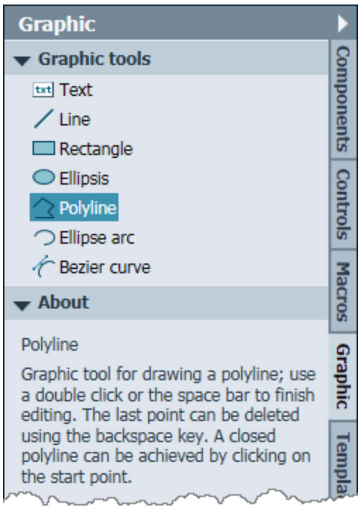
https://support.industry.siemens.com/cs/ww/en/view/109744640/87976571659

Graphic

On the "Graphic" task card, you can find graphics tools that you can use to create graphics in charts.

Using existing graphic elements, you can visualize your simulation in a chart according to your requirements.

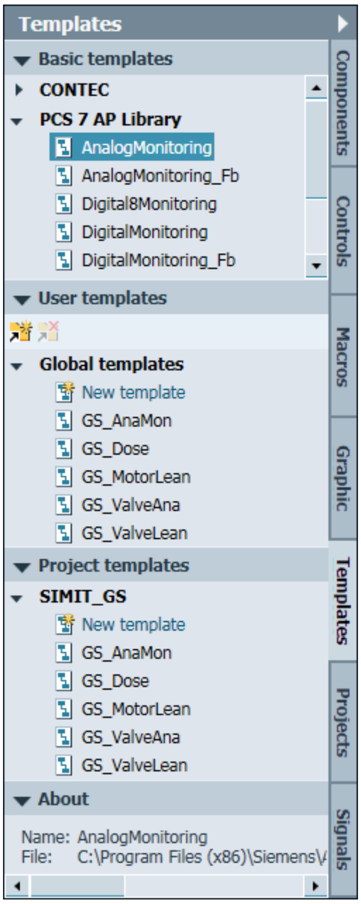
In addition to creating static graphic elements, SIMIT also allows you to create dynamic ones. Apart from process simulation, visualization of processes plays an important role in SIMIT.



Templates

The "Templates" task card consists of the following areas:

- Basic Templates
  Here, you can find the templates that SIMIT provides. For example: components that simulate the behavior of the PCS 7 APL (Advanced Process Library).
- User Templates
  Here, you can create your own components in global libraries. You can use these templates for any projects you like. These templates are not integrated in the project archive.
- Project Templates
  Here, you store your own project-specific templates. They are archived with the project.

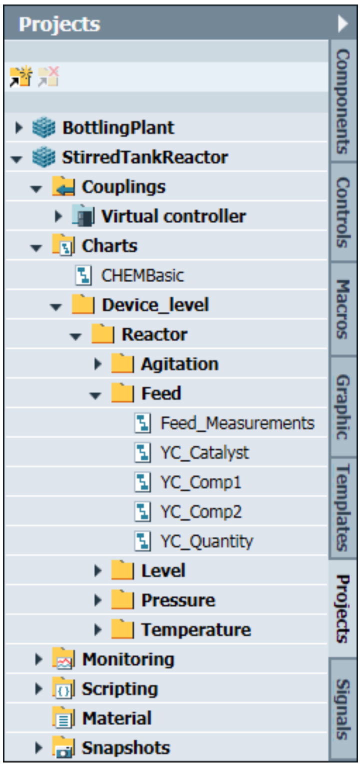


Projects

On the "Projects" task card, you can access the objects and simulation models of existing projects. This allows you to reuse existing tested submodels.

To do this, open the existing project and drag and drop the individual charts into your current project.

After this, you can edit the chart in any way you like or copy it.



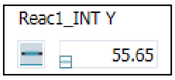
Signals

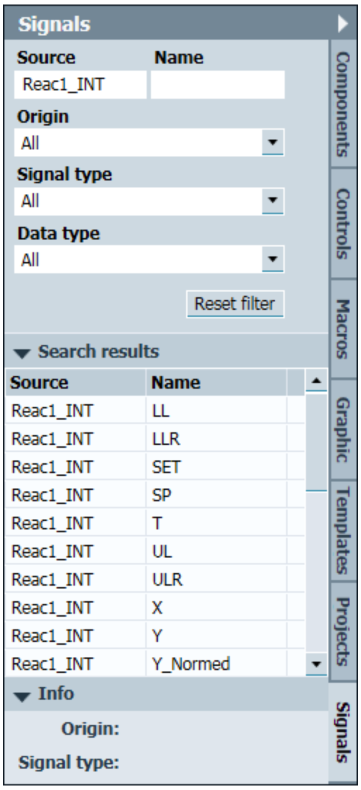
On the "Signals" task card, you can search for all of the signals in the project. A comprehensive filter function is available to make it easier to find signals.

To connect object links to a signal, for example, you can simply drag and drop the object to the link in the properties window.

Apart from this, you can have the system display the values of signals in the chart. To do this, drag and drop a signal onto a free surface in the chart. This creates the objects below and connects them to the signal:

- Text field to the signal name
- Signal isolator to force values at inputs and outputs
- Digital input to display and input a value

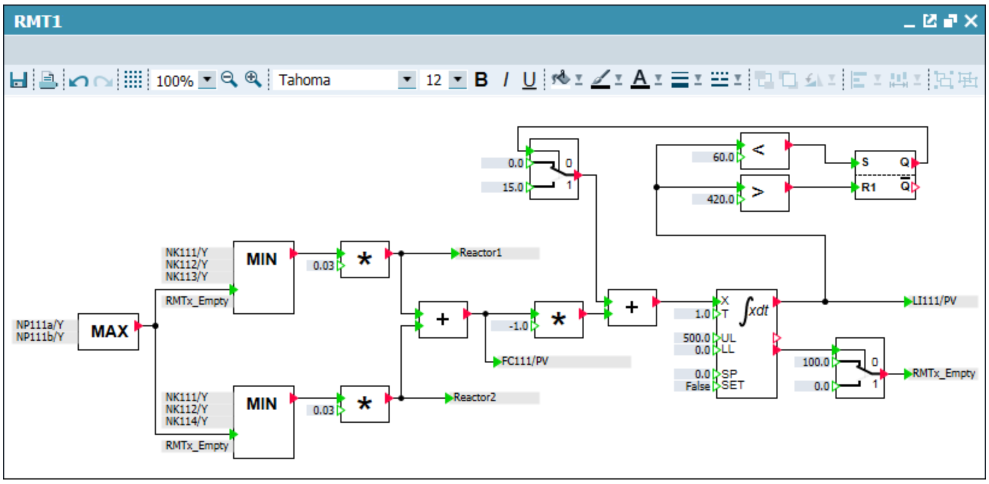




1.3.4 Work area

The system displays all of the SIMIT editors in the work area, e.g. the ones for charts, scripts, or curve displays. You open the appropriate editor by double-clicking on an object in project navigation.

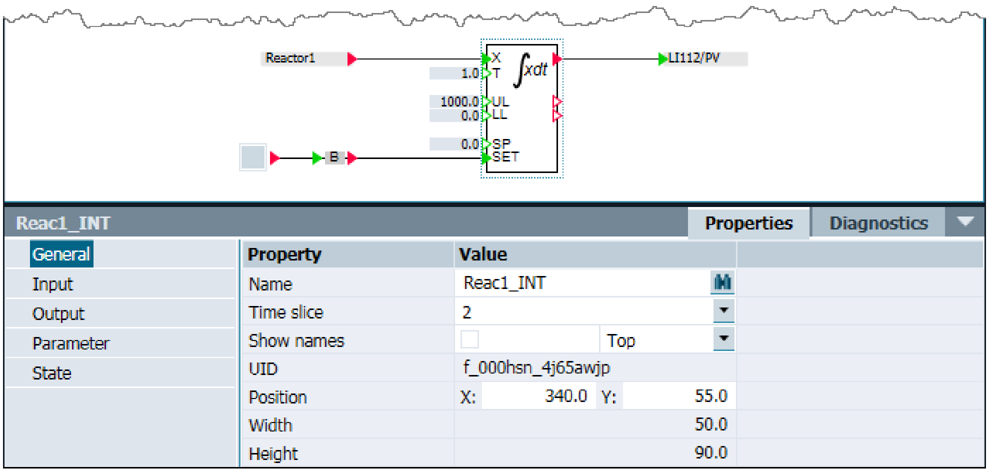
Figure 1-6: An open chart in the work area



1.3.5 Properties

In the properties window, the system displays all of the available properties of the selected object.

Figure 1-7: Properties of a selected object



At the integrator, for example, the properties below are available:

- General properties (object, time slice, position)
- Inputs (values, signnals)
- Outputs (values, signals)
- Parameters (initial value)
- Status

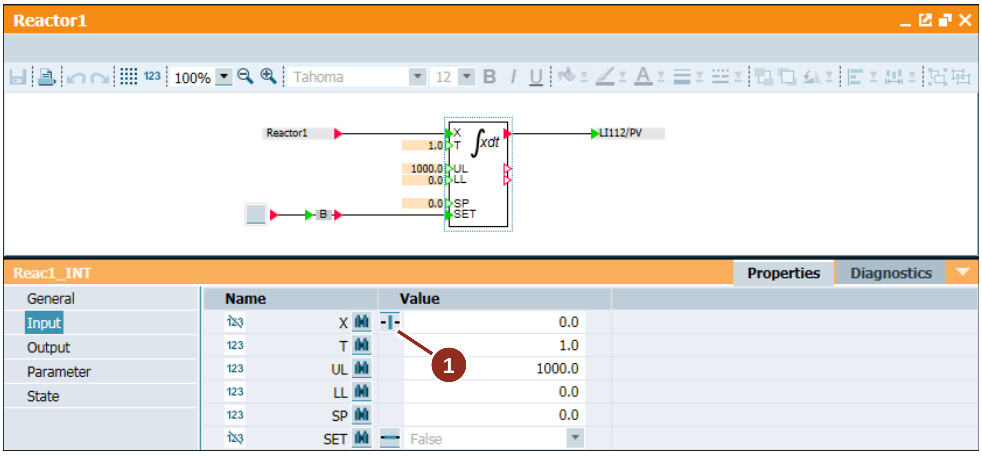
1.4 Changes during an ongoing simulation

In SIMIT, the system shows simulation mode by means of orange title bars or an orange background.

1.4.1 Parameters and values of objects

You can get the system to display the properties of objects by right-clicking on them.

Figure 1-8: Selected object during an ongoing simulation



You can change the values of connections that are not connected with signals at any time you like.

If connections are connected, the system displays the current value. If you disconnect the signal, you can, however, enter a manual (force) value. A signal isolator (1) is available to do this.

1.4.2 Changes to the simulation project

If you want to change something in the configuration of an ongoing simulation, you have the option of switching between simulation view and configuration view. To make changes to the configuration, proceed as follows:

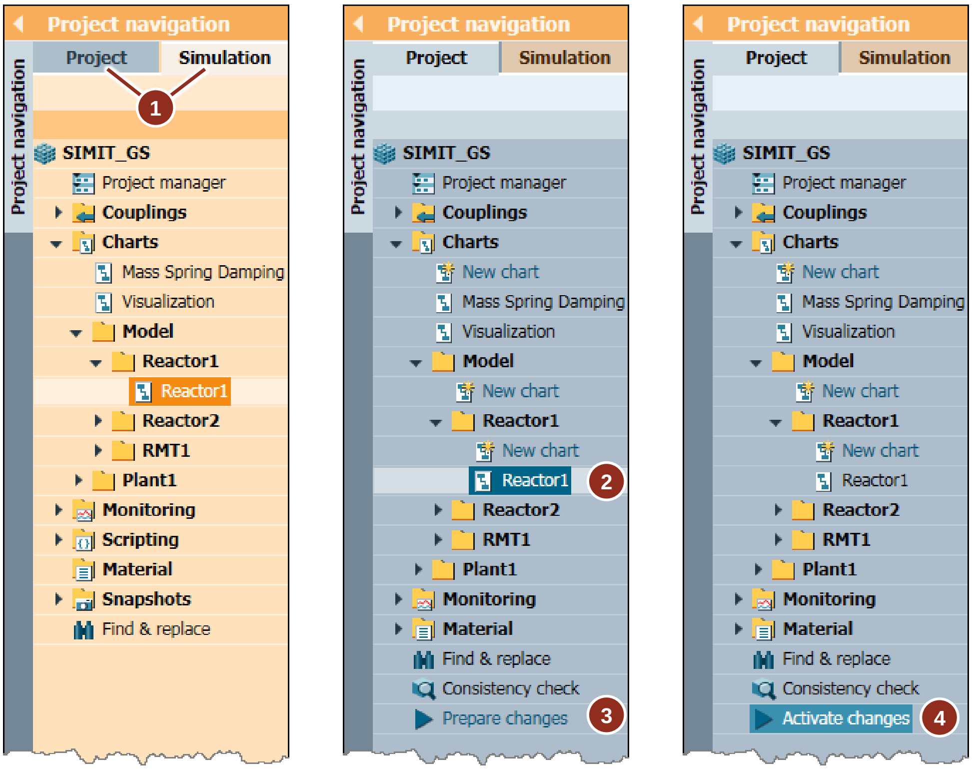
(1) In project navigation, click on the "Project" tab.

(2) Edit the desired object.

(3) Start the "Prepare changes" function.

(4) Start the "Activate changes" function.

Figure 1-9: Changing during an ongoing simulation



2 Presentation of the project

SIMIT Getting Started is carried out using a practical example.

On the PCS 7 Engineering Station, the "SIMIT_GS_MP" is used. The project is based on PCS 7 Getting Started "color_gs" and was created with the following changes:

- The project was updated to PCS7 Version V8.2.
- The "RMT2" subsystem was removed from the technological hierarchy.
- Individual control module types were generated from the process tag types.

Note You can find PCS 7 Getting Started and the associated original project in the "SIMATIC Process Control System PCS 7 Getting Started - Part 1 (V8.2 with APL)" manual:
https://support.industry.siemens.com/cs/ww/en/view/109485954

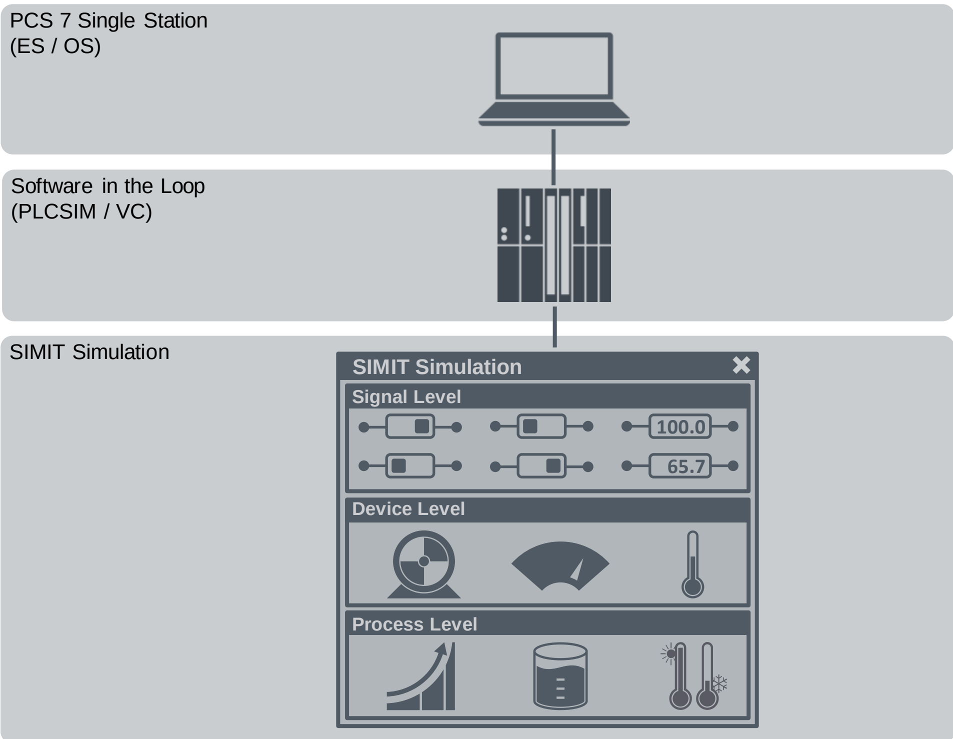
Project design

Getting Started is implementation of on a a PC station. The software below is installed on the PC.

Table 2-1

Software	Article no.	Note
SIMATIC PCS 7 V8.2	6ES7651-5AA28-0YA0	ES Single Station
SIMIT Simulation V9.0 SP1	6DL5260-0BX00-0YL8	SIMIT Simulation Professional
PLCSIM	-	Component of PCS 7
Virtual Controller V9	-	Component of SIMIT

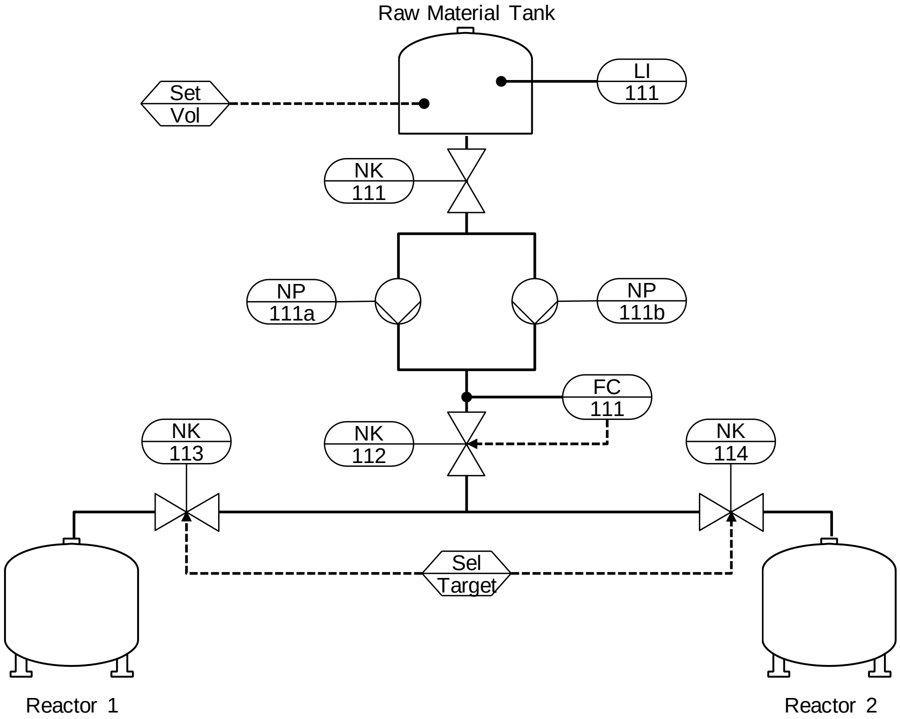
Figure 2-1: Software installation on the test system



Description of the sample system

The liquid basic substance is stored in a raw material tank and pumped as needed from there into one of the two reactors. It is possible to set the amount and the target reactor before starting the transfer sequence.

Figure 2-2: Flow chart of the sample system



The following designations of process tags have been used for the project:

Table 2-2

Abbreviation	Meaning	Note
LI…	Level Indicate	Analog value acquisition (fill level)
FI…	Float Indicate	Analog value acquisition (throughflow)
NK…	Valve	Shut-off valves
NP…	Motor	Drives, pumps
Sel…	Selection	Function for selecting a destination
Set…	Setpoint	Function for specifying a transfer amount

Task for the SIMIT Getting Started

In SIMIT Getting Started, create the simulation model for the "SIMIT_GS_MP" PCS 7 project. You will get to know the procedures below when creating simulation models using SIMIT Simulation:

- Creating a project
- Creating a PLCSIM or a Virtual Controller coupling
- Configuring the input and output signals
- Creating the actuators and sensors
- Using templates and the import function
- Designing the physical models of the raw material tank and the reactors
- Creating a simulation using a macro
- Configuring a visualization

3 Preparing for Getting Started

3.1 System configuration

For communication between the Operator Station (OS) and SIMIT to work, add the configured network addresses to your system. The VC always needs the IP address on the network adapter too. If you are using PLCSIM instead of the VC, you can omit the IP address of the AS. You can leave the settings of network addresses om your PC that have already been configured as they are.

The network addresses below were used in the sample project cartridge:

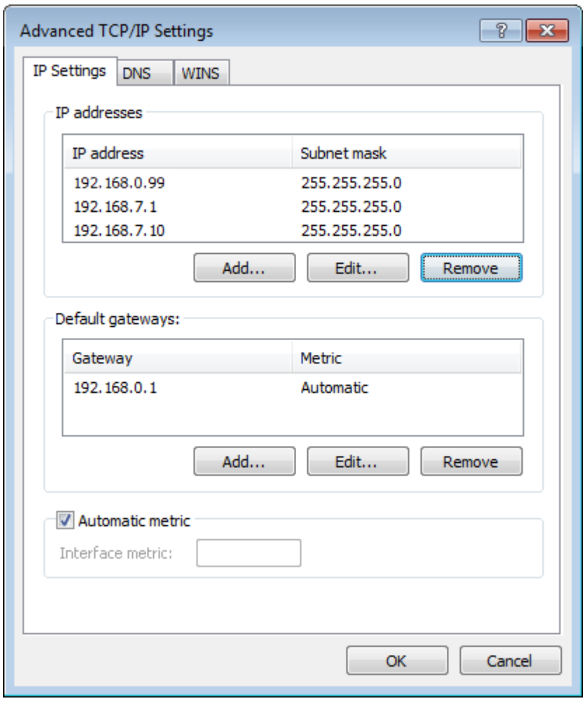
- OS01: 192.168.7.1 (Operator Station)
- AS01: 192.168.7.10 (Automation System)

The VC uses the network address of the AS that is configured in the SIMATIC Manager. To operate the VC, you must assign this address on your network adapter.

To set other IP addresses on your network adapter, proceed as follows:

1. Open the properties of your network adapter.
2. Open the properties of the TCP/IPv4 protocol.
3. In the "Advanced Settings" dialog, add the following IP addresses:
   - 192.168.7.1
   - 192.168.7.10 (when using the Virtual Controller only)
4. Apply the changed settings

Figure 3-1: Advanced TCP/IP settings



3.2 Configuring the demo project

First of all, download the .zip file from the entry page of this application example and unpack it. You can find the entry page by visiting the following link:

https://support.industry.siemens.com/cs/ww/en/view/109485954

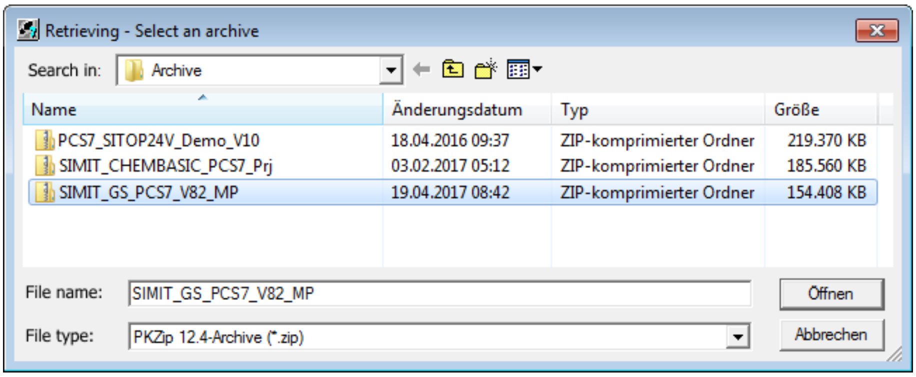
The .zip file contains the following project archives:

- SIMIT_GS_PCS7_V82_MP.zip
  (SIMATIC PCS 7 V8.2 multi-project archive)
- SIMIT_V90SP1_GS_VC.simarc
  (SIMIT Simulation V9.0 SP1 project archive with VC coupling)
- SIMIT_V90SP1_GS_PLCSIM.simarc
  (SIMIT Simulation V9.0 SP1 project archive with PLCSIM coupling)

3.2.1 Configuring the SIMATIC PCS 7 project

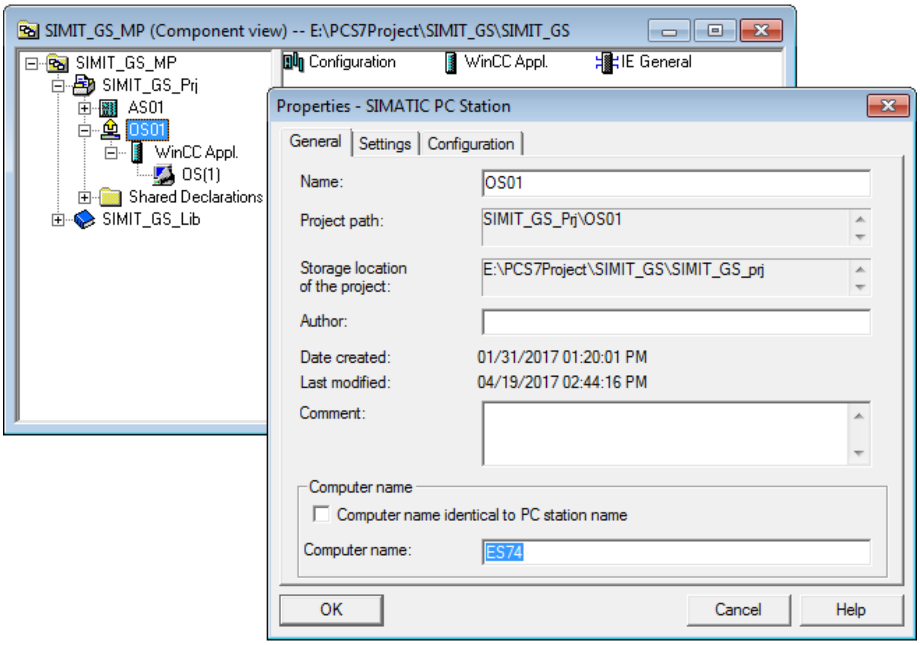
1. Start SIMATIC Manager. Unpack the PCS 7 project archive and open it:

Figure 3-2: PCS 7 project archive



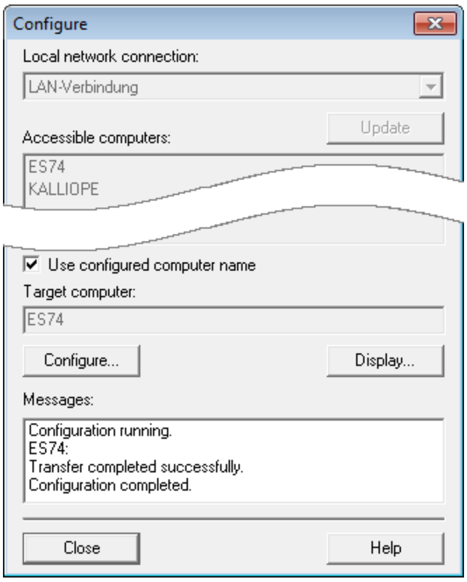
2. Open the properties of the OS station in Component View.
3. Enter the name of your computer in the "Computer name" field.

Figure 3-3: Changing the computer name on the OS station



4. Start the "PLC > Configure…" function

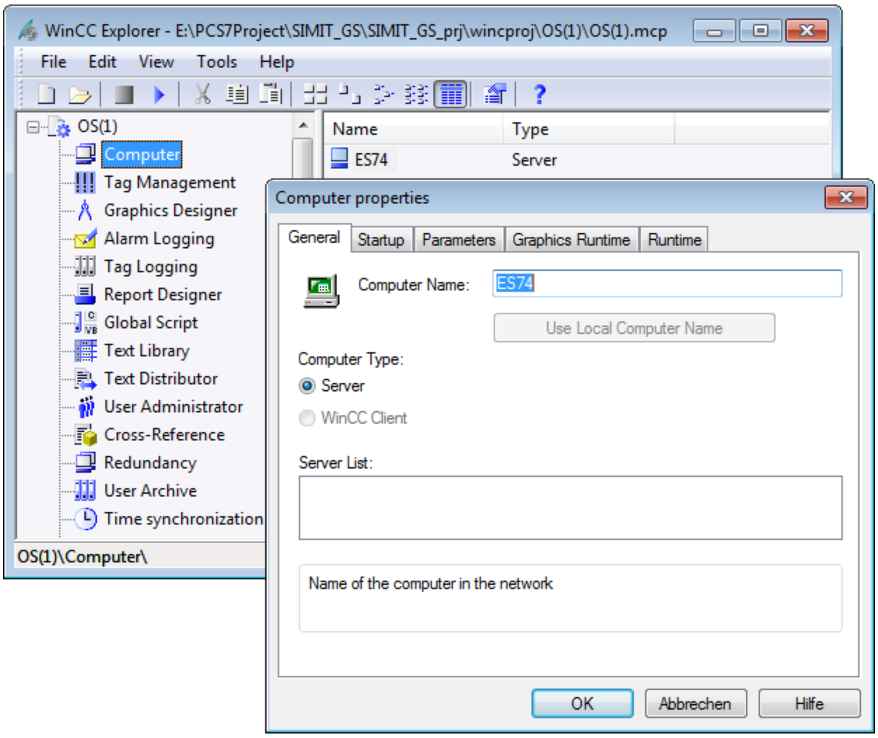
Figure 3-4: Configuring the Station Configuration Editor



5. Use the "PLC > Download" function to load the configuration into the Station Configuration Editor.

6. Open the OS project using WinCC Explorer.
7. Open the properties of the configured computer and enter the name of your PC by clicking on the "Use Local Computer Name" pushbutton.

Figure 3-5: Changes in the OS project



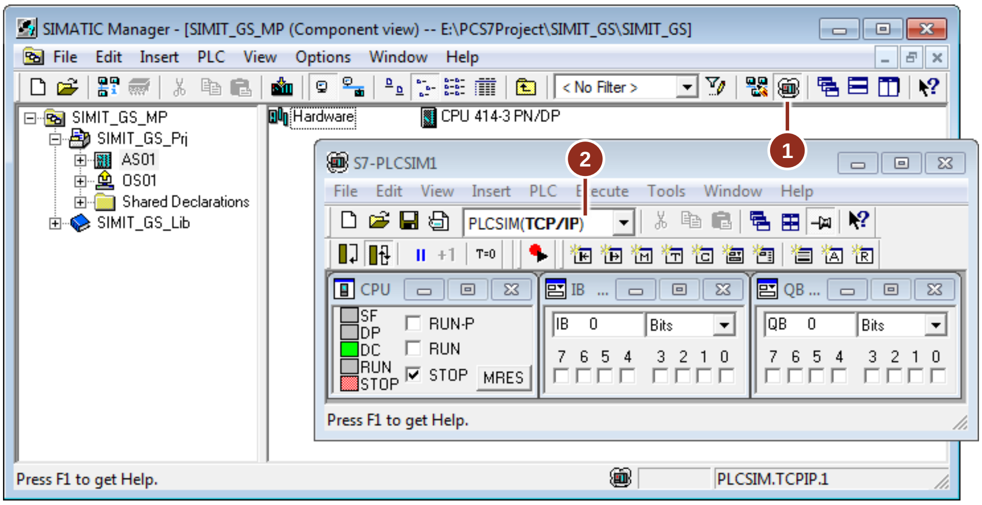
3.2.2 Configuring PLCSIM

Note

Skip this chapter if you are using the Virtual Controller.
To use PLCSIM in the SIMIT environment, you need PLCSIM version 5.4 SP5 or above. For SIMIT to be able to access the data of PLCSIM, SIMIT and PLCSIM must be installed on the same computer.

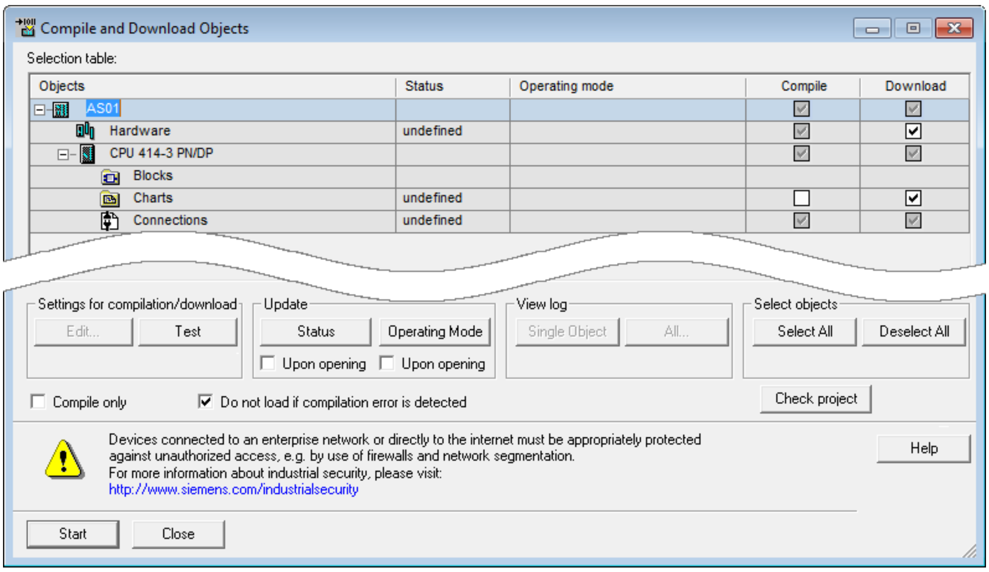
1. Start SIMATIC Manager and open the demo project.
2. Start PLCSIM by clicking on the pushbutton (1) in the menu.
3. In the drop-down list in PLCSIM. choose "PLCSIM (TCP/IP)" (2) .

Figure 3-6: Starting PLCSIM and changing interfaces



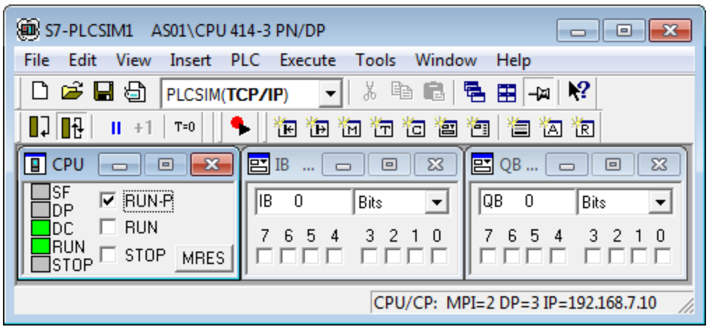
4. Load the configuration and the AS program in PLCSIM. To do this, run the "PLC > Compile and Download Objects…" command in the AS station's shortcut menu.

Figure 3-7 "Compile and Download Objects" dialog



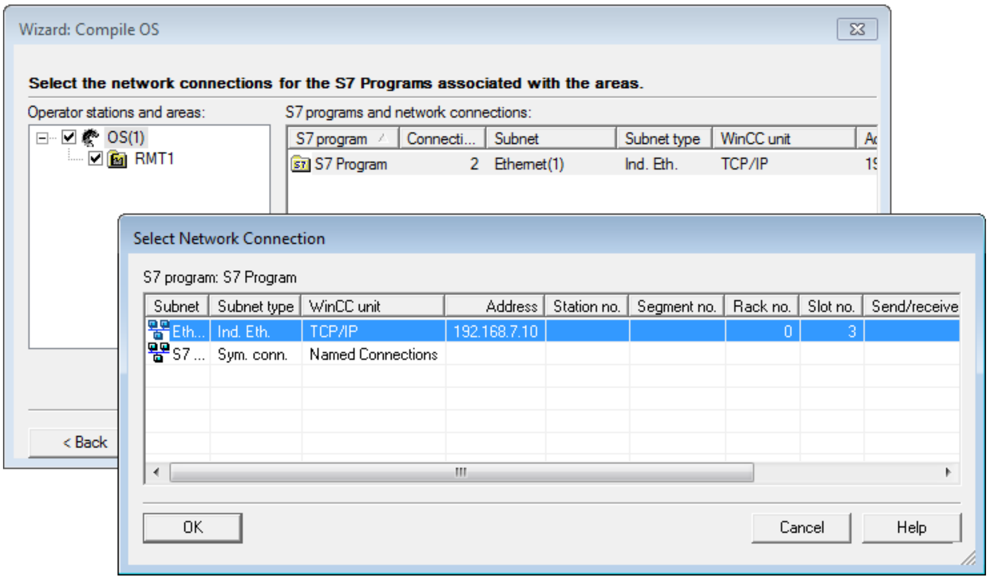
5. Set the PLCSIM to the "RUN" or "RUN-P" operating state.

Figure 3-8: PLCSIM in RUN-P operating state



6. Start the "Compile OS" wizard by running the "Compile" shortcut menu command on OS project "OS (1)".
7. As the third step in the wizard, change the network connection to TCP/IP.

Figure 3-9: Compile OS wizard – network connections



8. The last step is to complete the process by clicking on the "Compile" pushbutton.

3.2.3 Exporting the symbol table

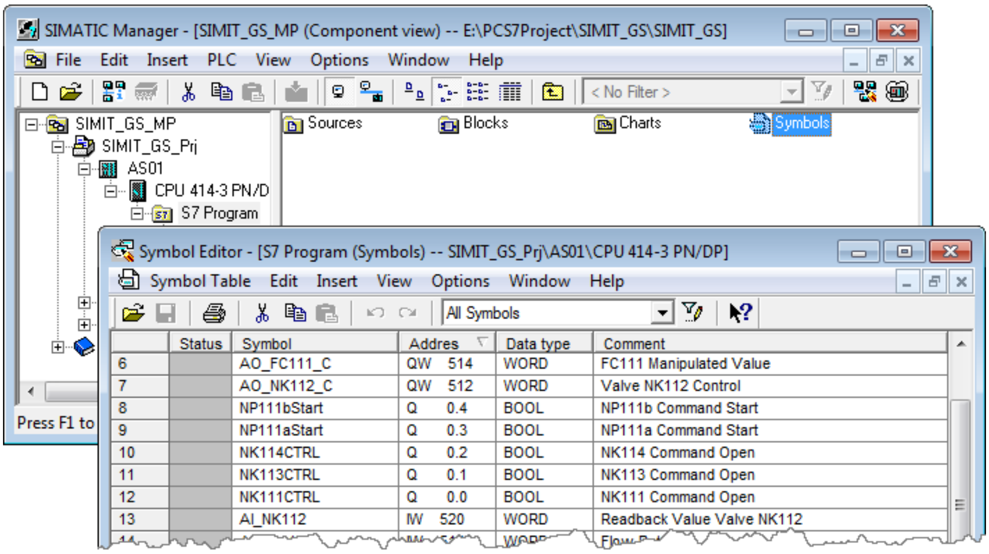
Note Skip this chapter if you are using the Virtual Controller.

SIMIT provides an import wizard for PLCSIM coupling, which allows you to easily import the symbols that are contained in the AS program.

This requires one-off exportation of the symbol table from the AS program.
To do this, proceed as follows:

1. Open the symbol table of the AS program.
2. Sort the table by addresses.

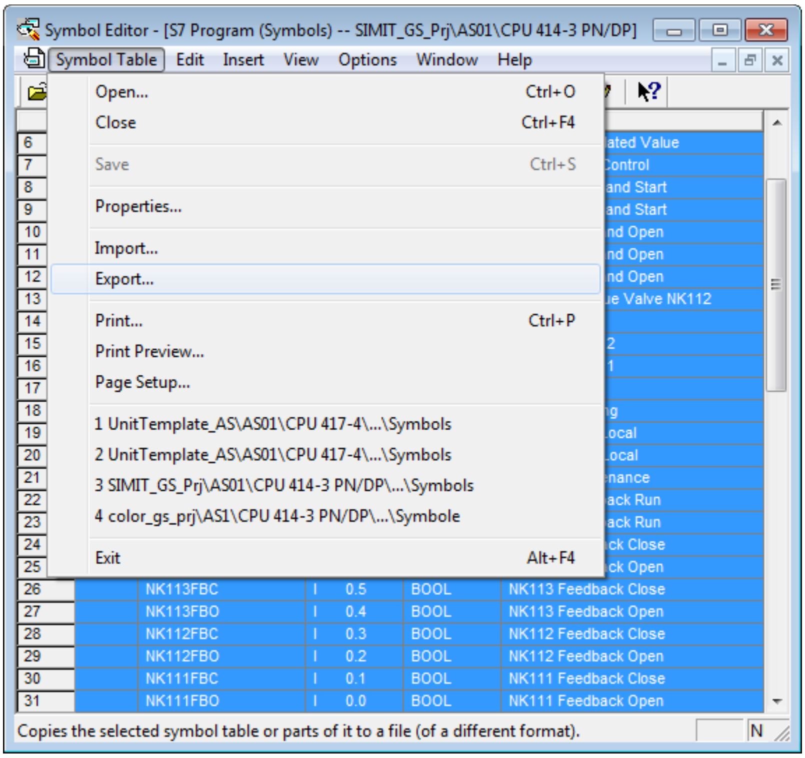
Figure 3-10: Symbol table of the AS program



3. Mark all of the input and output addresses. These will be need later on in SIMIT.

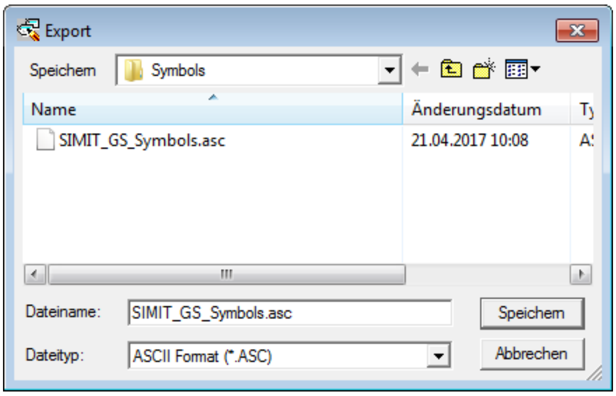
4. Export the symbols using the "Symbol Table > Export…" menu function.

Figure 3-11: Exporting the symbol table



5. Save the symbols in ASCII in a file called "SIMIT_GS_Symbols.asc".

Figure 3-12: Saving the export file



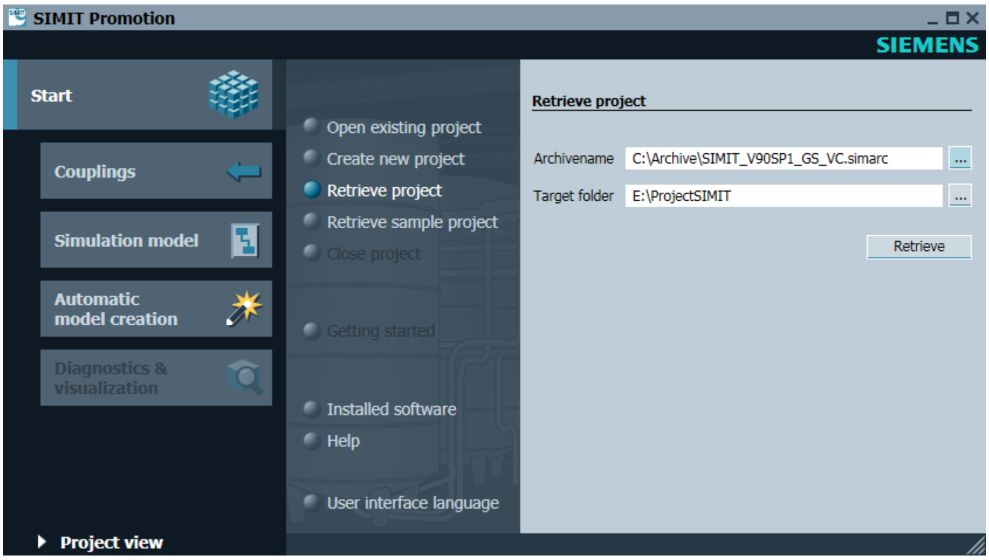
4 SIMIT simulation project

In the download, you will find two SIMIT projects that you can use for training purposes. Further on in the documentation, howver, we will configure the simulation model on a step-by-step basis. You only need these SIMIT projects if you do not want to carry out configuration yourself.

Start SIMIT Simulation Unpack and open the SIMIT project archive containing the coupling that you want to use:

- SIMIT_V90SP1_GS_VC.simarc – for use with the Virtual Controller
- SIMIT_V90SP1_GS_PLCSIM.simarc – for use with PLCSIM

Figure 4-1: Unpacking the SIMIT project archives

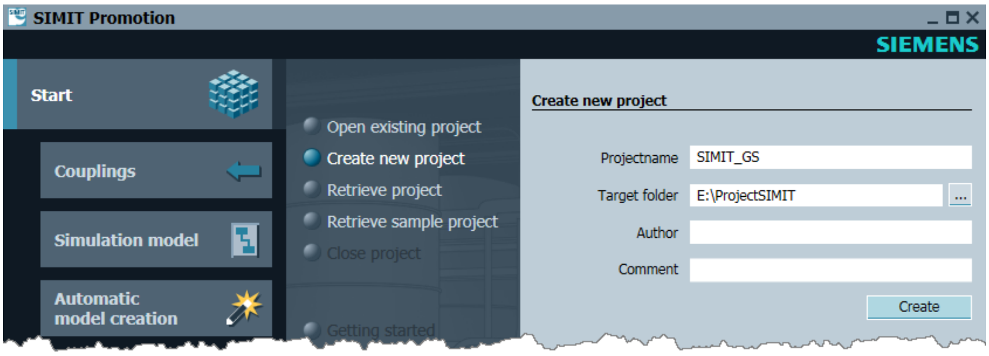


Creating a new project

Follow the steps below to create a new project in SIMIT:

1. Start SIMIT Simulation
2. Choose the "Create new project" option.
3. Enter a project name name, e.g. "SIMIT_GS" and click on the "Create" pushbutton.

Figure 4-2: Creating a new SIMIT project



5 Simulating the signal level

5.1 Configuring the coupling

Couplings are the interfaces between the automation system and the simulation model. When you select the coupling, you specify the communication partner with which SIMIT exchanges data. SIMIT makes available the following couplings:

- PROFIBUS DP
- PROFINET IO
- OPC SERVER
- OPC CLIENT
- SHM
- PLCSIM
- PRODAVE
- VIRTUAL CONTROLLER

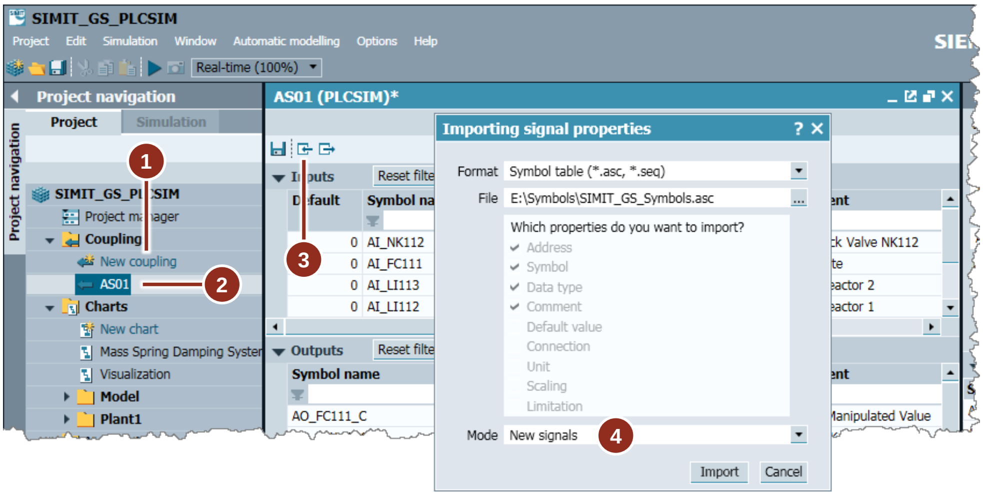
In the rest of this document, we will discuss the "PLCSIM" and "VIRTUAL CONTROLLER" couplings.

5.1.1 Creating a PLCSIM coupling

Note Skip this chapter if you are using the Virtual Controller.

1. Switch to SIMIT project view.
2. In the "Couplings" folder, double-click on the "New coupling" entry (1).
3. In the dialog, choose the "PLCSIM" option.
4. Change the name of the coupling to "AS01".
5. Double-click (2) on the coupling to open it in the work area.
6. Import (3) the input and output signals from file "SIMIT_GS_Symbols.asc".
   See Chapter: "3.2.3 Exporting the symbol table
7. Choose "New signals" mode (4) and start the import process.

Figure 5-1: Importing symbols from the AS program



Communication test with PLCSIM

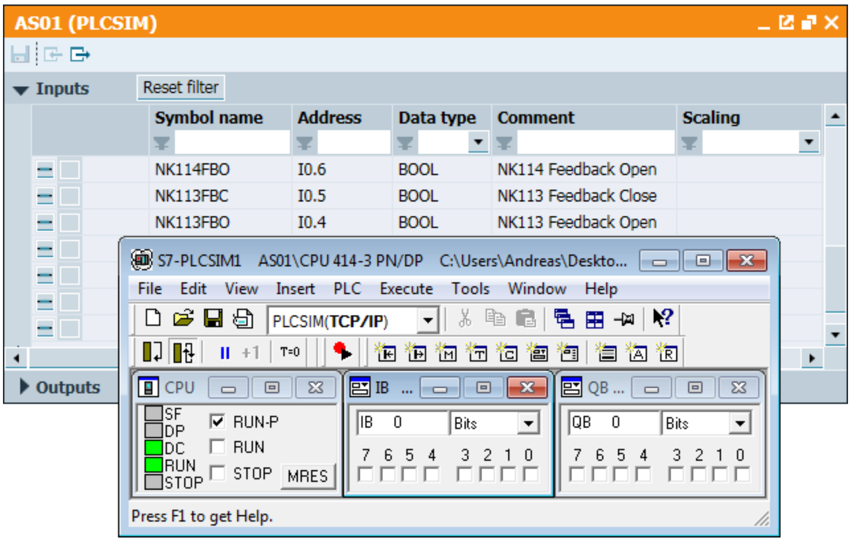
The signals have now been created in the SIMIT coupling. As early as this stage, you can carry out a communication test between SIMIT and PLCSIM.

Note

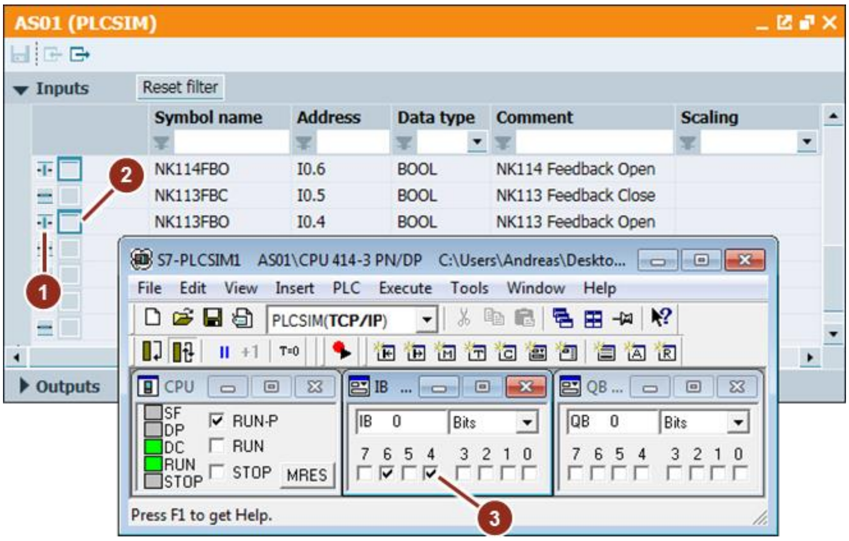
> Input signals in PCS 7 are input signals in SIMIT. In PCS 7, the input signals have reading properties; in SIMIT, they have writing ones.

Proceed as follows:

1. Start the simulation runtime (▶) in SIMIT. PLCSIM must have been started; otherwise, the system issues an error message.



2. Activate the signal isolator (1) so that you can operate the signal manually.
3. Activate a binary signal using the associated switch (2), e.g. "NK113FBO" with address "I0.4".
4. If the connection is working, the system displays the signal in PLCSIM in input peripheral "IB 0 Signal 4" (3).

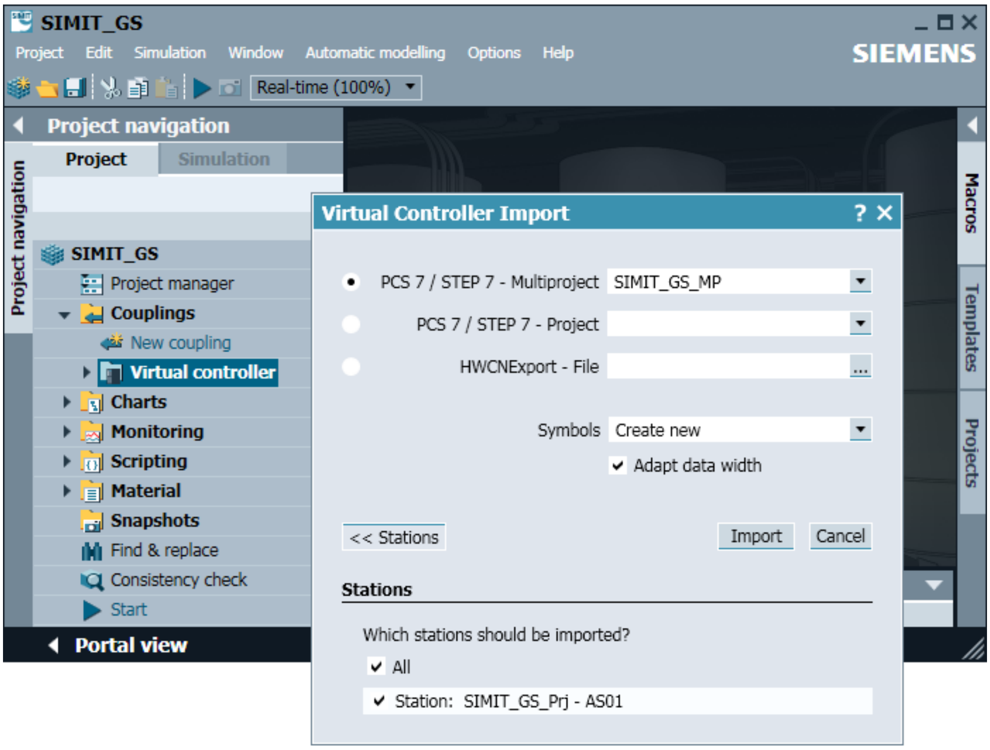


5.1.2 Creating a VC coupling

Note Skip this chapter if you are using PLCSIM.

1. Switch to SIMIT project view.
2. In the "Couplings" folder, double-click on the "New coupling" entry.
3. In the dialog, choose the "VIRTUAL CONTROLLER" option. The system opens the "Virtual Controller Import" dialog.
4. Choose the "SIMIT_GS_MP" PCS 7 multi-project.
5. Click on the ">> Stations" pushbutton to get an preview of all of the created ASes. Here, you can deselect any ASes that you do not want.
6. Click on the "Import" pushbutton.

Figure 5-2



7. Double-click on coupling "AS01" to open it in the work area.

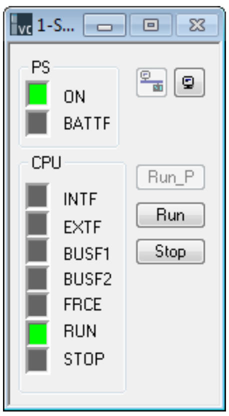
Communication test with VC

All of the input and output signals have now been configured in the coupling. As early as this stage, you can carry out a communication test between SIMIT and the VC.

Note Input signals in PCS 7 are input signals in SIMIT. In PCS 7, the input signals have reading properties; in SIMIT, they have writing ones.

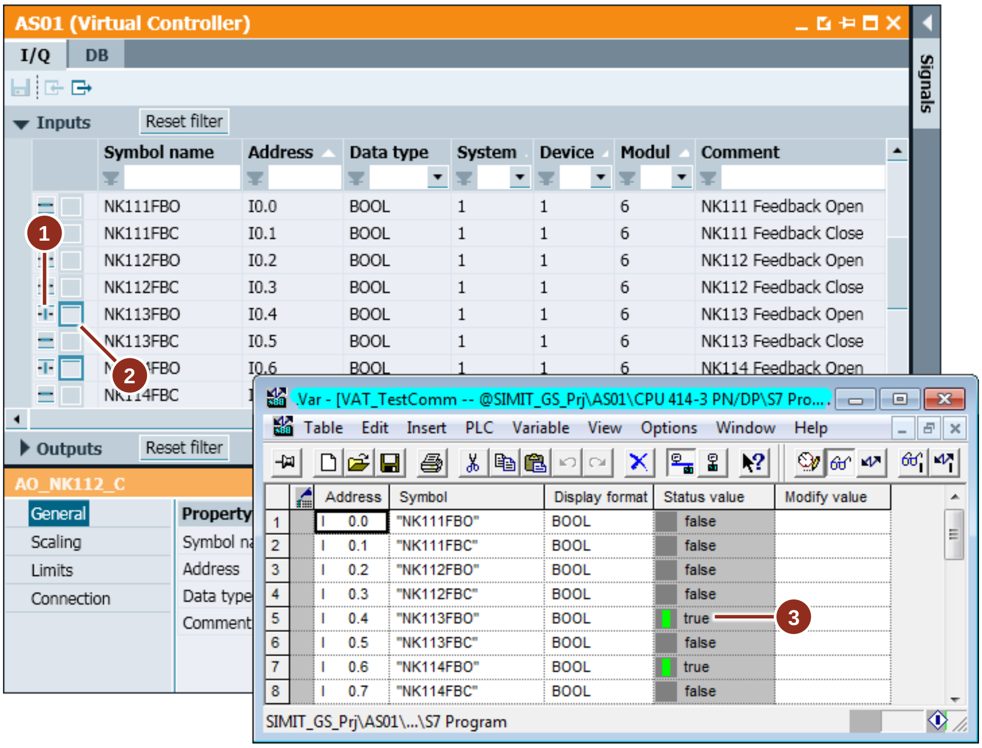
Proceed as follows:

1. Start the simulation in SIMIT (▶). This also starts the VC program.
2. Load the hardware configuration of the AS and the AS program in the VC. To do this, you can use the "PLC > Compile and Download Objects…" function in the SIMATIC Manager.



3. Open variable table "VAT_TestComm" and start the Monitor function (👓). You can find this table in the "Blocks" folder of the AS program in the SIMATIC Manager.
4. Activate the signal isolator (1) so that you can operate the signal manually.
5. Activate a binary signal using the associated switch (2), e.g. "NK113FBO" with address "I0.4".
6. If the connection is working, the system displays the signal in the variable table.

Figure 5-3



5.2 Normalization of the analog signals

In PCS 7, channel blocks, e.g. "PCS7AnIn", convert process values from a raw value to a process value, or "PCS7AnOu", for example, from a process value to a raw value. The raw value is converted into a voltage or current value depending on the configured measurement type. The raw value is usually processed in the PCS 7 program as a 16-bit variable. Depending on the configured digital/analog converter, the least significant bits are truncated so that the number of bits corresponds to that of the analog converter. This means that, in a 12-bit converter, the four least significant bits of the 16-bit variable are ignored.

Data exchange between PCS 7 and SIMIT via the programming interface is performed completely, i.e. the raw value is read or written in 16 bits (corresponds to two bytes). In SIMIT, there are two options for converting a raw value to the corresponding physical value.

- Scaling using standard components
- Scaling in the coupling editor

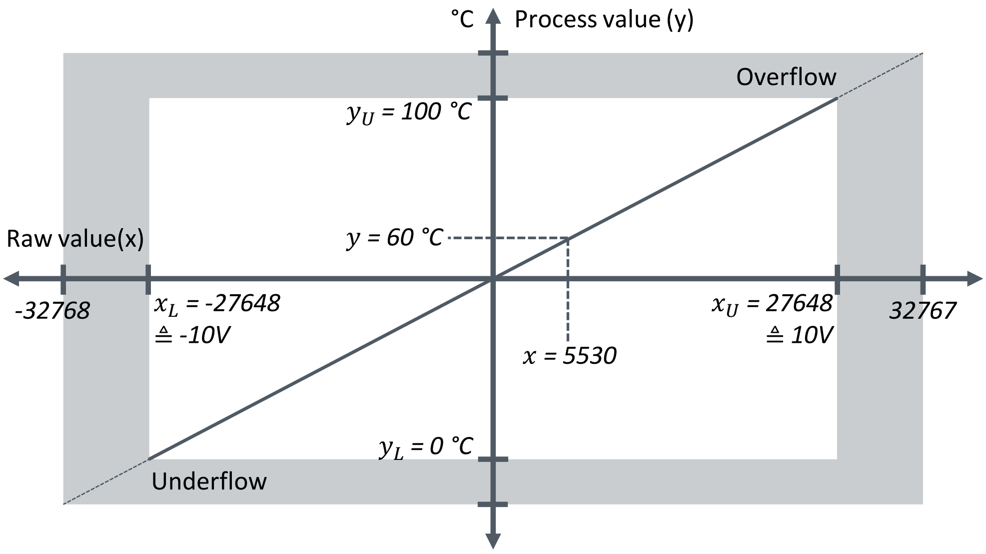
Linear scaling is carried out using the following formula:

$$f(x) = \frac{y - y_L}{x - x_L} = \frac{y_A - y_L}{x_A - x_L}$$

Table 5-1: Tags

Tag	Description
x	Raw value (input value)
x_L	Lower limit of raw value
x_A	Upper limit of raw value
y	Process value (output value)
y_L	Lower limit of process value
y_A	Upper limit of process value

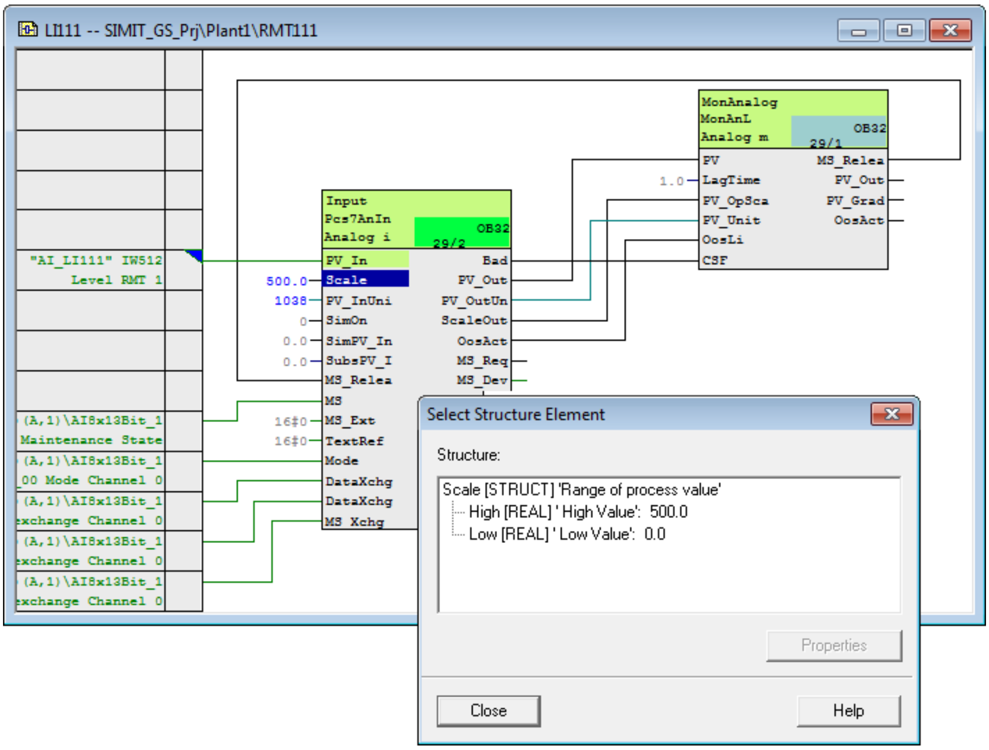
Figure 5-4 Graphic representation of the linear transformation



All of the values in the nominal range between -27648 and +27648 are converted into a process value. In PCS 7, the process values that exceed the nominal range are further converted by the channel block until they are outside the boundaries of the overflow or underflow. In SIMIT however, no further conversion takes place if the raw value exceeds the nominal range. Once the limit is reached, the converted process value remains constant.

To be able to carryout normalization, you must know the process value limits. In the PCS 7 project, these limits are normally configured on the input and output signals. Refer to the appropriate CFC plans to find out the process value limits.

Figure 5-5: Determining the process limit values



In Getting Started, the analog signals below are used with the corresponding limit values:

Table 5-2 Process limit values

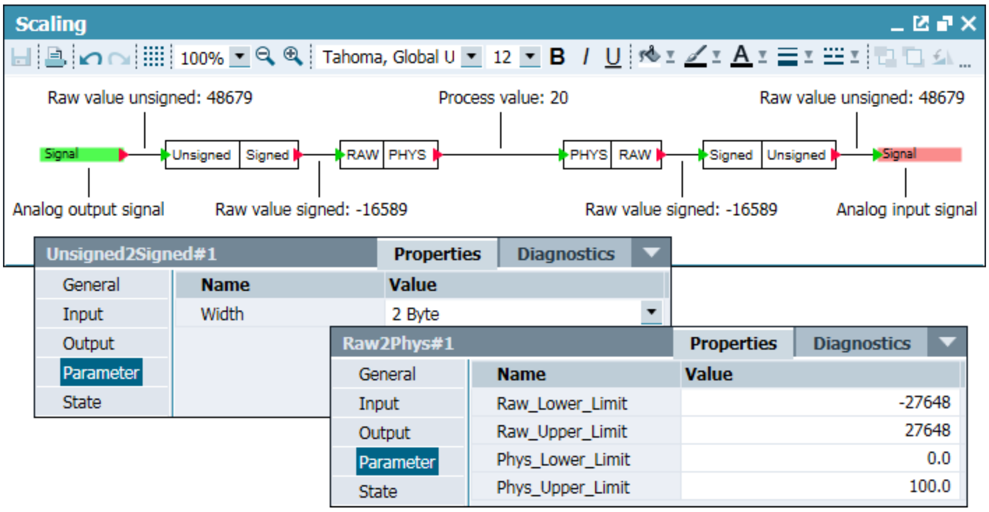
Name	Type	Lower limit value	Upper limit value	Description
AI_LI111	Input	0	500	Raw material tank fill Level
AI_LI112	Input	0	1000	Fill level of reactor 1
AI_LI113	Input	0	1000	Fill level of reactor 2
AI_FC111	Input	0	3	Transfer flow rate
AI_NK112	Input	0	100	Valve position NK112
AO_NK112_C	Output	0	100	Setpoint valve position NK112

5.2.1 Scaling using standard components

First of all, the raw values of the PCS 7 output signals are converted from "Unsigned" to "Signed" and then linear scaling is carried out on them. You convert the values of input signals in the reverse order.

The illustration below shows the scaling structure using components from the SIMIT standard library for a complete chain, i.e. from the output to the input.

Figure 5-6: Scaling of process tags



The "Unsigned2Signed" and "Signed2Unsigned" objects must be parameterized with 2 bytes (16 bits). Objects "Raw2Phys" and "Phys2Raw" must be parameterized with the process value limits. The raw value limits have already been prepopulated.

5.2.2 Scaling in the coupling editor

You can also scale process tags directly in the coupling editor. Due to the number of conversion components that are necessary for type conversion, it is advisable to use this variant. In the rest of the Getting started, however, scaling will be carried out in the chart.

In the coupling editor you can choose the from the following scaling methods:

- Unipolar
  Raw value range: 0 to 27648
  Process value range: Settable
- Bipolar
  Raw value range: -27648 to 27648
  Process value range: Settable
- User-defined
  Raw value range: Settable
  Process value range: Settable

Note

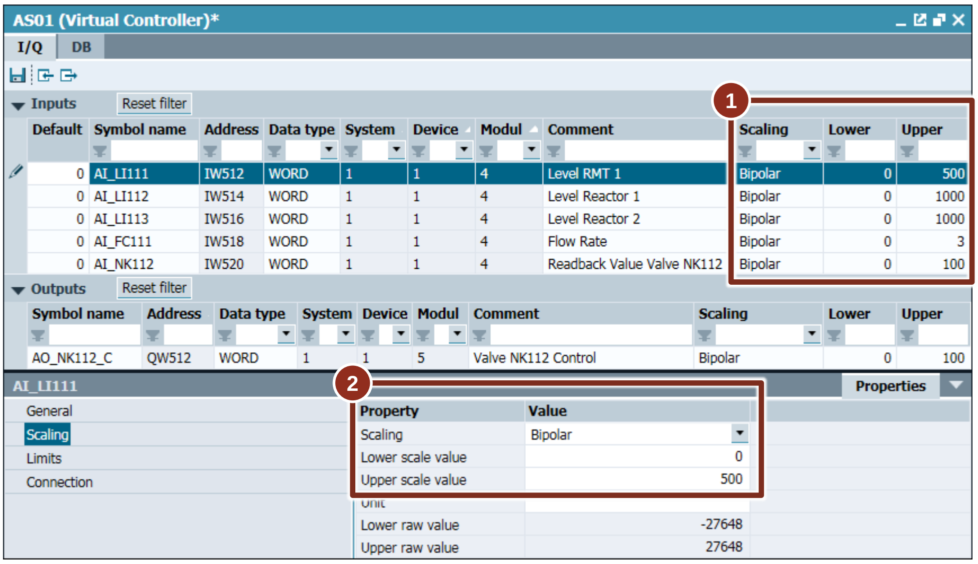
You can select other scaling methods for input signals. You cannot parameterize them either on the raw value limits or on the process value limits and they are preset for special temperature measuring devices. For more information, refer to chapter 2.3 of the SIMATIC SIMIT (V9.0 SP1) manual.
https://support.industry.siemens.com/cs/ww/en/view/109744640/87934457995

The value range is derived from the resolution of the A/D converter, and for SIMATIC S7 is generally around -27648 to +27648. For this reason, we only use bipolar scaling in the Getting Started document.

Proceed as follows:

1. In SIMIT, open the coupling editor by double-clicking on the "AS01" coupling.
2. Parameterize the scaling method and the process value limits. You have the option of setting the parameters in the list (1) or in the properties of the selected signal (2).

Figure 5-7: Scaling in the coupling editor



5.3 Current version

In this chapter, you have configured the coupling to the emulated automation system and imported all of the necessary signals.

As early as this stage, you can carry out signal tests and monitor signal changes in SIMIT by the AS program.

6 Simulating the device level

In the previous chapter, we configured the coupling to the emulated AS. In SIMIT, you have read and write access to the signals.

In a real application, the AS sends the control signals to the actuators, e.g. pump drives or valve actuators or it receives the process values of the sensors, e.g. level gages. In this chapter, you will configure the actuators and sensors.

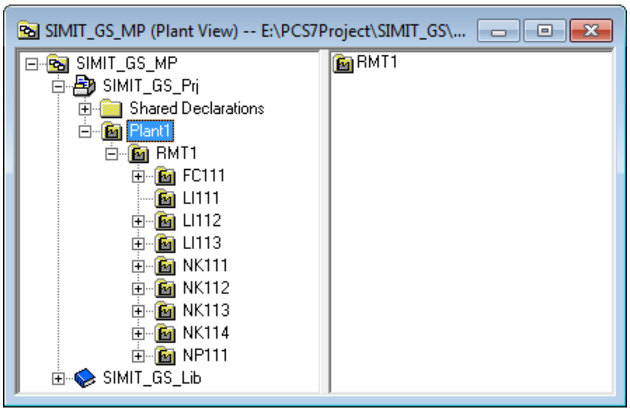
First of all, you create the simulation of a device manually. After this, you get to know the SIMIT help system and wizards that make possible effective configuration.

6.1 Creating the folder structure

The charts for the simulation are created in the "Charts" folder. It is not absolutely necessary to create a folder structure in SIMIT; however, it does give a much clearer overview of the project.

Ideally, you would create the same folder structure in the SIMIT project as in the technological hierarchy in the PCS 7 project. At the latest, this is important when you use the SIMIT import functions.

Figure 6-1: Technological hierarchy in the PCS 7 project

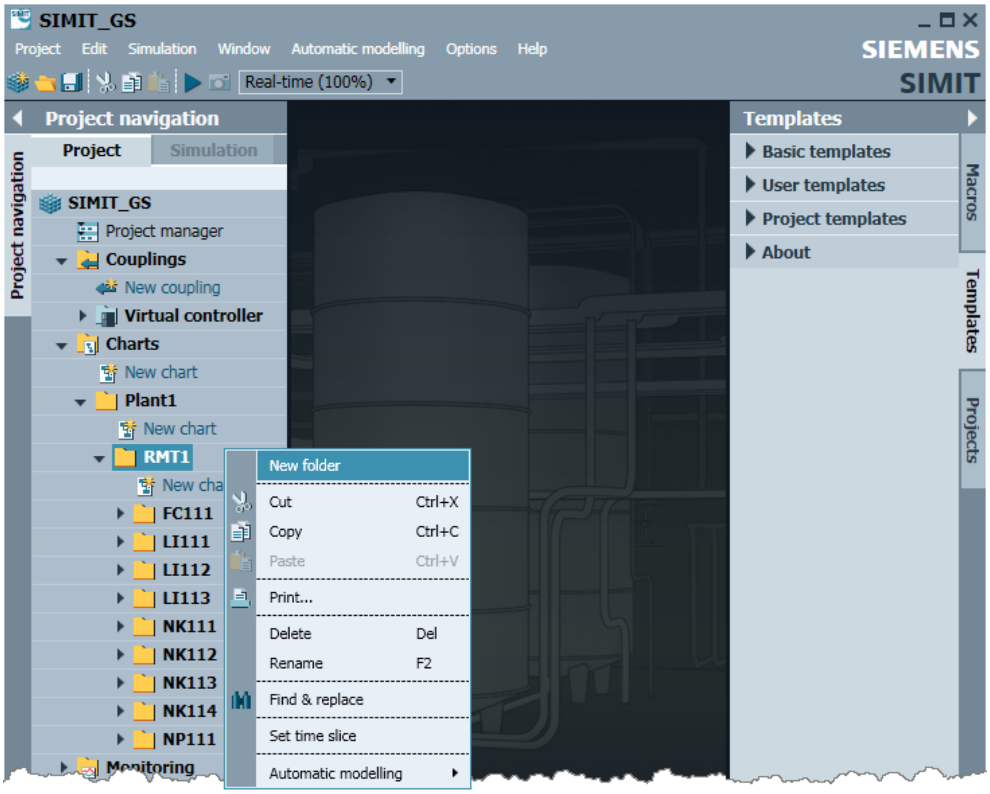


Follow the steps below to create folders in SIMIT:

1. In the shortcut menu of the "Charts" folder, choose the "New folder" item.
2. Change the name to "Plant1".
3. In the shortcut menu of the "Plant" folder, choose the "New folder" item.
4. Change the name to "RMT1".
5. Within folder "RMT1", create the following sub-folders:
   FC111, LI111, LI112, LI113, NK111, NK112, NK113, NK114, NP111.

In each folder, the system automatically creates the "New Chart" function.

Figure 6-2: Creating a folder structure in SIMIT



6.2 Creating your first simulation model

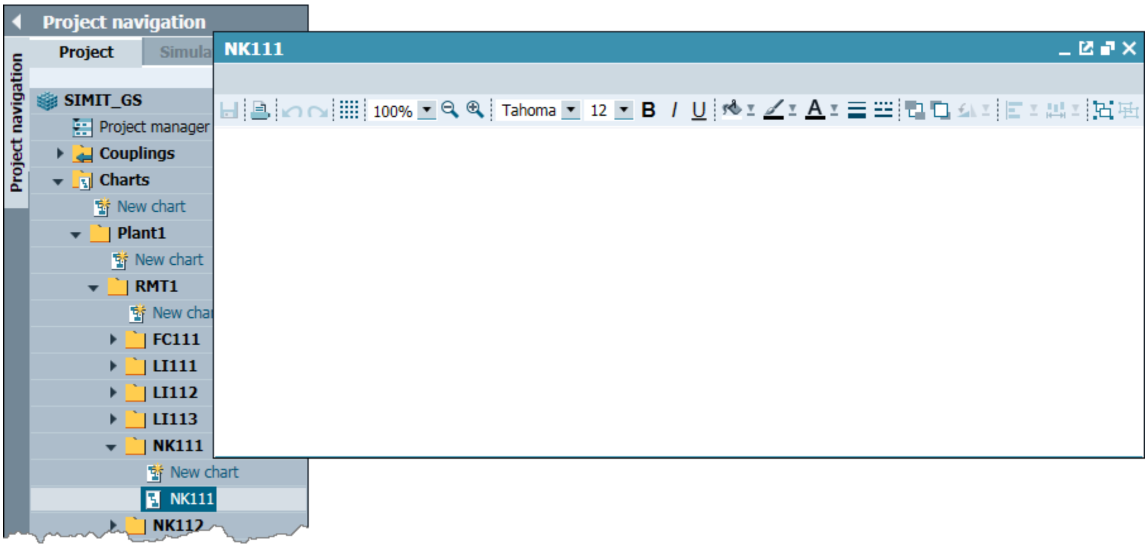
You configure simulation models in SIMIT using charts. The first thing to do is to create the simulation model of valve "NK111".

6.2.1 Creating a new chart

Proceed as follows:

1. Open folder "NK111"
2. Double-click on the "New Chart" function.
3. Rename the chart to "NK111".
4. Open the chart by double-clicking in the work area.

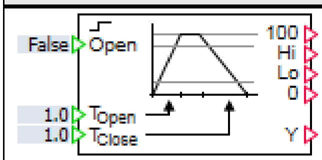
Figure 6-3: New chart

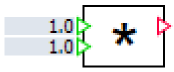
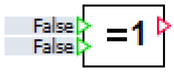


6.2.2 Pasting components

Simulation of the valve actuator needs the following components:

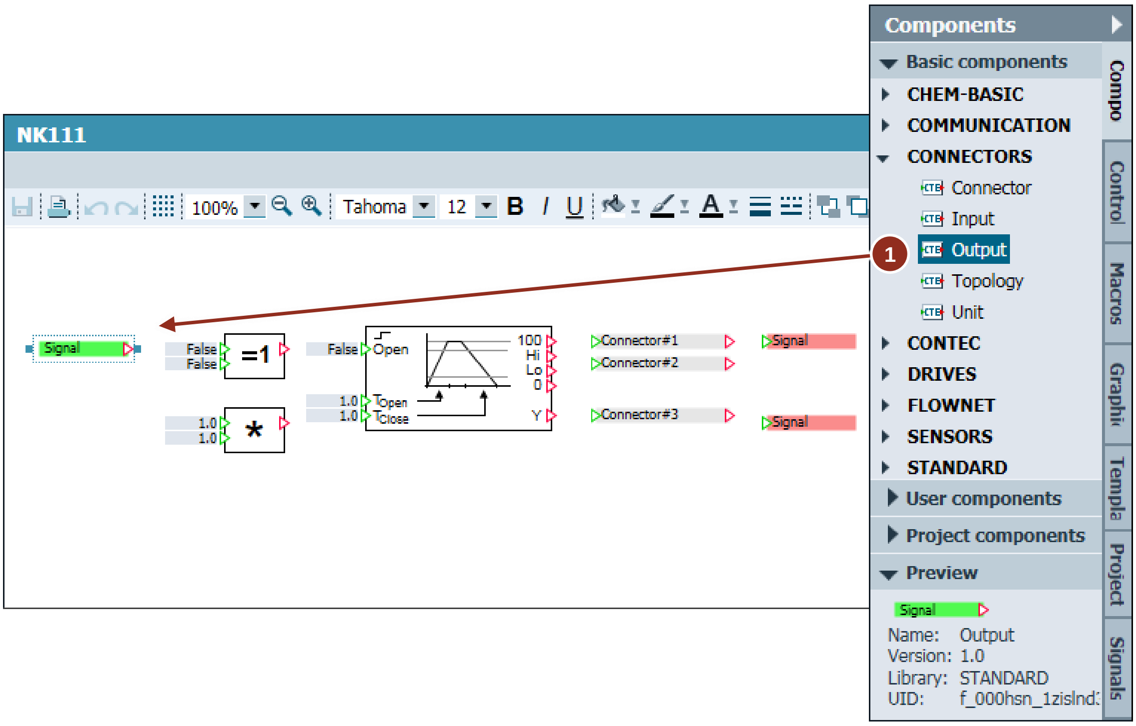
Table 6-1

Symbol	Qty.	Name	Library	Description
		DriveV1	Drives	Simulates the actuator of a valve.
	3	Connector	Connectors	Connectors make connections within the project.
		Output	Connectors	Is connected to the control signal of the AS.

Symbol	Qty.	Name	Library	Description
	2	Input	Connectors	Is connected to the input signals of the AS.
		MUL	Default	Multiplier
		XOR	Default	Exclusive OR

Drag and drop the components (1) that are described in the table to the chart.

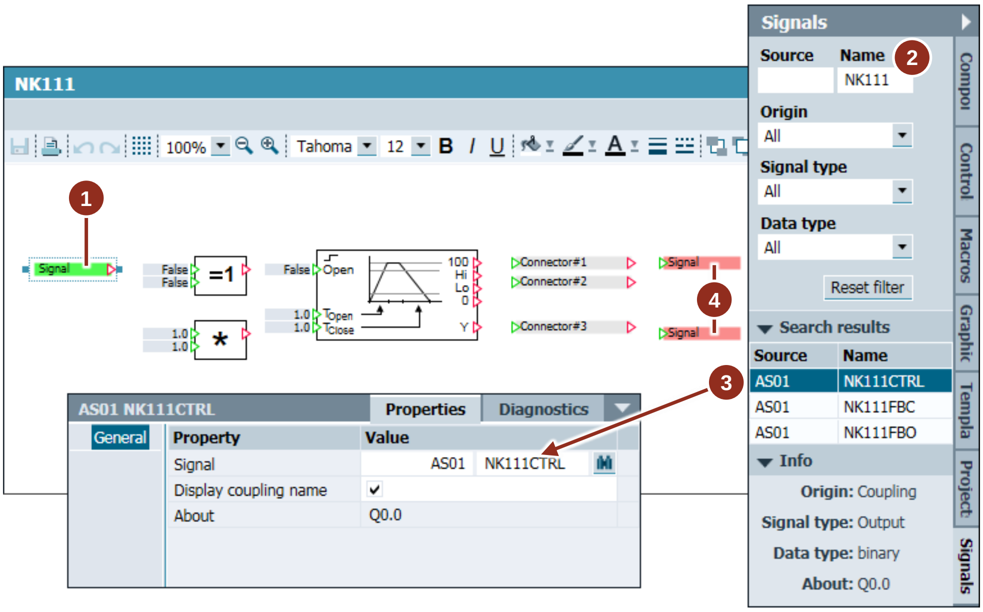
Figure 6-4: Pasting library components



6.2.3 Parameterizing components

1. Highlight the output connector (1). This displays its parameters in the properties window.
2. In the tool window, switch to the "Signals" task card and filter the signals (2) by the name "NK111".
3. Parameterize the output connector with signal "AS01 - NK111CTRL". This is the control signal of the AS program for opening and closing the valve. Drag the signal (3) from the signal list onto the "General > Signal" property of the output connector.
4. Highlight the input connectors (4) and parameterize signals "AS01/NK111FBO" and "AS01/NK111FBC" in the same way. These two signals give feedback about the valve setting.

Figure 6-5: Configuring the input and output connectors



Note	Several options are available to configure the "input" and "output" components: 3. The value of the signal is entered manually. This method is not recommended, since, on the one hand, you need to know the name and, on the other hand, typing errors can occur that must be eliminated later on. 4. You drag and drop the value of the signal by means of the "Signals" task card. 5. You drag the signal from the "Signals" task card while holding down the Shift key and drop it onto the chart. The system automatically creates the appropriate "Input" or "Output" component. This is the most effective and advisable method.

5. Parameterize the other components as described in the table below:

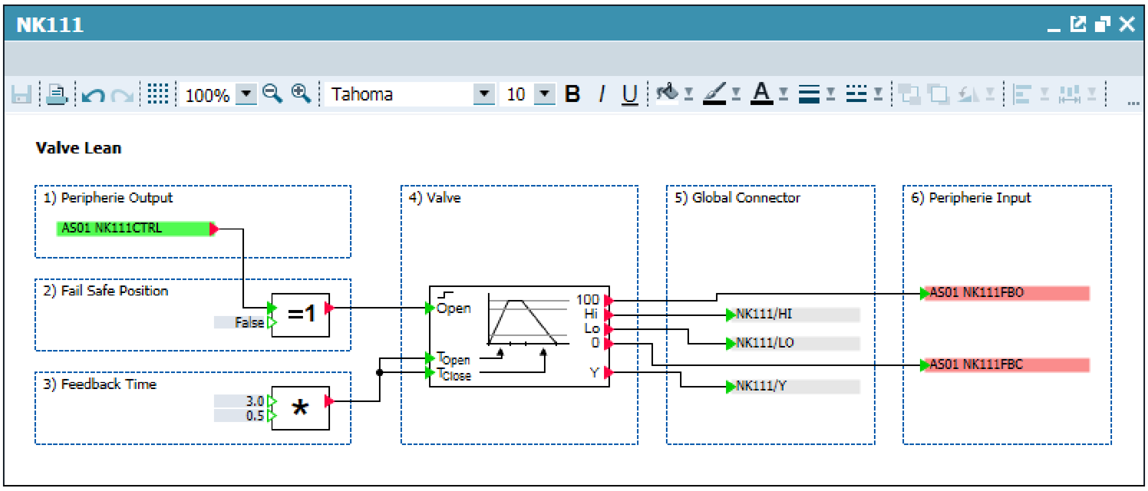
Table 6-2

Component	Parameter	Value	Description
MUL	General > Name	NK111_MUL	IN1 = Monitoring time of motor feedback in PCS 7 Response time = Half the monitoring time
	Input > X1	3.0	
	Input > X2	0.5	
XOR	General > Name	NK111_XOR	If IN2="true", at output OUT, the system inverts the signal at input IN1.
	Input > IN2	False	
DriveV1	General > Name	NK111_Drive	Simulates the valve setting (Y) and the feedback signal.
Connector	General > Name	NK111/HI	Global connector
Connector	General > Name	NK111/Lo	Global connector
Connector	General > Name	NK111/Y	Global connector

6.2.4 Interconnecting components

Interconnect the components as shown in the illustration below. It is only possible to connect outputs (▷) with inputs (▷).

Figure 6-6: Finished simulation model for valve "NK111"



Using component "XOR" at the input of the valve, you can invert a High and Low signal. This is relevant for the safe position of the valve. The MUL component at the "TOpen" and "TClose" inputs should provide dynamically adapted behavior. Here, the feedback monitoring time (3s) of the block in the AS program is set to half (factor 0.5).

The global connectors are for interchanging values between the individual charts. You can use them either as an output or as an input.

6.3 Creating further templates using CMT Import

Since creating charts for actuators and sensors manually is very complex, it is advisable to use the automatic model creation features offered by SIMIT. In this application example, automatic model creation is described using the "CMT Import" function.

Note

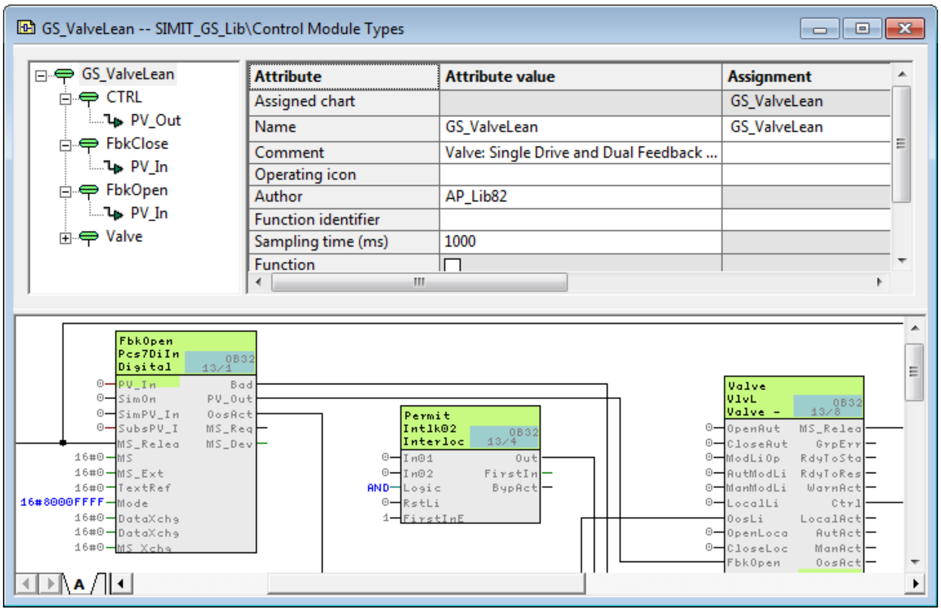
For more information on the various functions provided by SIMIT for automatic creation of models, refer to chapter 5 "Automatic model creation" of the "SIMATIC SIMIT (V9.0 SP1)" Operating Manual.

https://support.industry.siemens.com/cs/ww/en/view/109744640/73693220363

For you to be able to use the "CMT Import" function in SIMIT, it is assumed that the points below apply:

- The AS program has been configured using control module type (CMTs).

Figure 6-7: Control module type "GS_ValveLean"



- The AS program has been exported into an XML file (Export XML).
- The templates in SIMIT have been configured to match the CMTs in PCS 7.

NOTICE

Export XML

The "Export XML" function in SIMIATIC Manager is a SIMIT-specific function and is only available if you install it via a framework installation or from the SIMIT installation folder.

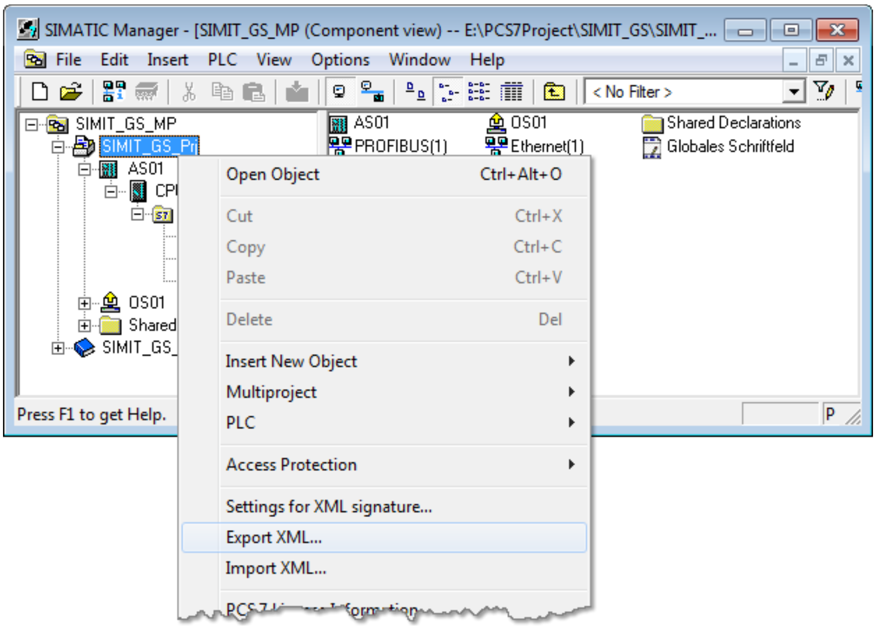
You can find the setup routine of "Export XML" in folder "XMLTRANSFER_..." in the SIMIT installation folder.

6.3.1 Exporting an XML file from PCS 7

Follow these steps to create the XML file:

1. Open the "SIMIT_GS_MP" PCS 7 multi-project.
2. In the context menu of the AS project, choose the "Export XML…" item.
3. Choose a suitable storage location and click on the "Save" pushbutton to complete exportation.

Figure 6-8: Shortcut menu item "Export XML…" in the SIMATIC Manager:



6.3.2 Creating the "GS_ValveLean" template from a chart

You create the templates on the "Templates" task card. There, you can create "User Templates" and "Project Templates". The project templates are stored automatically in folder "ppatt" of the project path. They are an integral part of the project and are archived with the project at archiving. "User Templates" are not stored in the project path and are not archived either. They are available for all of the projects being edited on the PC.

Since there are other valves in the PCS 7 project apart from "NK111", you can use the "NK111" chart that you already created in chapter "6.2 Creating your first simulation model" as the first template. In this case, placeholder tags replace the parameters of the objects in the chart. The placeholders and partial placeholders below are available:

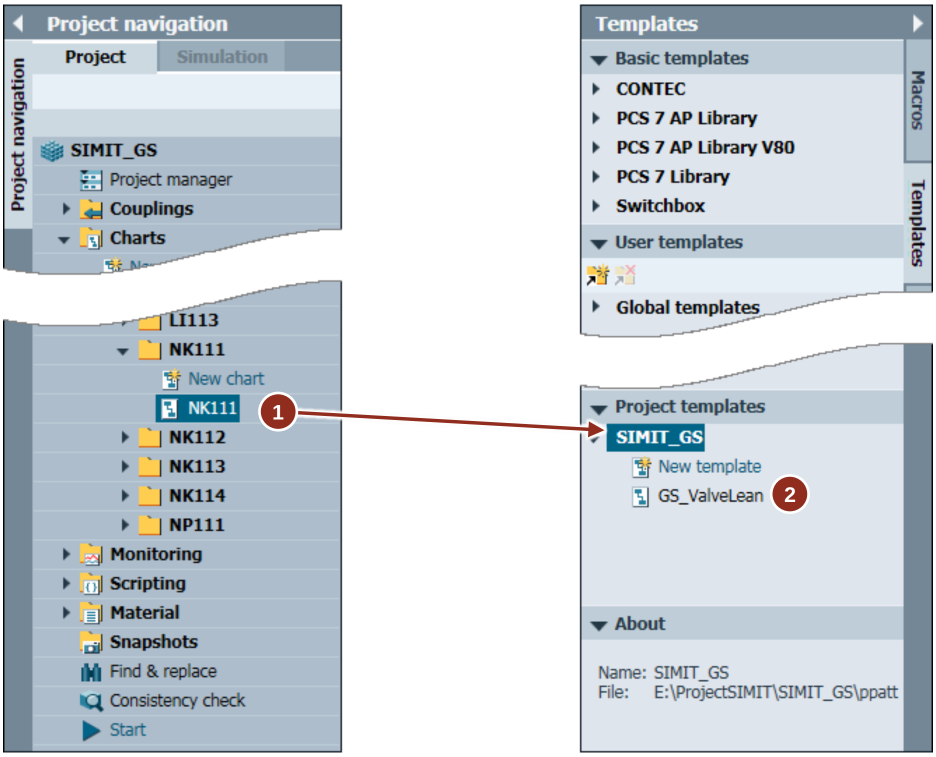
- ChartName
- COUPLING
- {$ChartName}...
- {$COUPLING}…

The {$} symbol indicates that the placeholder function is activated on the parameter. The placeholder function is not active if this symbol {$} is shown.

To create the template, proceed as follows:

1. In the tool window, choose the "Templates" task card.
2. In the project window, highlight chart "NK111" (1) and drag it to the project templates.

Figure 6-9: Creating templates from charts



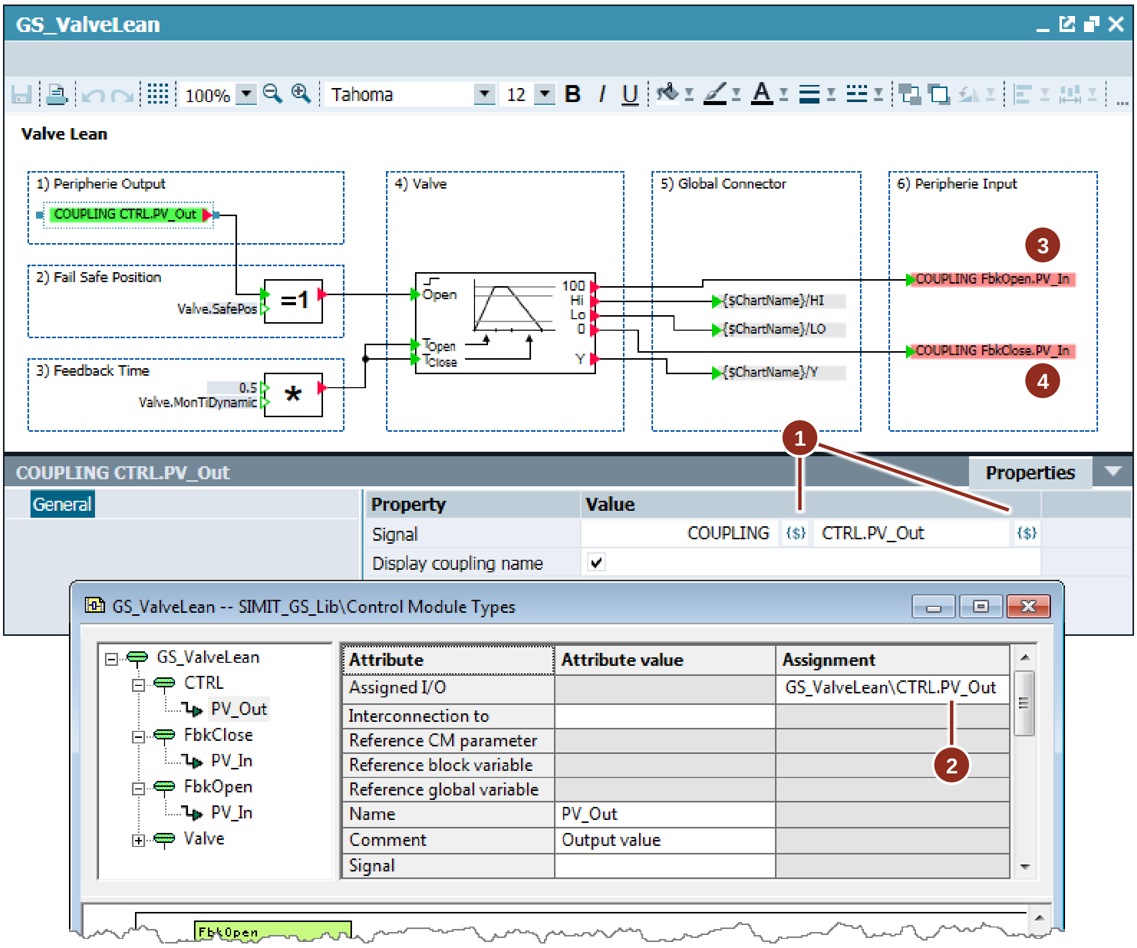
3. Rename the new template "GS_ValveLean" (2). To do this, choose shortcut menu item "Rename".

Note The name of the template must be identical to the name of the CMT in the PCS 7 project.

4. Double-click on the template to open it:
5. Select the output connector in the template.
6. In the "General > Signal" property, activate the placeholder function ({$} 1) for both input fields.
7. Enter the "COUPLING" placeholder in the first field. When the simulation model is generating of later, this placeholder will be changed to the connection name.

8. Enter the tag name "CTRL.PV_Out" in the second field. The name must match the tag name from the CMT (2).
9. Repeat the procedure for both input signals. Enter here the "COUPLING / FbkOpen.PV_In" (3) and "COUPLING / FbkClose.PV_In" (4) placeholders.

Figure 6-10: Changing the parameters in the template



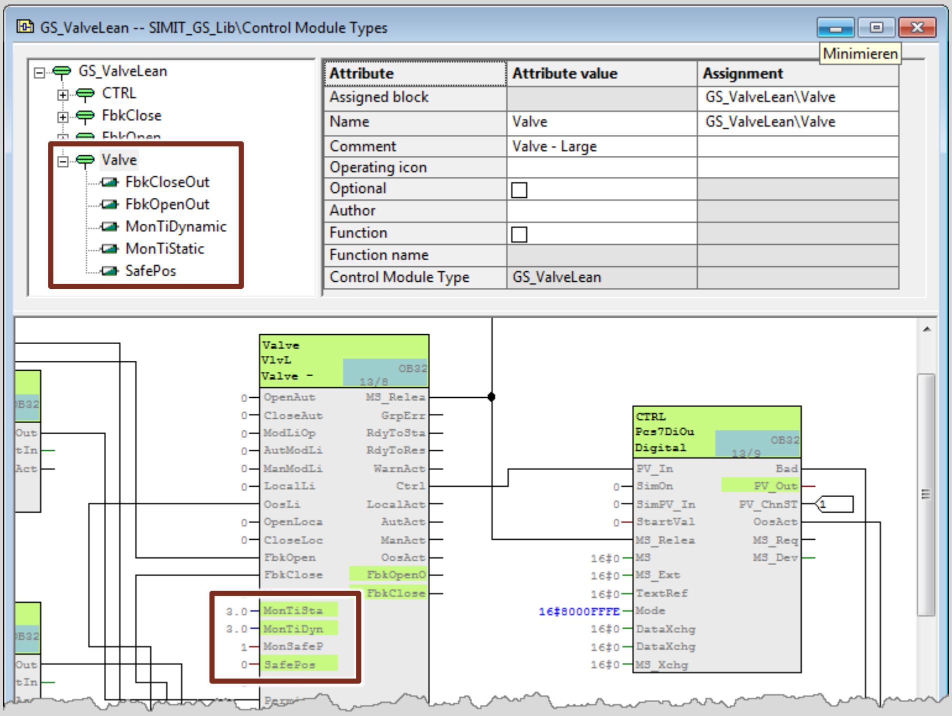
10. Adapt the parameters of the other objects based on the table below:

Table 6-3

Component	Parameter	Value
MUL	General > Name	{$ChartName}_MUL
	Input > X1	0.5
	Input > X2	Valve.MonTiDynamic {$}
XOR	General > Name	{$ChartName}_XOR
	Input > IN2	Valve.SafePos {$}
DriveV1	General > Name	{$ChartName}_Drive
Connector	General > Name	{$ChartName}/HI
Connector	General > Name	{$ChartName}/Lo
Connector	General > Name	{$ChartName}/Y

Note

Values "Valve.MonTiDynamic" and "Valve.SafePos" are parameters that are set in the control module on the "Valve" block. Function "CMT Import" reads out the value from the export file and assigns the value to the corresponding inputs in the simulation chart. You must also configure these parameters on the CMT.



6.3.3 Creating additional templates

The control module types below are used in the PCS 7 project:

- GS_ValveLean
- GS_MotorLean
- GS_AnaMon
- GS_ValveAna
- GS_Dose

You are already familiar with the general procedure for creating charts and templates from the "GS_ValveLean" template. You can now create the remaining templates on the basis of the following tables.

NOTICE	**Scaling** In these templates, scaling is carried out using the standard objects in the chart. If you have already carried out scaling in the coupling, remove it.

GS_MotorLean

Simulates the feedback of a simple drive.

Create a new template called "GS_MotorLean" and insert the components and controls below:

Table 6-4: Components of "GS_MotorLean"

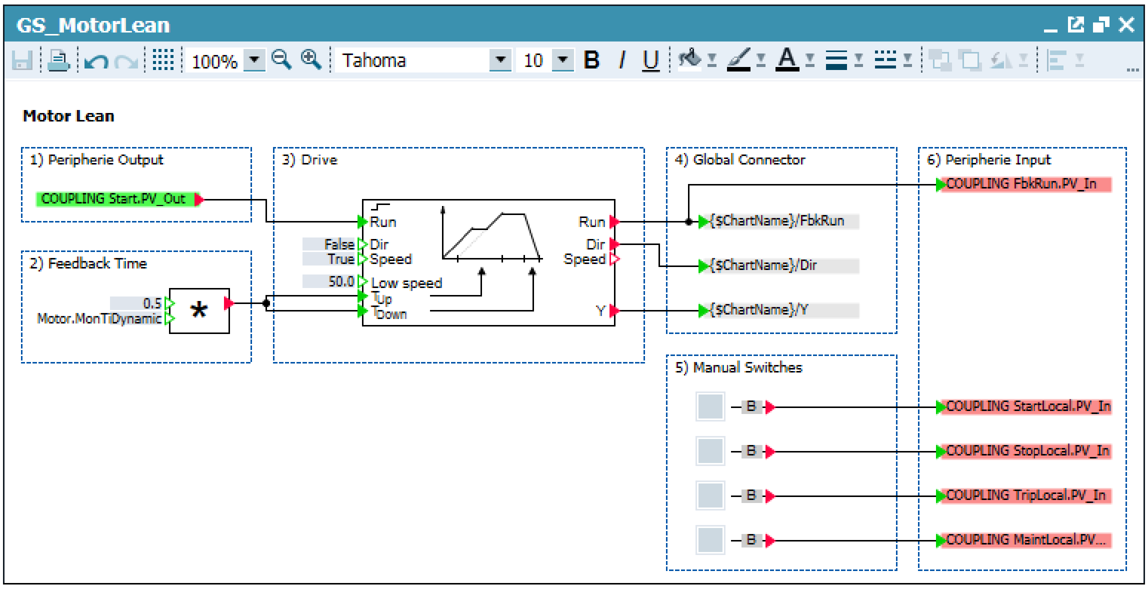
Component	Parameter	Value
Connectors > Output	General > Signal	COUPLING {$} / Start.PV_Out {$}
Standard > Analog Basic > MUL	General > Name	{$ChartName}_MUL
	Input > X1	0.5
	Input > X2	Motor.MonTiDynamic {$}
Drives > DriveP1	General > Name	{$ChartName}_DriveP1
Connectors > Connector	General > Name	{$ChartName}/FbkRun
Connectors > Connector	General > Name	{$ChartName}/Dir
Connectors > Connector	General > Name	{$ChartName}/Y
Standard > Misc > BConnector	General > Name	{$ChartName}_StartLocal_BCon
Standard > Misc > BConnector	General > Name	{$ChartName}_StopLocal_BCon
Standard > Misc > BConnector	General > Name	{$ChartName}_TripLocal_BCon
Standard > Misc > BConnector	General > Name	{$ChartName}_MaintLocal_BCon
Connectors > Input	General > Signal	COUPLING {$} / FbkRun.PV_In {$}
Connectors > Input	General > Signal	COUPLING {$} / StartLocal.PV_In {$}
Connectors > Input	General > Signal	COUPLING {$} / StopLocal.PV_In {$}
Connectors > Input	General > Signal	COUPLING {$} / TripLocal.PV_In {$}
Connectors > Input	General > Signal	COUPLING {$} / MaintLocal.PV_In {$}

Table 6-5: Controls of "GS_MotorLean"

Control	Parameter	Value
Input > Pushbutton	General > Name	{$ChartName}_StartLocal_Push
Input > Pushbutton	General > Name	{$ChartName}_StopLocal_Push
Input > Pushbutton	General > Name	{$ChartName}_TripLocal_Push
Input > Pushbutton	General > Name	{$ChartName}_MaintLocal_Push

Interconnect the components as shown in the illustration below:

Figure 6-11: "GS_MotorLean" template



Note The input signals for manual switches are optional connections of control module types. The control modules that do not use this option are not created in the SIMIT charts of these input connectors either.

GS_AnaMon

Simulates the feedback of an analog value process tag.

Create a new template called "GS_AnaMon" and insert the components, controls, and macros below:

Table 6-6: Components of "GS_AnaMon

Component	Parameter	Value
Connectors > Connector	General > Name	{$ChartName}/PV
Standard > AnalogExtended > Selection	General > Name	{$ChartName}_Selection
Standard > AnalogBasic > ADD	General > Name	{$ChartName}_ADD
	Input > X2	250.0
Standard > Conv > Phys2Raw	General > Name	{$ChartName}_Phys2Raw
	Parameter > Phys_Lower_Limit	Input.Scale\Low {$}
	Parameter > Phys_Upper_Limit	Input.Scale\High {$}
Standard > Conv > Signed2Unsigned	General > Name	{$ChartName}_Signed2Unsigned
	Parameter > Width	2 bytes
Connectors > Input	General > Signal	COUPLING {$} / Input.PV_In {$}

Table 6-7: Controls of "GS_AnaMon"

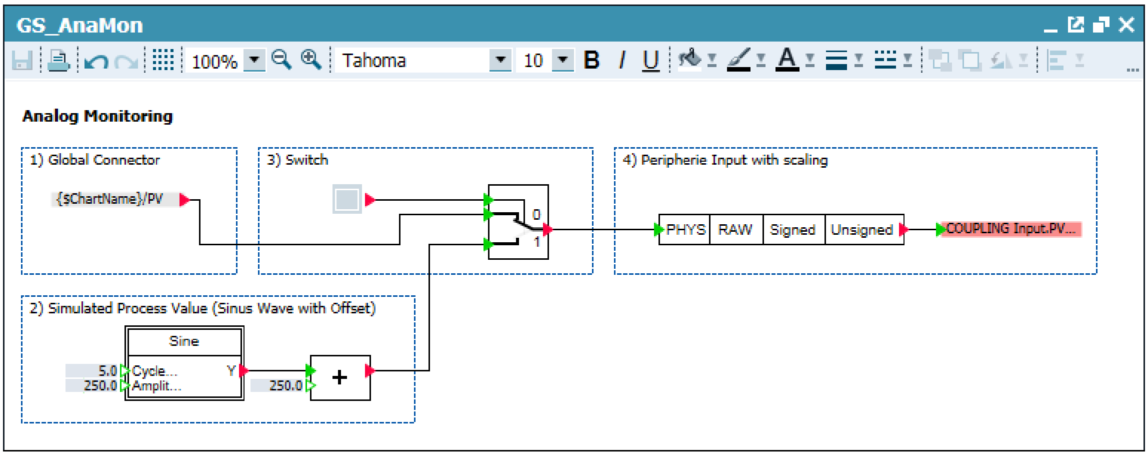
Control	Parameter	Value
Input > Switch	General > Name	{$ChartName}_Switch

Table 6-8: Macros of "GS_AnaMon"

Macro	Parameter	Value
Basic macros > Sine	General > Name	{$ChartName}_Sine
	Input > CycleDuration	5.0
	Input > Amplitude	250.0

Interconnect the components as shown in the illustration below:

Figure 6-12



Note Using the switch, you can switch over between the simulated process value and a sine curve. You can use the sine curve to test limit alarms, for example. Simulation of the process value is carried out at the process level and in described further on in the Getting started document.

GS_ValveAna

Simulates the feedback of an analog valve.

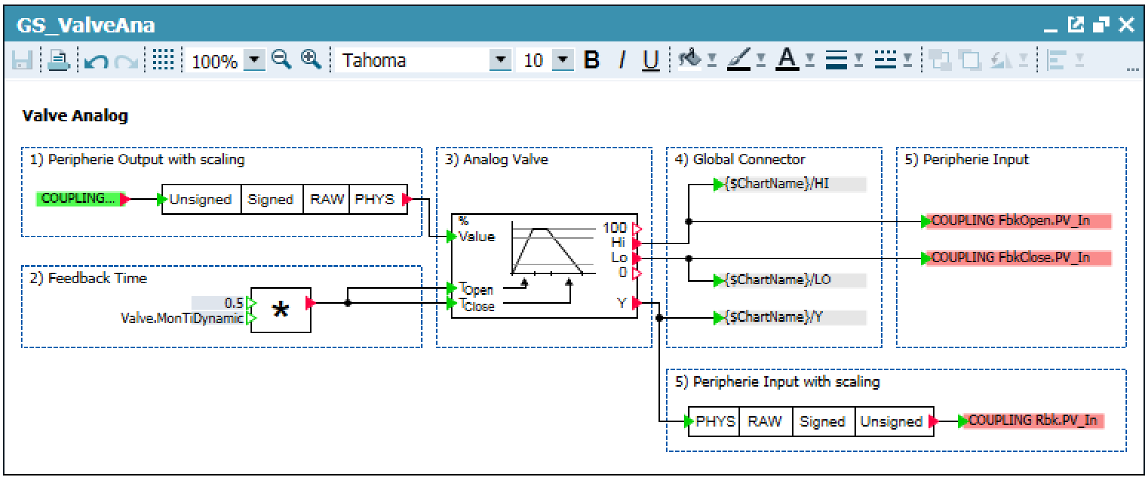
Create a new template called "GS_ValveAna" and insert the components below:

Table 6-9: Components of "GS_AnaMon

Component	Parameter	Value
Connectors > Output	General > Signal	COUPLING {$} / CTRL.PV_Out {$}
Standard > Conv > Unsigned2Signed	General > Name	{$ChartName}_Unsigned2Signed
	Parameter > Width	2 bytes
Standard > Conv > Raw2Phys	General > Name	{$ChartName}_Raw2Phys
	Parameter > Phys_Lower_Limit	CTRL.Scale\Low {$}
	Parameter > Phys_Upper_Limit	CTRL.Scale\High {$}
Standard > AnalogBasic > MUL	General > Name	{$ChartName}_MUL
	Input > X1	0.5
	Input > X2	Valve.MonTiDynamic {$}
Drives > DriveV4	General > Name	{$ChartName}_DriveV4
Connectors > Connector	General > Name	{$ChartName}/HI
Connectors > Connector	General > Name	{$ChartName}/LO
Connectors > Connector	General > Name	{$ChartName}/Y
Standard > Conv > Phys2Raw	General > Name	{$ChartName}_Phys2Raw
	Parameter > Phys_Lower_Limit	Rbk.Scale\Low
	Parameter > Phys_Upper_Limit	Rbk.Scale\High
Standard > Conv > Signed2Unsigned	General > Name	{$ChartName}_Signed2Unsigned
	Parameter > Width	2 bytes
Connectors > Input	General > Signal	COUPLING {$} / FbkOpen.PV_In {$}
Connectors > Input	General > Signal	COUPLING {$} / FbkClose.PV_In {$}
Connectors > Input	General > Signal	COUPLING {$} / Rbk.PV_In {$}

Interconnect the components as shown in the illustration below:

Figure 6-13



GS_Dose

Simulates the feedback of an analog value process tag.

Create a new template called "GS_MotorLean" and insert the components and controls below:

Table 6-10: Components of "GS_AnaMon

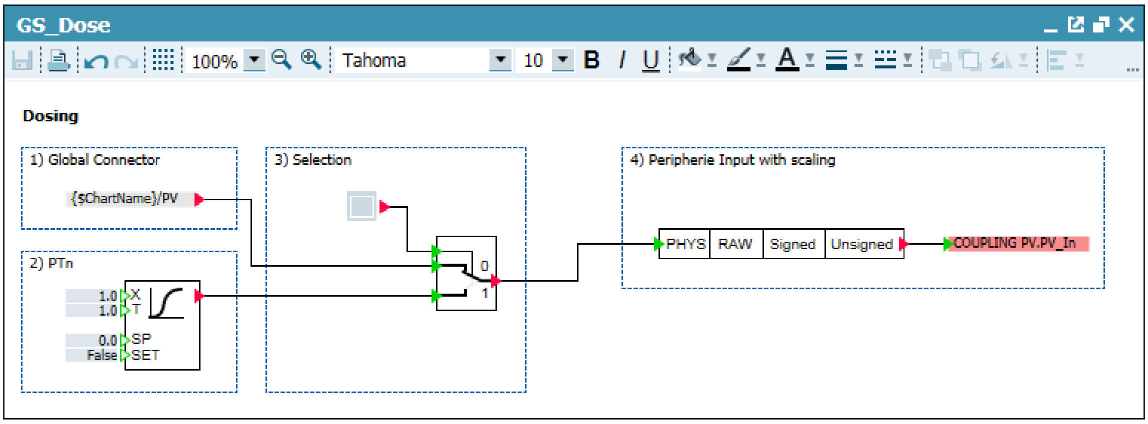
Component	Parameter	Value
Connectors > Connector	General > Name	{$ChartName}/PV
Standard > AnalogExtended > PTn	General > Name	{$ChartName}_PTn
Standard > BinaryBasic > Selection_B	General > Name	{$ChartName}_Selection
Standard > Conv > Phys2Raw	General > Name	{$ChartName}_Phys2Raw
	Parameter > Phys_Lower_Limit	PV.Scale\Low {$}
	Parameter > Phys_Upper_Limit	PV.Scale\High {$}
Standard > Conv > Signed2Unsigned	General > Name	{$ChartName}_Signed2Unsigned
	Parameter > Width	2 bytes
Connectors > Input	General > Signal	COUPLING {$} / PV.PV_In {$}

Table 6-11: Controls of "GS_AnaMon"

Control	Parameter	Value
Input > Switch	General > Name	{$ChartName}_Switch

Interconnect the components as shown in the illustration below:

Figure 6-14



Note

Using the switch, you can switch over between the simulated process value and a substitute value. Simulation of the process value is carried out at the process level and in described further on in the Getting started document.

6.4 CMT Import

The templates that we created in chapter 6.3 are the basis of CMT Import. AT CMT import, the system creates in SIMIT a chart from the corresponding template for each instance of a CMT.

For example: the PCS 7 project contains three control modules of type "GS_AnaMon" (LI111, LI112, LI113). AT CMT import, the system now generates three simulation charts (LI111, LI112, LI113) from the "GS_AnaMon" template.

The placeholder tags at the input signal are replaced by the coupling that is used and the symbol that is interconnected at input "Input.PV_In". For example:

- COUPLING / Input.PV_In → AS01 / AI_LI111
- COUPLING / Input.PV_In → AS01 / AI_LI112
- COUPLING / Input.PV_In → AS01 / AI_LI113

In the case of the partial placeholders that are used, the system replaces the corresponding characters. For example:

- {$ChartName}/PV → LI111/PV
- {$ChartName}/PV → LI112/PV
- {$ChartName}/PV → LI113/PV

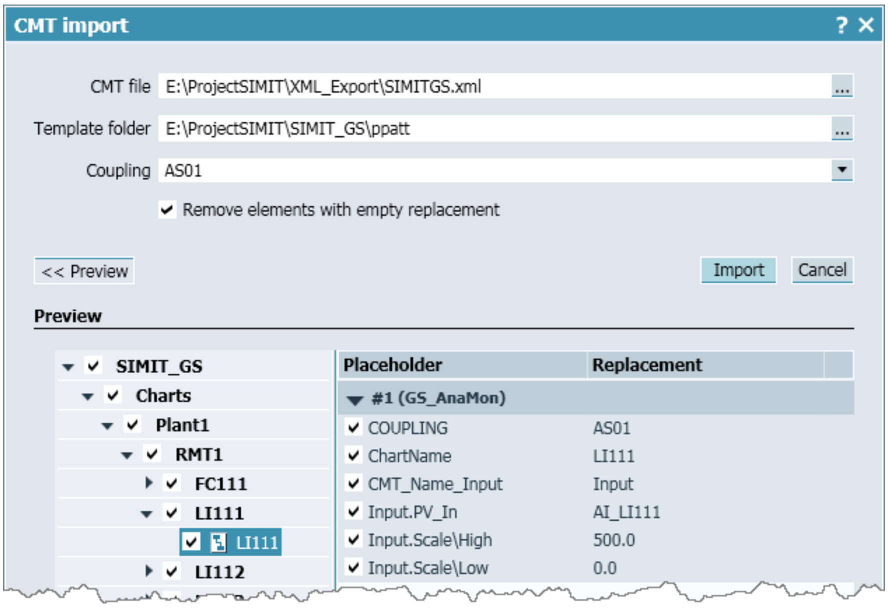
Carrying out CMT import

Proceed as follows:

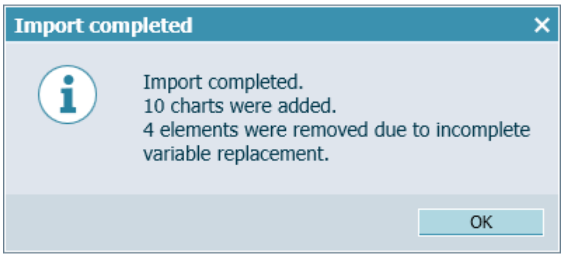
1. Start CMT import using menu item "Automatic modelling > CMT import
2. In the "CMT Import" dialog, choose the following settings:
   - In field CMT file, choose the XML file that you created in chapter "6.3.1 Exporting an XML file from PCS 7".
   - In the "Template folder" field, select the path to the "ppatt" template folder of the project.
   - Select the coupling that you want to use.

Note The "Preview" pushbutton adds an area to the dialog in which you can have the system display all of the data that was created by importing.

Figure 6-15: Parameters at CMT import

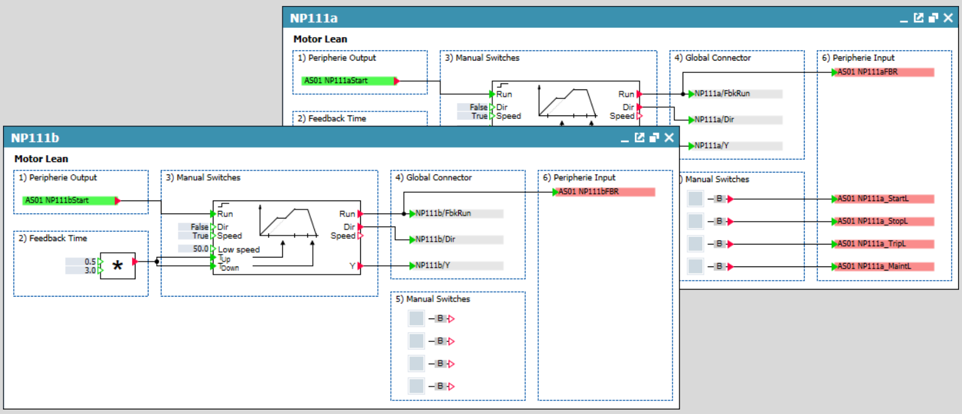


3. Click on the "Import" pushbutton. If charts of the same name already exist, you can overwrite the existing ones or cancel importation.
4. After importing successfully, the system displays a dialog containing a summary.



Note

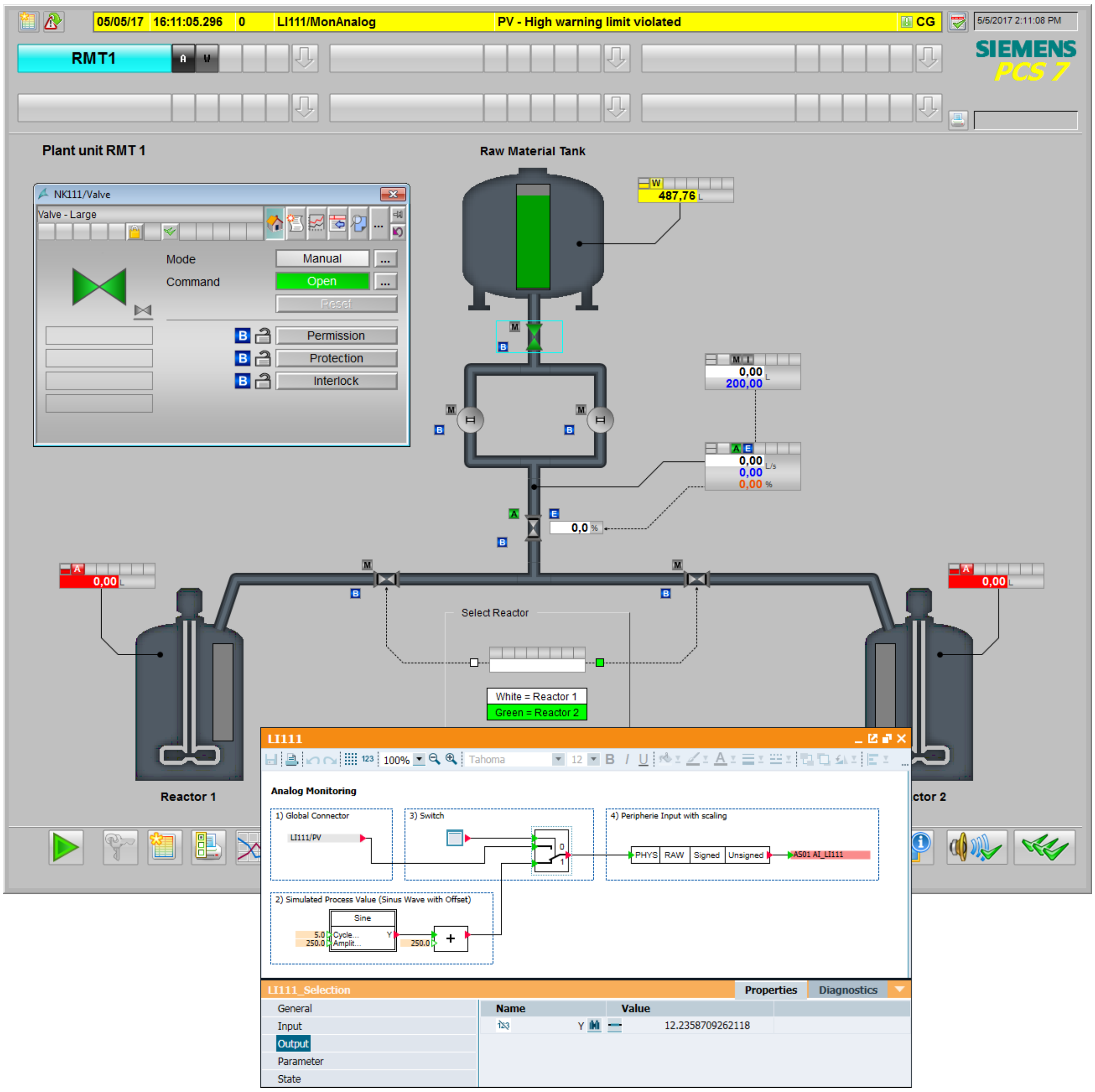
In the PCS 7 project, CMT "GS_MotorLean" was created with optional connections. Control module "NP111b" does without these connections.
For this reason, the system indicates that it was not possible to create four tags.
The four inputs were removed from the plan.



6.5 Current version

Im chapter 6, you created the simulation models of the actuators and sensors. At this time, you can already monitor the behavior of the actuators and sensors in OS Runtime. You can now open and close valves or start drives. Apart from this, you can supply a substitute value, e.g. a sine value for fill levels, to the process tags by operating the corresponding switch on the chart. However, realistic operation of the plant is not yet possible.

Figure 6-16



7 Simulating the process level

At the process level, you simulate the physical contexts of the production facility. Modeling of processes and the associated physical relationships will not be covered in full with pre-made components. This process can be somewhat more complex.

Requirements

The following conditions apply to the creation of physical relationships:

- The physical relationship that is to be simulated (or a simplified form of it) must be known
- The device level must already have been created in SIMIT

Preliminary considerations

First, you must consider how to replicate the process to be simulated. When doing this, the effort involved in the development of the process model should not exceed the benefits. A simple process model is usually sufficiently accurate to test the automation program.

In the present application example, simulation is needed for the following processes:

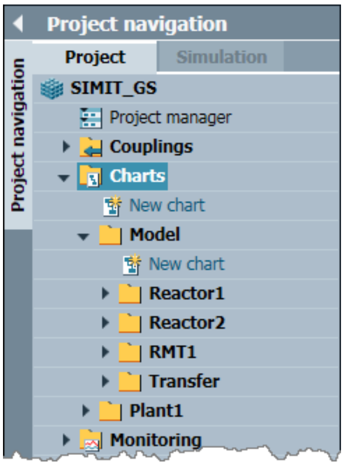
- Fill level of raw material tank (0 - 500 L)
- Fill level of reactor 1 and reactor 2 (0 - 1000 L)
- Flow rate of RMT to reactor 1 or 2 (0 - 3 L/s)

7.1 Creating the folder structure

To simulate the process, add the "Model" area to the folder structure.
Follow the steps below to create folders in SIMIT:

1. In the shortcut menu of the "Charts" folder, choose the "New folder" item and rename the new folder "Model".
2. Within folder "Model", create the following sub-folders:
   RMT1, Reactor1, Reactor2 and Transfer.

Figure 7-1: Folder structure



7.2 Modeling material transportation

The amount of material that is transferred from the raw material tank to the reactors is collected at measuring point "FC111".

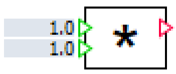
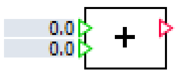
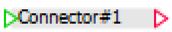
The following functions are simulated on the chart:

- The pumping capacity is collected (0 - 100%)
- The valve settings are collected (0 - 100%)
- The volumetric flow is calculated (0 - 3 L/s)

Follow the steps below to configure simulation of the transfer rate:

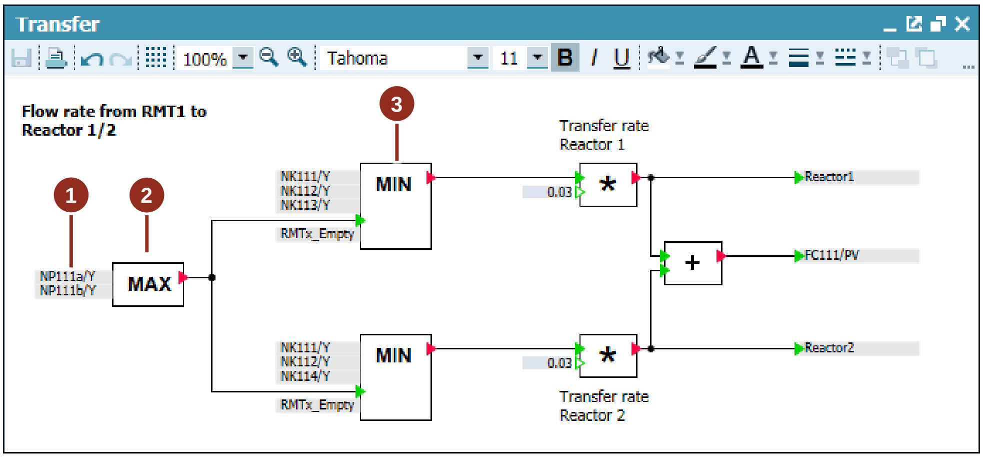
1. In the "Transfer" folder create a new chart called "Transfer".
2. Paste all of the components into the chart that are described in the table below.

Table 7-1

Symbol	Qty.	Name	Library	Description
	3	MinMax	Standard > AnalogExtended	Outputs the minimum or maximum value. The number of inputs is variable.
	2	MUL	Standard > AnalogBasic	Multiplies 1 to n analog values
		ADD	Standard > AnalogBasic	Adds 1 to n analog values
	13	Connector	Connectors	Connectors make connections within the project

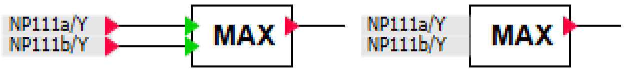
3. Place, connect, and parameterize the components as shown in the illustration below. Also change the names of all of the objects such that you can assign them to the associated chart, e.g. "Transfer_MUL1" for a multiplier.

Figure 7-2: "Transfer" simulation chart

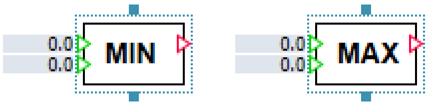


Configuration notes

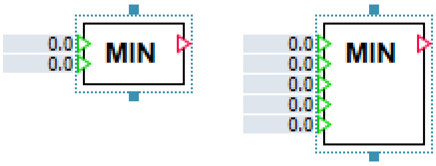
(1) You can link connectors directly with the connections of other components by dragging and dropping them to a connection. This means that the connection in the form of a line is omitted and the "▶▶" component connections are not displayed. Both variants have equal priority.



(2) Some components can carry out different tasks. For example: the "MinMax" component can check one or more values for a minimum value or a maximum value. You can choose the type in the properties under "Parameters > MinMax".



(3) On some components, you can configure a different number of inputs. You can simply drag the sizing handles to increase the size of the component and in this way display further connections.



7.3 Modeling the raw material tank

We only need a fill level for the raw material tank (RMT). The system acquires the fill level at measuring point "LI111".

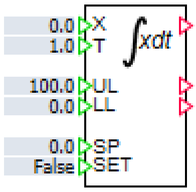
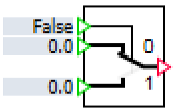
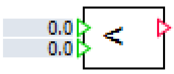
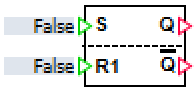
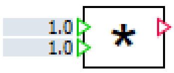
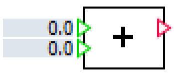
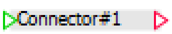
The following functions are simulated on the chart:

- The system uses an integrator to acquire the fill level. A positive value on the integrator allows the fill level to rise and a negative one allows it to drop.
- The maximum volume is 500 L.
- The volumetric flow is applied from the "Transfer" chart.
- If a lower value is exceeded, the fill level is incremented by 15 l/s until the fill level exceeds an upper value.

Follow the steps below to configure simulation of the fill level:

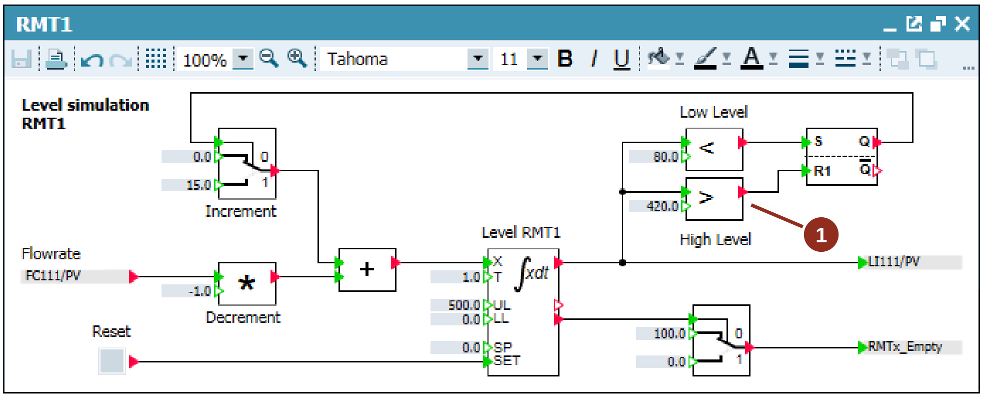
1. In the "RMT1" folder create a new chart called "RMT1".
2. Paste all of the components into the chart that are described in the table below.

Table 7-2

Symbol	Qty.	Name	Library	Description
		PTn	Standard > AnalogExtended	Integrator to calculate the fill level
	2	Selection	Standard > AnalogExtended	Chooses between two analog values
	2	Compare	Standard > AnalogExtended	Comparator can be configured as lower than or higher than
		RS_FF	Standard > BinaryExtended	RS-Flip Flop – resetting has priority
		MUL	Standard > AnalogBasic	Multiplies 1 to n analog values
		ADD	Standard > AnalogBasic	Adds 1 to n analog values
	3	Connector	Connectors	Connectors make connections within the project

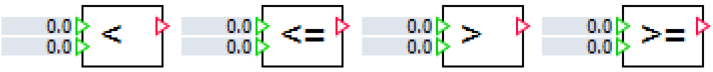
3. Place, connect, and parameterize the components as shown in the illustration below. Also change the names of all of the objects such that you can assign them to the associated chart, e.g. "RMT1_INT" for the integrator.

Figure 7-3: "RMT1" simulation chart



Configuration notes

Some components can carry out different tasks. For example: the "Compare" component (1) can carry out a comparison in four different ways. You can choose the type in the properties under "Parameters > Comparison".



7.4 Modeling of reactor 1

We only need a fill level for reactor 1. The system acquires the fill level at measuring point "LI112".

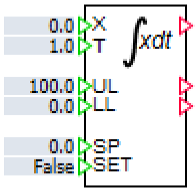
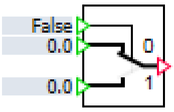
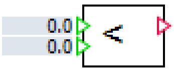
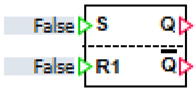
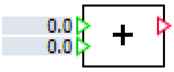
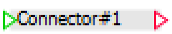
The following functions are simulated on the chart:

- The system uses an integrator to acquire the fill level. A positive value on the integrator allows the fill level to rise and a negative one allows it to drop.
- The maximum volume is 1000 L.
- The volumetric flow is applied from the "Transfer" chart.
- If an upper value is exceeded, the fill level is decremented by 15 l/s until the fill level falls short of a lower value.

Follow the steps below to configure simulation of the fill level:

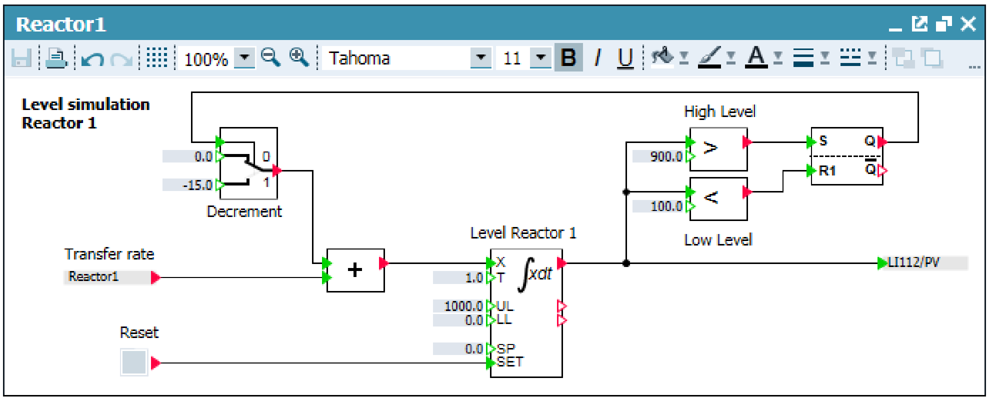
1. In the "Reactor1" folder create a new chart called "Reactor1".
2. Paste all of the components into the chart that are described in the table below.

Table 7-3

Symbol	Qty.	Name	Library	Description
		PTn	Standard > AnalogExtended	Integrator to calculate the fill level
		Selection	Standard > AnalogExtended	Chooses between two analog values
	2	Compare	Standard > AnalogExtended	You can configure comparators for different comparisons
		RS_FF	Standard > BinaryExtended	RS-Flip Flop – resetting has priority
		ADD	Standard > AnalogBasic	Adds 1 to n analog values
	2	Connector	Connectors	Connectors make connections within the project

3. Place, connect, and parameterize the components as shown in the illustration below. Also change the names of all of the objects such that you can assign them to the associated chart, e.g. "Reactor1_INT" for the integrator.

Figure 7-4

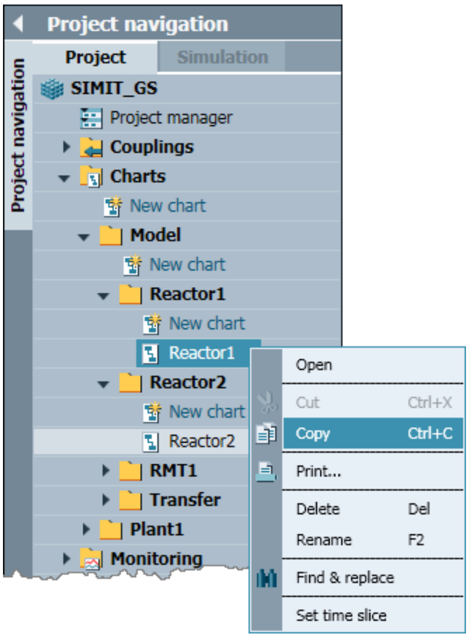


7.5 Modeling of reactor 2

The simulation model for reactor 2 is identical to the one for reactor 1. For this reason, you can simply copy the chart and then adapat the parameters appropriately. The system acquires the fill level at measuring point "LI113".

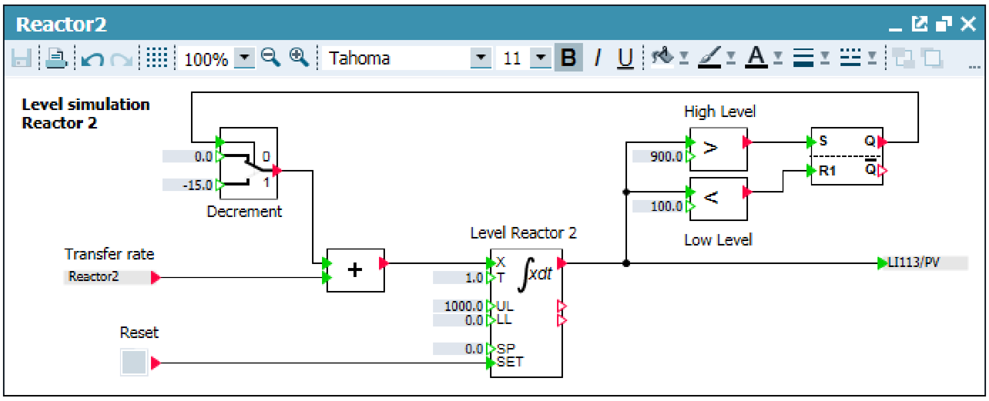
Proceed as follows:

1. Copy the "Reactor1" chart and paste it into the "Reactor2" folder.
2. Rename the chart to "Reactor2".



3. Change the parameters and the names of all of the objects to match the use of reactor 2.

Figure 7-5


Reactor2
100%
Tahoma
11
Level simulation
Reactor 2
0.0
-15.0
0
1
Decrement
Transfer rate
Reactor2
Reset
+
Level Reactor 2
X
T
UL
LL
SP
SET
1.0
1000.0
0.0
0.0
∫xdt
High Level
900.0
>
100.0
<
Low Level
S
R1
Q
LI113/PV

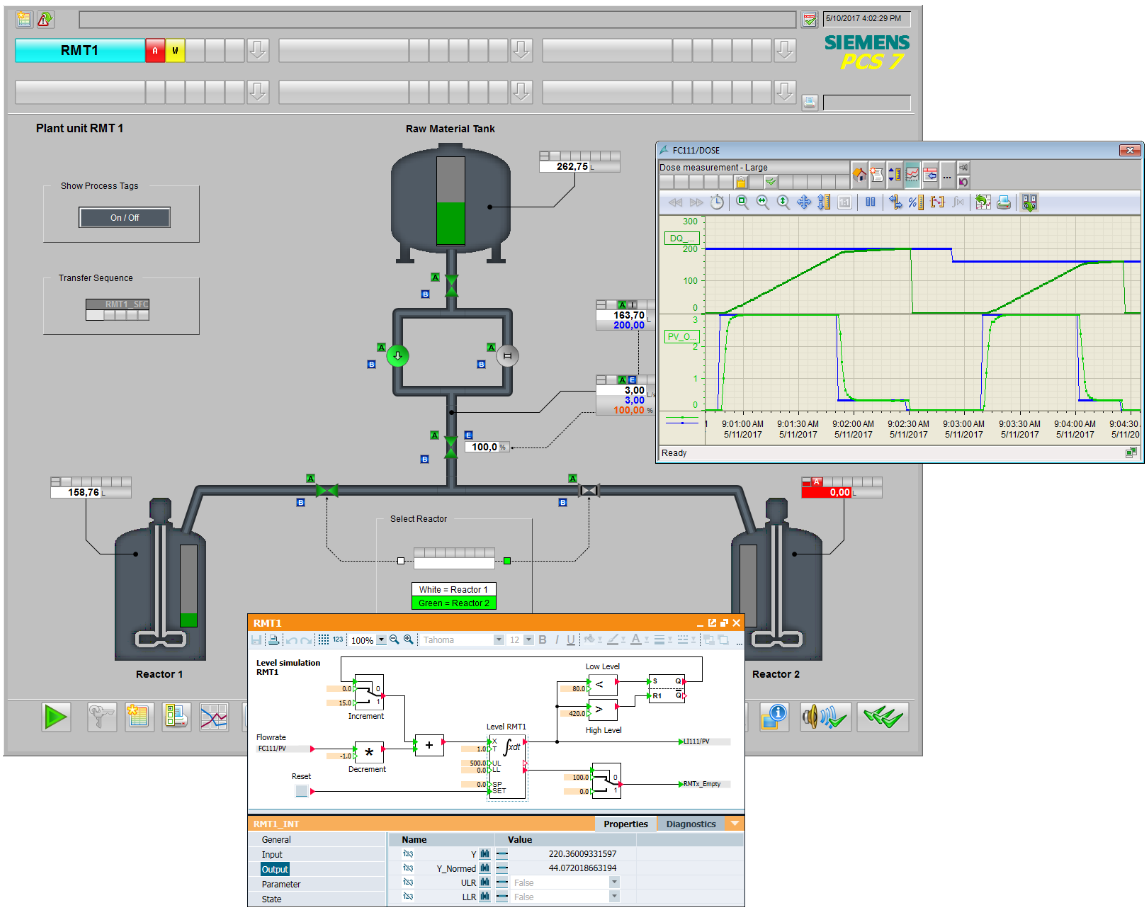
7.6 Current version

In chapter 7, you configured simulation of the physical behavior of the system. You are now in a position to test the AS program for correct functioning.

With OS Runtime started, you can, for example, execute the sequential function chart for material transfer or analyze the behavior of dosing with changed parameters.

In the simulation charts, it is also possible to view the individual process values in the properties of the components.

Figure 7-6



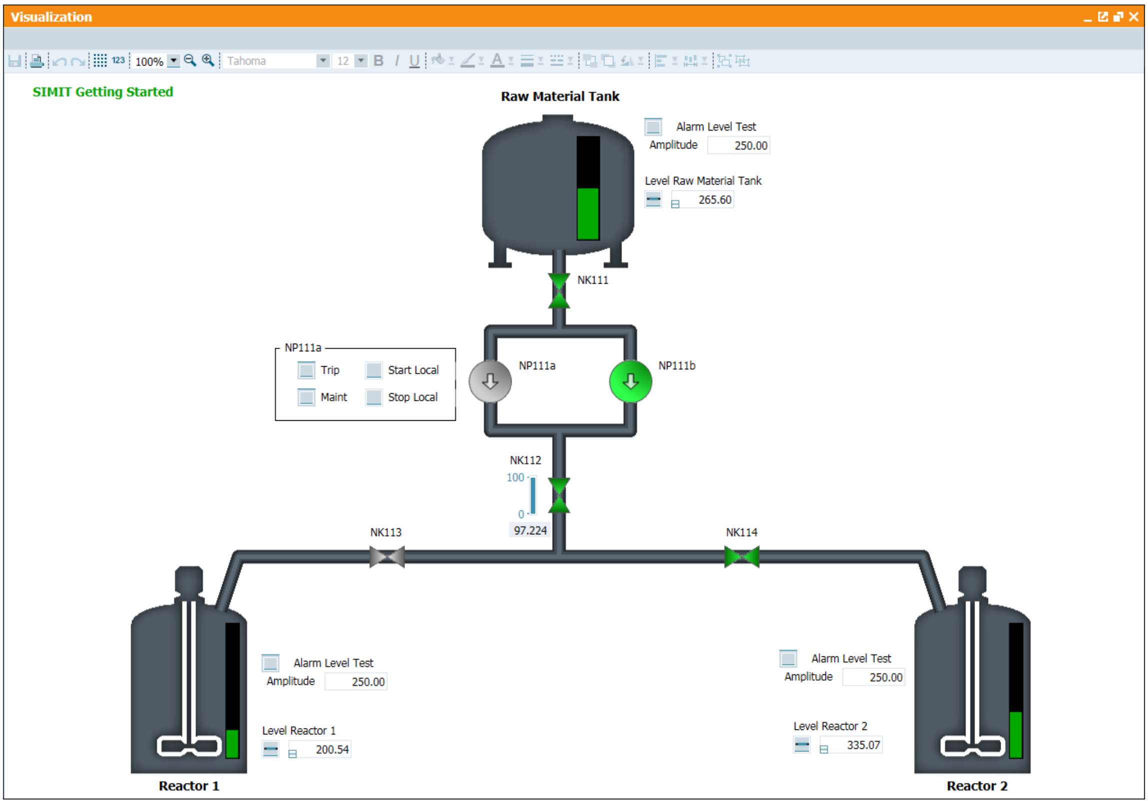
8 Visualization in SIMIT

SIMIT gives you the option of adding a graphical user interface to the simulation. In this way, you can carry out operation and monitoring of the system directly in the simulation regardless of whether your automation system has a visualization facility or not.

There are several components in the Tools menu that allow you to do this. On the "Controls" task card, you can find components like analog or digital displays, for example, that you can use to display process values. Apart from this, there are switches, pushbuttons, and sliders there that can be used to enter values and signals.

In this chapter, you will create in SIMIT the graphical user interface for your system as shown in the screenshot below:

Figure 8-1: Graphical user interface in SIMIT



The illustration includes static and dynamic components as well as components for the input and output of values and signals.

8.1 Creating graphics

In principle, it does not matter how you create the graphics for visualization. The best thing to do is to use the graphics tool that you are most familiar with. SIMIT can display the following image formats: "bmp", "jpg", "jpeg", "png", "gif", "tiff", and "ico". You can also use the integrated SIMIT graphics functions to draw the image; however, you will soon realize that the options are very limited and your charts will not be very clear.

Note All of the images that have been created in this application example are also in the download file that has been made available.

You need the graphics below for the sample system:

- Vertical and horizontal valves in the "Closed" (gray) and "Open" (green) operating states.
- Drives in the "Off" (gray) and "On" (green) operating states.
- A background image including the containers and pipes
  (Since the valves and drives are always visible, you can draw the gray variants directly into the background image)

If you can access configuration of the OS (OS-Engineering) from PCS 7, you can also use these graphics easily. We will address this procedure below.

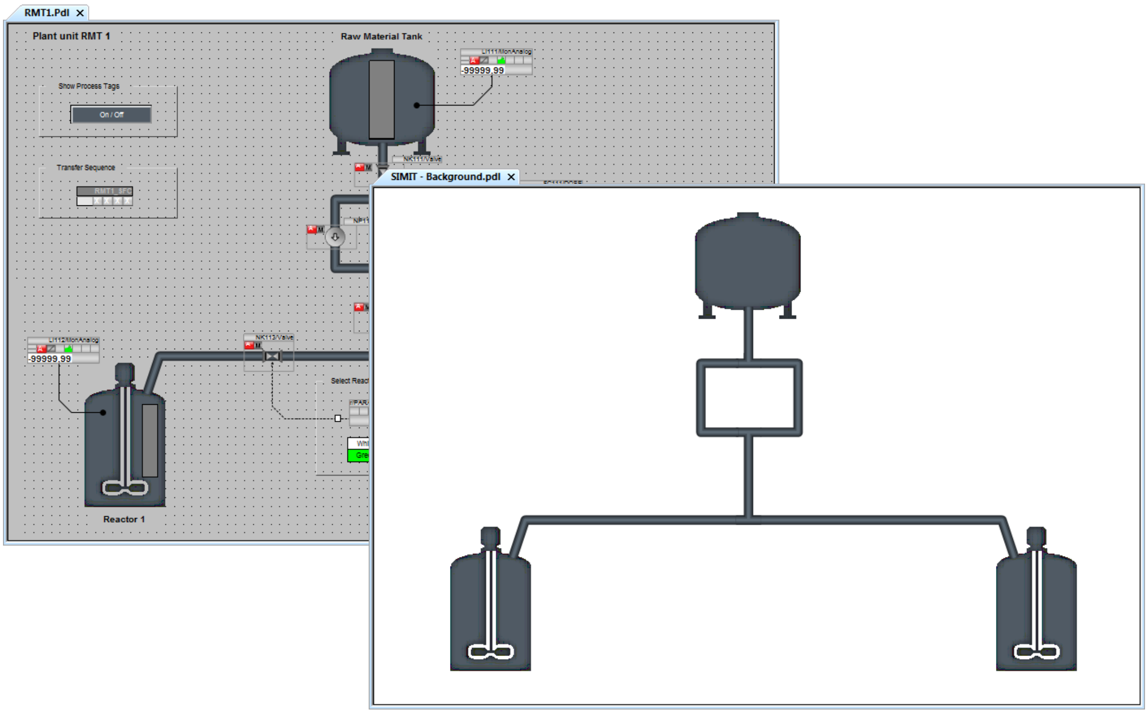
Note The images that are used in the example are in folder "SIMIT_GS_Pics" in the download archive.

8.1.1 Preparing the background image

Follow the steps below to prepare the background image:

1. Open the plant display using the WinCC Graphics Designer. Create a copy of it (Save as….)
2. Adapt the background color of the WinCC picture (e.g. to white) and deactivate the grid display.
3. Remove all of the components from the picture that you do not want to use later.

Figure 8-2: Original and adapted WinCC picture



8.1.2 Creating symbols

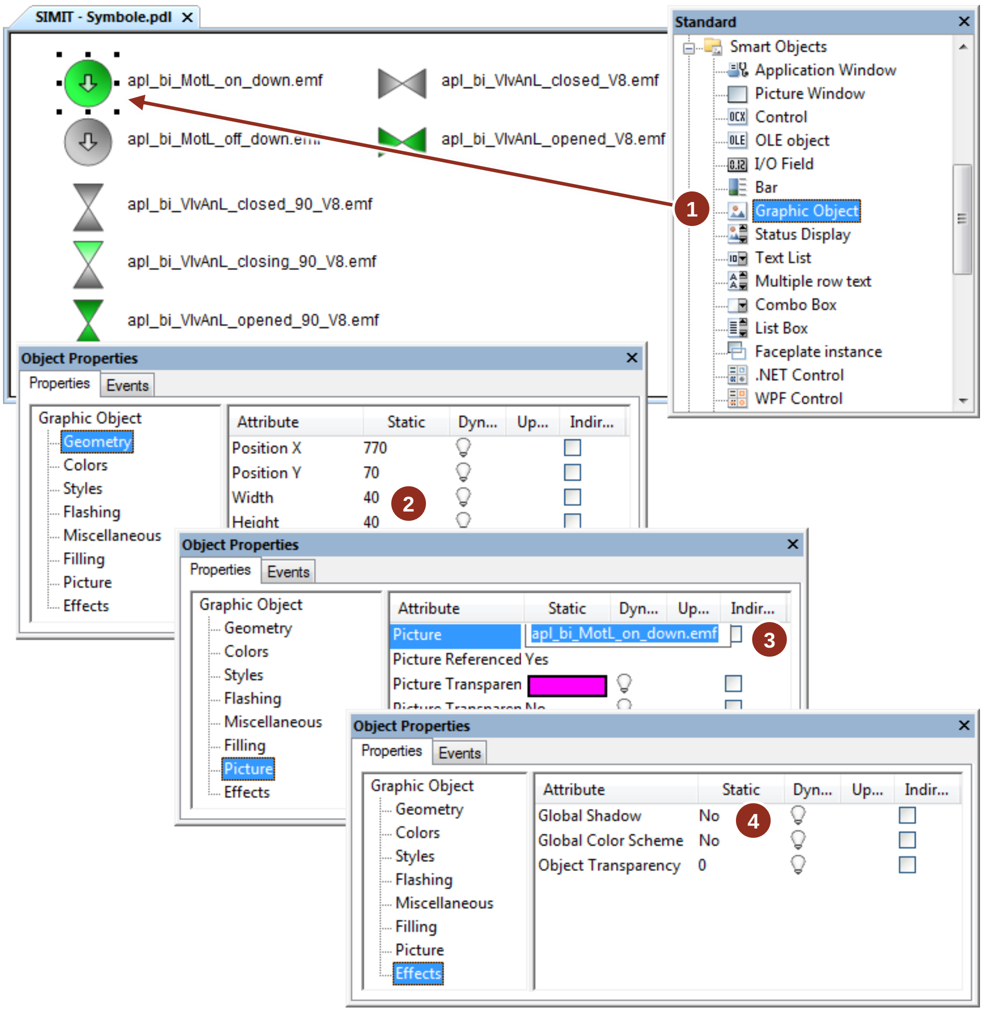
You can create the symbols of the drives and valves in the Graphics Designer too. There are a number of options available for this, e.g. you could use the "Dissolve user objects" function to break down APL block icons into all of their individual components. However, they contain a lot of objects and are unclear.

The easiest thing to do is to insert a graphics object into the WinCC picture and to assign the corresponding APL symbol as an image. All of the APL symbols in the OS project are in EMF format.

Proceed as follows:

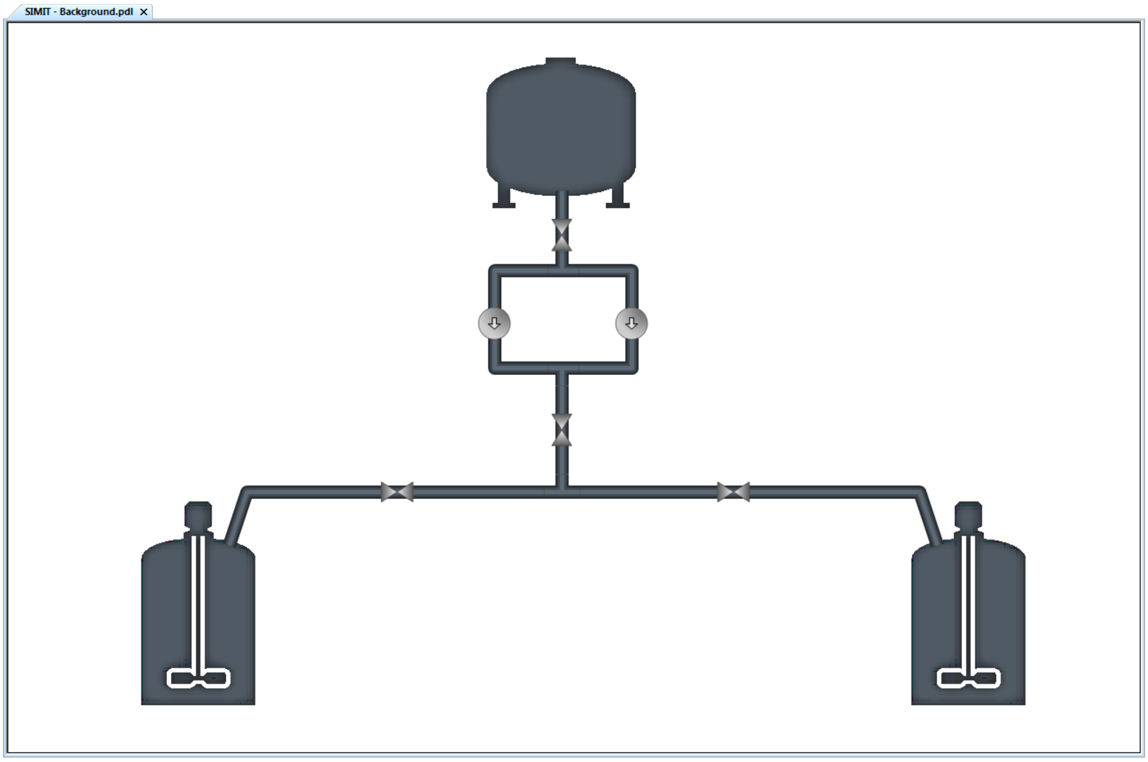
1. Create a new WinCC picture called "SIMIT Symbols.pdl".
2. Insert a "Graphic Object" (1) in the WinCC picture and choose a square shape (2). For example: "Geometry > Width / Height = 40".
3. Select the APL pictogram for a started drive "apl_bi_MotL_on_down.emf" (3).
4. In the properties choose the "Effects > Global Shadow / Global Color Scheme = No" setting.
5. Create six copies of the inserted picture and and choose the pictograms below for display:
   - apl_bi_MotL_off_down.emf
   - apl_bi_VlvAnL_closed_V8.emf
   - apl_bi_VlvAnL_opened_V8.emf
   - apl_bi_VlvAnL_closed_90_V8.emf
   - apl_bi_VlvAnL_closing_90_V8.emf
   - apl_bi_VlvAnL_opened_90_V8.emf

Figure 8-3



6. Copy the gray symbols into the WincC picture that shows the background of SIMIT.

Figure 8-4: SIMT background picture created in WinCC Graphics Designer



Note Since the gray symbols in SIMIT are always displayed, they can also be components of the background picture. This will reduce the number of objects in the SIMIT project later.

8.1.3 Creating graphics

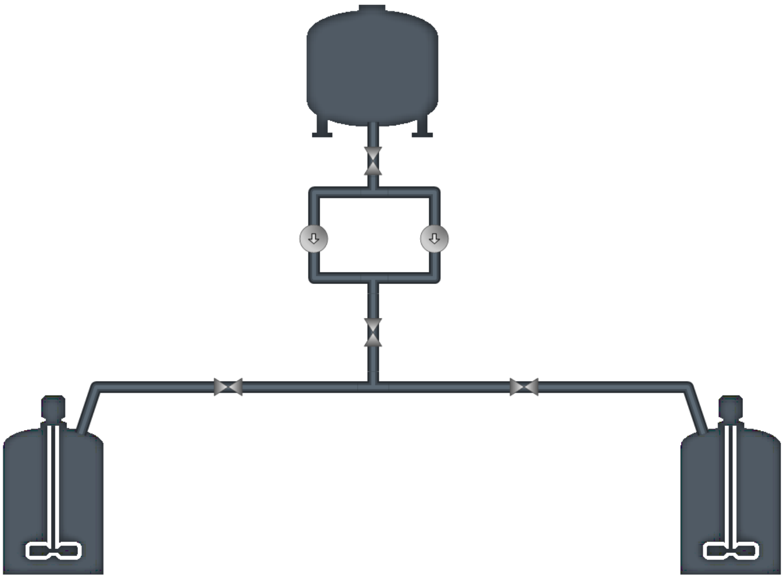
Using the WinCC Graphics Designer, you can export pictures but this is only possible in EMF format that SIMIT cannot read. This makes it necessary to convert to a different format using an external graphics program.

To minimize the amount of time and effort needed for this, the best thing to do is to use a screenshot tool (like the Windows 7 Snipping Tool, for example) to create the graphic.

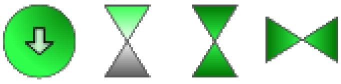
Proceed as follows:

1. Open the picture containing the SIMIT background in the Graphics Designer. Make sure that the display size is 100%.
2. Grab a screenshot and crop the unneeded areas. If you use the Snipping Tool, you can use the Rectangular Snip setting to select the necessary area when you grab the screenshot.

3. Save the picture in PNG format. The background image should look something like this:



4. Open the picture containing the SIMIT symbols in the Graphics Designer. Make sure that the display size is 100%.
5. In each case, grab a screenshot of the colored symbols and crop the unneeded areas.
6. The individual symbols should be in the same image format as was set in WinCC. For example: 40x40 pixels.
7. Save the individual symbols in PNG format. The symbols should look something like this:

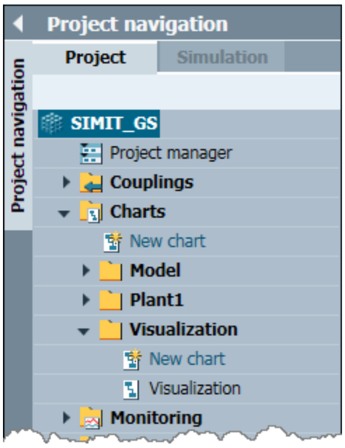


Note Use PNG-format graphics with a transparent background for the pictograms. This improves the display quality in the SIMIT project.

8.2 Creating visualization in SIMIT

To visualize the process, add the "Visualization" area to the folder structure. Proceed as follows:

1. In the shortcut menu of the "Charts" folder, choose the "New folder" item and rename the new folder "Visualization".
2. In the folder create a new chart called "Visualization".

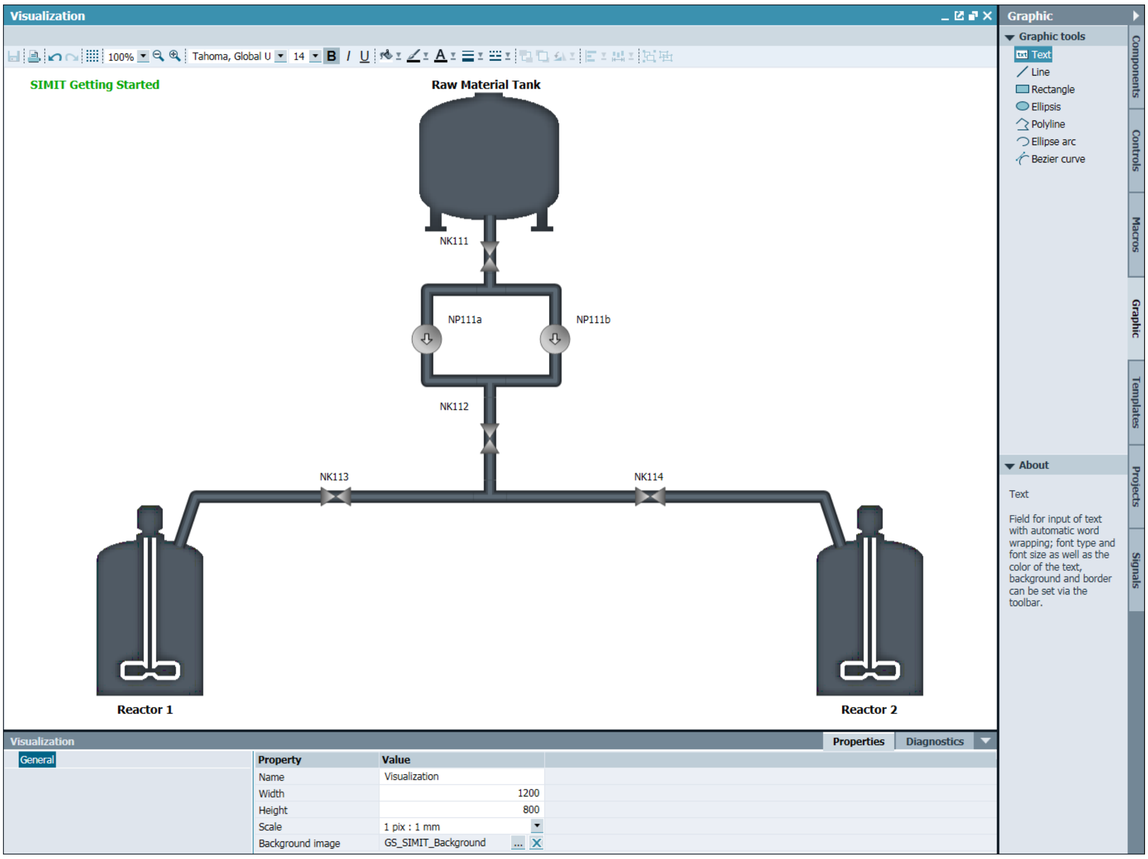


8.2.1 Creating a plant display

Follow the steps below to configure the background image:

1. Open the "Visualization" chart and adapt the size to the background image that you created before. In the example, the background image measures 1200 x 800 pixels
2. In the properties, choose the background image in "General > Background image".
3. Use the "Text" graphic object to insert labels into the picture.

Figure 8-5



8.2.2 Displaying process values

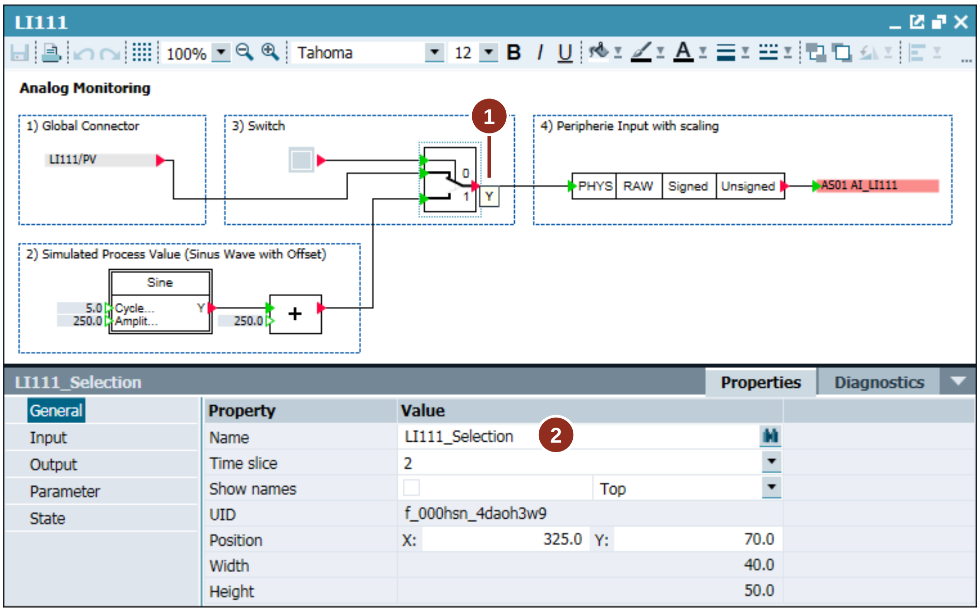
You can simply drag signals into the chart from the signal list. In this way, you insert the objects below into the chart:

- A text field containing the signal name
- A digital input control that has already been configured with the signal to be displayed
- A signal isolator for manual entry of a substitute value (forcing)

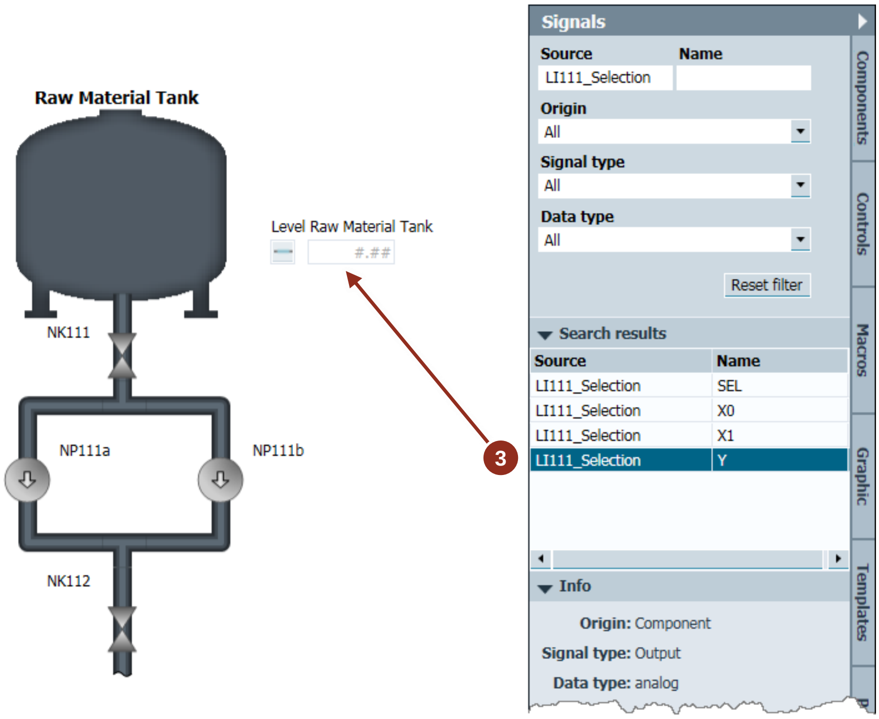
However, not every signal is suitable for display on the chart. For example: signal "AS01/LI111" (fill level of raw material tank) has already been converted into the raw format for the controller. To be able to display the fill level correctly on the chart, you can use a signal that is available before conversion.

Proceed as follows:

1. Open simulation chart LI111.
2. Select the last component (1) before conversion and make a note of the name (2) of the object.



3. Search for output signal "Y" of the object in the signal list and drag and drop it to the visualization chart.



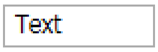
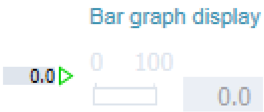
4. Change the label of the signal to "Level Raw Material Tank".
5. Add the signals below in the same way:
   - Reactor 1: "LI112_Selection / Y"
   - Reactor 2: "LI113_Selection / Y"

8.2.3 Configuring operator controls and displays

In the simulation, it is intended for the system to be able to display the fill levels of the containers using a simulated process value or a sine value. At pump drive "NP111a" it should be possible to make local inputs using four signals. It should be possible to visualize the servo position of analog valve "NK112" using a bar graph.

You need the following objects:

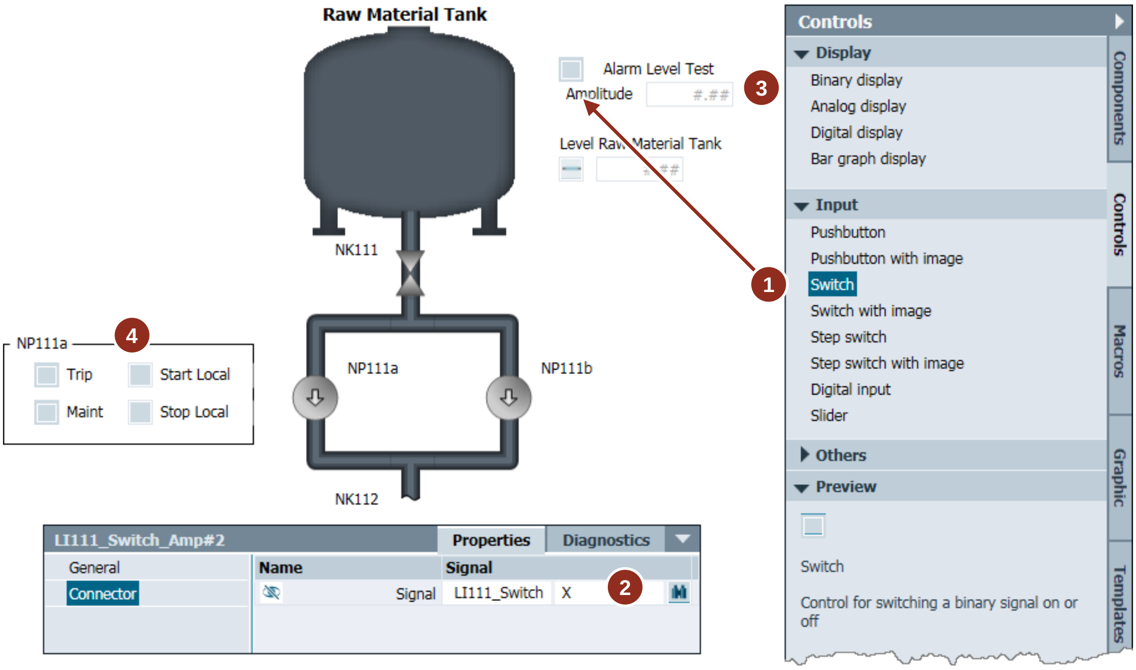
Table 8-1

Symbol	Name	Library	Description
	Text	Graphic > Graphic tools	Text box
	Switch	Controls > Input	Switch for a binary signal
	Digital input	Controls > Input	Input field for process values
	Bar graph display	Controls > Display	Graphic display for process value

Proceed as follows:

1. Using Controls > Input > Switch, add a switch (1) to the chart and parameterize the following properties:
   - General > Name = "LI111_Switch_Amp"
   - Connector > Signal = "LI111_Switch / X" (2)
2. Using Controls > Input > Digital Input, add an input field to the chart and parameterize the following properties:
   - General > Name = "LI111_DI_Amp"
   - Connector > Signal = "LI111_Sine / Amplitude"
3. Add text boxes with corresponding labels.

Figure 8-6: Inserting operator controls



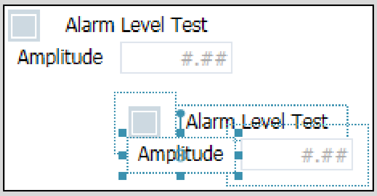
4. Use the same method to add controls and labels for reactors 1 and 2 and pump drive "NP111a" (4). Use the signals that are described in the table below:

Table 8-2

Control	Parameter	Value
Input > Switch	General > Name	LI112_Switch_Amp
	Connector > Signal	LI112_Switch / X
Input > Digital Input	General > Name	LI112_DI_Amp
	Connector > Signal	LI112_Sine / Amplitude
Input > Switch	General > Name	LI113_Switch_Amp
	Connector > Signal	LI113_Switch / X
Input > Digital Input	General > Name	LI113_DI_Amp
	Connector > Signal	LI113_Sine / Amplitude
Input > Switch	General > Name	NP111a_Switch_Trip
	Connector > Signal	NP111a_TripLocal_Push / X
Input > Switch	General > Name	NP111a_Switch_Maint
	Connector > Signal	NP111a_MaintLocal_Push / X
Input > Switch	General > Name	NP111a_Switch_StartL
	Connector > Signal	NP111a_StartLocal_Push / X
Input > Switch	General > Name	NP111a_Switch_StopL
	Connector > Signal	NP111a_StopLocal_Push / X

Note

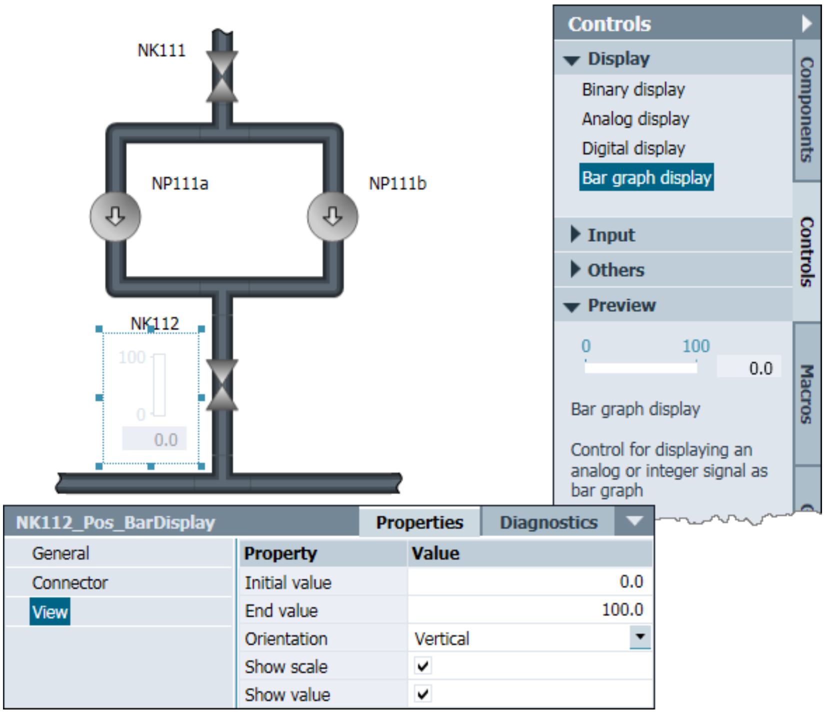
You can reduce the time and effort needed for configuration by selecting several objects and copying them. To do this, drag a frame around the desired objects and press <Ctrl+C> to copy them and <Ctrl+V> to paste them.



After this, change the parameters of the copied objects.

5. Use "Controls > Display > Bar graph display" to add a bar graph display with the parameters below to the chart:
   - General > Name = "NK112_Pos_BarDisplay"
   - Connector > Signal = "NK112_DirveV4 / Y"
   - View > Orientation = "Vertical"

Figure 8-7: Visualizing the valve position



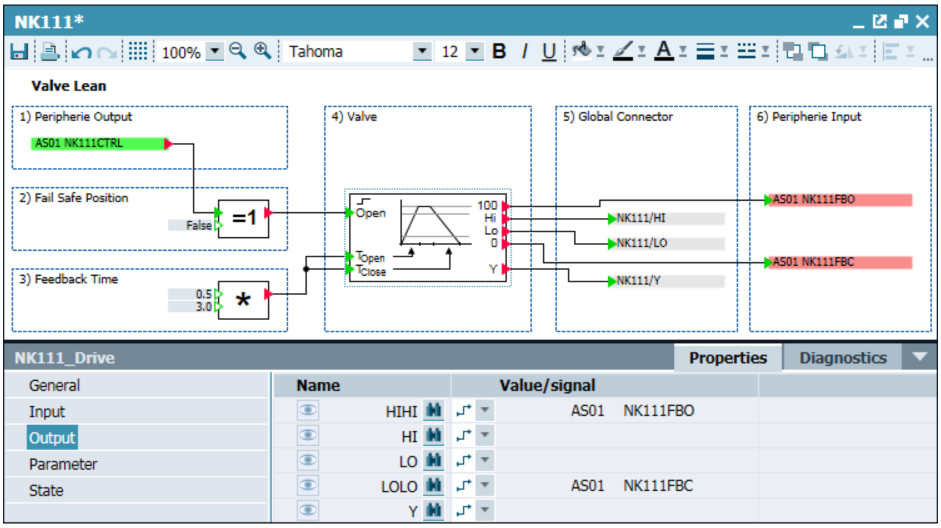
8.2.4 Animating device icons

You need several animated graphics in the chart to visualize the drives, the operating status of a valve, or the fill level of a container.

Below, we will show you how to dynamize objects from the "Graphic > Graphic tools" library. The additional "Animations" property is available for these objects.

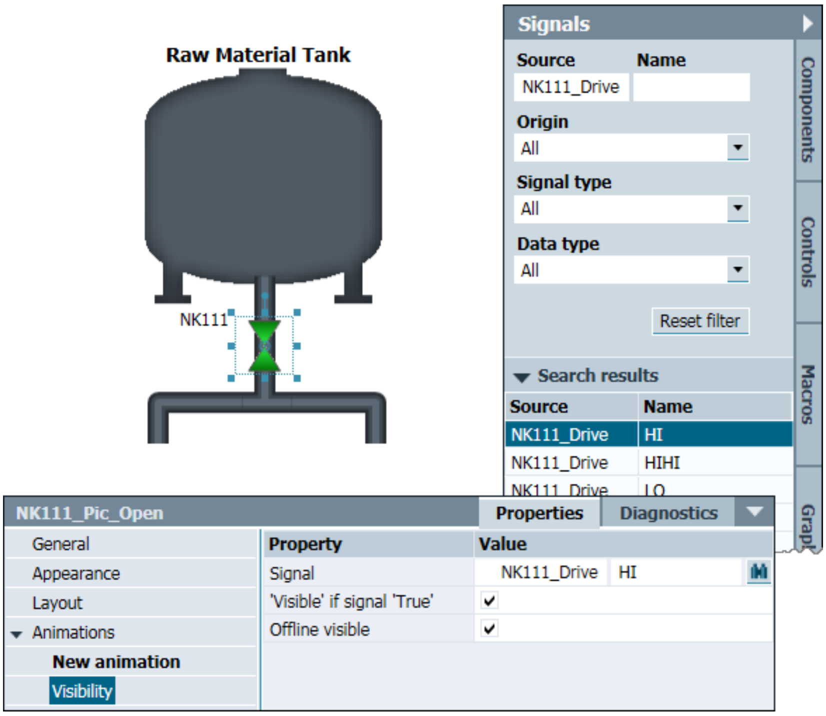
Follow the steps below to display an open valve:

1. Insert a rectangle in the chart with the following parameters: (The side length of the picture to be displayed is 40 pixels)
   - General > Name = "NK111_Pic_Open"
   - Appearance > Fill color > Image… = "GS_SIMIT_VlvL_90_Opened.png"
   - Appearance > Line color = "No color"
   - Layout > Width = "40.0"
   - Layout > Height = "40.0"
2. Position the picture directly above the representation of the closed valve NK111 in the background. If you use the supplied sample graphics, this is position X=566, Y=208.
3. In chart NK111, determine the signal that is set when the valve is open. For valve NK111, the signal is "NK111_Drive / Hi".



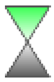
4. Add a new animation to the graphic object. To do this, double-click in Properties on "Animations > New animation". Create the animation "Visibility" using the following parameters:
   - Animations > Visibility > Signal = "NK111_Drive / Hi"
   - Animations > Visibility > Visible if signal 'True' = Yes

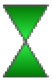
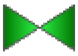
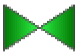
Figure 8-8: Animated valve symbol



5. Create six copies of the graphic object that you have just generated and change the parameters on the basis of the table below:

Table 8-3

Symbol	Parameter	Value
Motor NP111a 	General > Name	LI111a_Pic_On
	Appearance > Fill Color > Image	GS_SIMIT_MotL_Run.png
	Animations > Visibility > Signal	NP111a_DriveP1 / FB_Run
	Animations > Visibility > Visible if signal 'True'	Yes
Motor NP111b 	General > Name	LI111b_Pic_On
	Appearance > Fill Color > Image	GS_SIMIT_MotL_Run.png
	Animations > Visibility > Signal	NP111b_DriveP1 / FB_Run
	Animations > Visibility > Visible if signal 'True'	Yes
Valve NK112  (Display from 5%)	General > Name	NK112_Pic_Half
	Appearance > Fill Color > Image	GS_SIMIT_VlvL_90_Half.png
	Animations > Visibility > Signal	NK112_DriveV4 / LO
	Animations > Visibility > Visible if signal 'True'	No

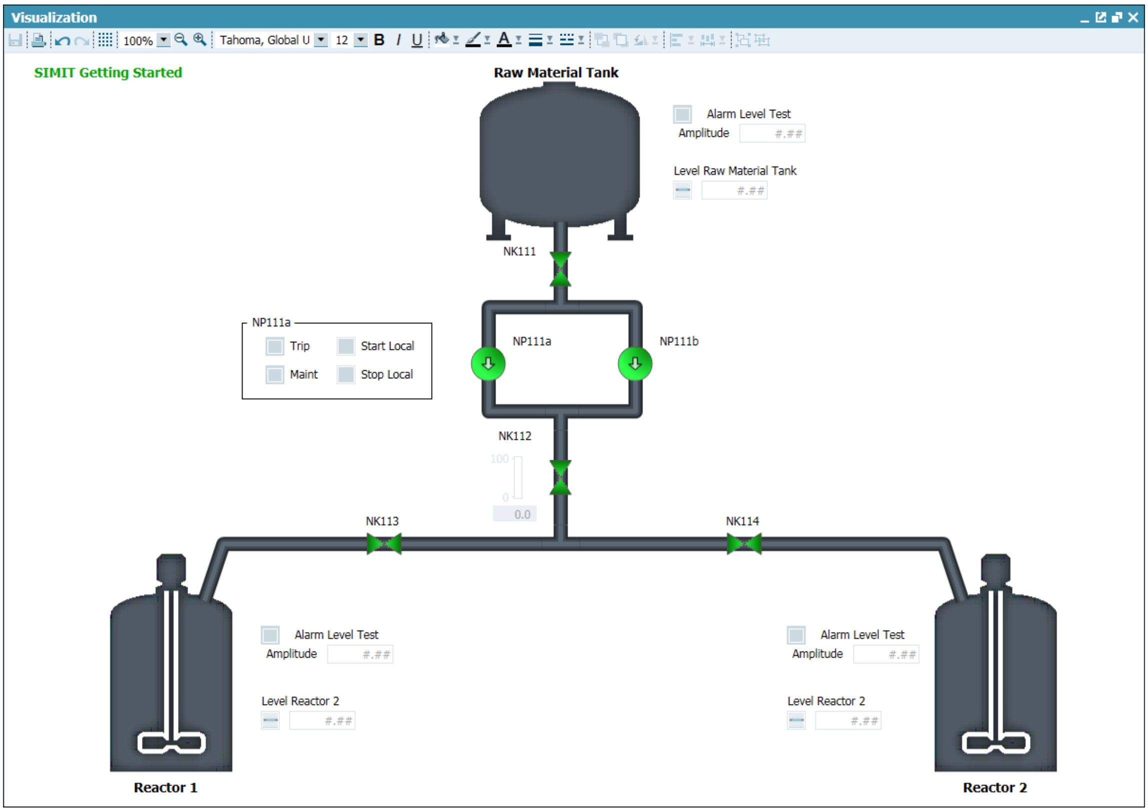
Symbol	Parameter	Value
Valve NK112  (Display from 95%)	General > Name	NK112_Pic_Open
	Appearance > Fill Color > Image	GS_SIMIT_VlvL_90_Opened.png
	Animations > Visibility > Signal	NK112_DriveV4 / HI
	Animations > Visibility > Visible if signal 'True'	Yes
Valve NK113 	General > Name	NK113_Pic_Open
	Appearance > Fill Color > Image	GS_SIMIT_VlvL_Opened.png
	Animations > Visibility > Signal	NK113_Drive / HI
	Animations > Visibility > Visible if signal 'True'	Yes
Valve NK114 	General > Name	NK114_Pic_Open
	Appearance > Fill Color > Image	GS_SIMIT_VlvL_Opened.png
	Animations > Visibility > Signal	NK114_Drive / HI
	Animations > Visibility > Visible if signal 'True'	Yes

Note

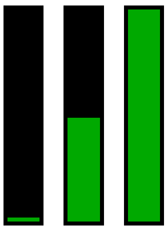
From 5% opening onward, valve NK112 shows the symbol for the intermediate position. From 95% onward, the system displays the symbol for the open valve. When configuring, make sure that the "open" symbol is above the "half-open" symbol. You can use the commands on the toolbar to change the display level.

6. Place all of the symbols exactly above the corresponding ones in the background image.

Figure 8-9: Visualization using animated symbols



8.2.5 Animating tank fill levels



To animate fill levels, you need two rectangles that are above one another in each case. The rear rectangle is the background and represents the empty tank. This rectangle is not animated. The size and position of the front rectangle are animated. It represents the fill level of the tank.

When the tank is 100% full, you can see a border of about one pixel around the rear rectangle, which gives the impression of a frame.

Proceed as follows:

1. Insert a "Rectangle" graphic object in the chart with the following parameters:
   - General > Name = "LI111_Bar_Back"
   - Appearance > Fill color = black
   - Appearance > Line color = "No color"
   - Layout > Position = X=610, Y=50
   - Layout > Width = 25
   - Layout > Height = 115
2. Insert another rectangle in the chart with the following parameters:
   - General > Name = "LI111_Bar_Front"
   - Appearance > Fill color = green
   - Appearance > Line color = "No color"
   - Layout > Position = X=611, Y=163

- Layout > Width = 23
- Layout > Height = 1

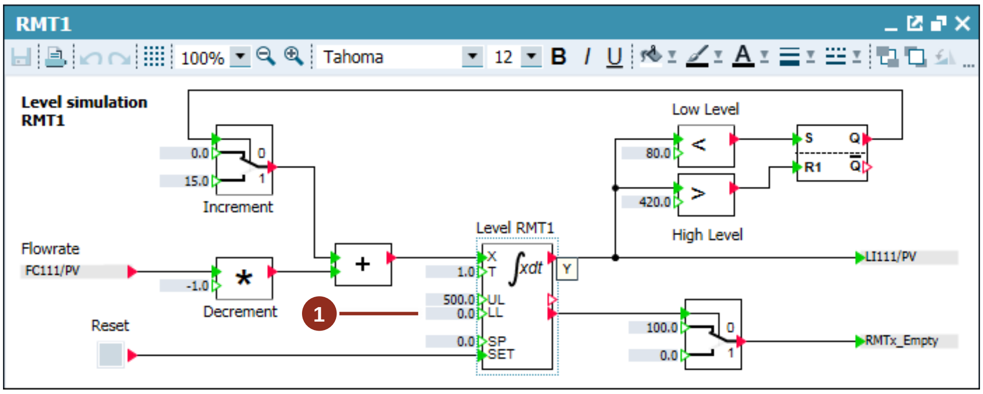
Note The size and positions of the rectangles depending on the background graphic that you use, of course. The coordinates that are stated here refer to the supplied background image.

This is now the initial representation of the bar graph. For animation, you must first determine a few values and make calculations. You need the parameters below for the animation:

- Process value (signal)
- Range (minimum and maximum fill level
- Scaling factor
- Relative distance

You determine signal (LI111_Selection / Y) for the process value as shown in chapter "8.2.2 Displaying process values". You can determine the minimum fill level and the maximum fill level (1) in chart RMT1.

Figure 8-10: Determining values in the chart



The object is only scaled in the Y direction starting from the parameterized height (1 pixel) and it is calculated using the following formula:

$$Scaling\ Y = 100 * Height + 100 = 100 * 113 + 100 = 11400$$

Note The height corresponds to the height of the rectangle when 100% filled.

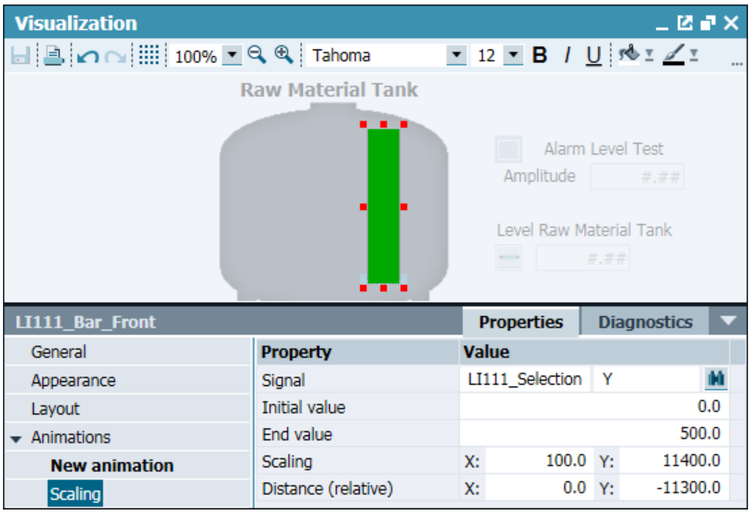
Since the rectangle is always drawn from the top right, you must use the formula below to calculate the distance relative to the height:

$$Distance\ Y = -1 * 100 * Height = -1 * 100 * 113 = -11300$$

3. Use "New Animation > Scaling" to create scaling for the "LI113_Bar_Front" object with the following parameters:
   - Scaling > Signal = "LI111_Selection / Y"
   - Scaling > Initial value = 0.0

- Scaling > End value = 500.0
- Scaling > Scaling = X=100.0, Y=11400.0
- Scaling > Distance = X=0.0, Y=-11300.0

Figure 8-11: Animated rectangle to represent a fill level



4. Create two copies of both rectangles for the reactors' fill level displays and change the parameters in accordance with the following table:

Table 8-4

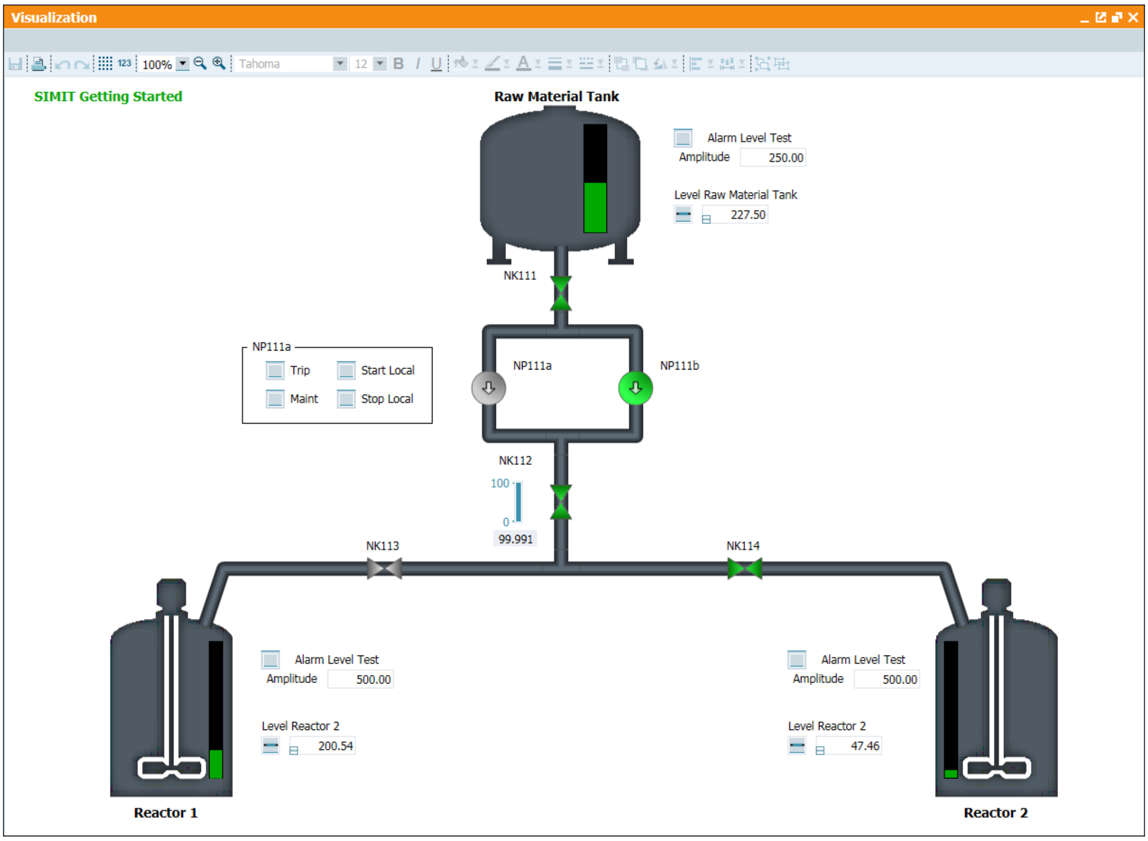
Object	Property	Value
Bar graph of reactor 1 background	General > Name	LI112_Bar_Back
	Layout > Position	X=215, Y=595
	Layout > Width	15
	Layout > Height	145
Bar graph of reactor 1 foreground	General > Name	LI112_Bar_Front
	Layout > Position	X=216, Y=738
	Layout > Width	13
	Layout > Height	
	Animations > Scaling > Signal	LI112_Selection / Y
	Animations > Scaling > Initial Value	0.0
	Animations > Scaling > End Value	1000.0
	Animations > Scaling > Scaling	X=100, Y=14400
	Animations > Scaling > Distance	X=0, Y=-14300
Bar graph of reactor 2 background	General > Name	LI113_Bar_Back
	Layout > Position	X=990, Y=595
	Layout > Width	15
	Layout > Height	145
Bar graph of reactor 2 foreground	General > Name	LI113_Bar_Front
	Layout > Position	X=991, Y=738
	Layout > Width	13
	Layout > Height	
	Animations > Scaling > Signal	LI113_Selection / Y

Object	Property	Value
	Animations > Scaling > Initial Value	0.0
	Animations > Scaling > End Value	1000.0
	Animations > Scaling > Scaling	X=100, Y=14400
	Animations > Scaling > Distance	X=0, Y=-14300

8.3 Current version

In chapter 8, you added a graphical user interface to the simulation. You are now in a position to monitor the simulation in SIMIT and to make some settings.
The system shows the process values of the tank fill levels as text and as graphics. The icons of the drives and the valves display the current status.

Figure 8-12: Graphical user interface in SIMIT.



9 Appendix

9.1 Service and Support

Industry Online Support

Do you have any questions or need assistance?

Siemens Industry Online Support offers round the clock access to our entire service and support know-how and portfolio.

The Industry Online Support is the central address for information about our products, solutions and services.

Product information, manuals, downloads, FAQs, and application examples – all the information you need is accessible with just a few mouse clicks at: https://support.industry.siemens.com

Technical Support

The Technical Support of Siemens Industry provides you fast and competent support regarding all technical queries with numerous tailor-made offers – ranging from basic support to individual support contracts.

Please send queries to Technical Support via Web form: www.siemens.de/industry/supportrequest

Service offer

Our scope of services includes, inter alia, the following:

- Product trainings
- Plant data services
- Spare parts services
- Repair services
- On-site and maintenance services
- Retrofitting and modernization services
- Service programs and contracts

You can find detailed information on our range of services in the service catalog: https://support.industry.siemens.com/cs/sc

Industry Online Support App

The "Siemens Industry Online Support" app provides you with optimum support, including while on the road. The app is available for Apple iOS, Android and Windows Phone: https://support.industry.siemens.com/cs/ww/en/sc/2067

9.2 Links and Literature

Table 9-1

No.	Topic
\1\	Siemens Industry Online Support https://support.industry.siemens.com
\2\	Link to the entry https://support.industry.siemens.com/cs/ww/en/view/77362399
\3\	"SIMATIC SIMIT (V9.0 SP1)" manual https://support.industry.siemens.com/cs/ww/en/view/109744640
\4\	SIMIT Simulation Overview https://support.industry.siemens.com/cs/ww/en/view/109746429

9.3 Documentation of changes

Table 9-2

Version	Date	Change
V1.0	06/2017	First version