QmixELEMENTS

Manual

Software

October 2015

cetoni

Software License

The software and the documents provided are protected by copyright. Installation declares your acceptance of the conditions of the licensing contract.

License Agreement

cetoni GmbH grants the purchaser the non-exclusive and non-transferable license right to use the software on a standalone computer or a networked computer system (LAN). It is expressly forbidden to copy or duplicate the software in part or in whole in any way. The same is applicable for mixing and combining the software with other software. The purchaser may create one back-up copy of the software for archival purposes. cetoni GmbH reserves the right to alter, develop and improve the software or develop new software to replace it. cetoni is not obliged to inform the purchaser about any alterations, new developments or improvements to the software or make these available to him. No legally binding assurance of certain features is given. cetoni GmbH is not liable for damages unless the damages are as a result of wilful intent or gross negligence on the part of cetoni GmbH or its employees. Any liability for indirect, collateral, or consequential damages is excluded.

cetoni GmbH

Wiesenring 6

D- 07554 Korbußen

Phone: +49 36602 338-0

Fax: +49 36602 338-11

E-Mail: info@cetoni.de

Internet: www.cetoni.de

1 Summaries and directories

1.1 Table of contents

1.2 Change history

Revision	Change
20110722	Creation of software manual QmixElements
20111215	Documentation neMESYS-, QmixV- and Qmix Q- Plugin added Scripting system documented
20120312	Documentation of new core script functions added (Show Message und Interrupt Script)
20120319	Documentation of new core script functions added(Parallel Sequence)
20120522	Documentation of tubing pump plugin, image analysis plugin and video builder plugin added
20120523	Variable system documented device script functions documented
20120724	Improvements and changes of variable system documented Updated core script functions documentation Documentation of I/O channel scaling and configuration added Added documentation for Conditional Loop and Conditional Sequence
20120816	Added chapter about creating device configurations Updated rotAXYS Documentation (new features *Add Move XY Script* and *Add Move Z Script* and Space Navigator usage documented)
20120927	Documentation of rotAXYS / neMAXYS plugin updated (documented manually setting of speeds and documented script based speed setting)
20121012	Added documentation of *User Input* function, variable output via *Show Message* function documented properly
20121114	Added documentation for Data Logger and Camera plug-ins
20121220	Documented how to change function captions
20130118	Added process data graph documentation
20130206	Add documentation for *Create Property Variable* function Add documentation for single axis control functionality (manual and

	script functions) Updated neMESYS continuous flow documentation with new features Added documentation for PID Control function
20130311	Added documentation of Startup Screen Added documentation for process data graph curve color selection
20130402	Added documentation for Device Configurator functionalities Add and Exchange devices
20130508	Added documentation for neMESYS continuous flow cross flow feature
20130523	Updated Valve-Plugin documentation Updated Controller-Plugin documentation, added documentation for controller channel scaling Added camera video recording feature documentation Documented Well Plate Configuration dialogue
20130715	Updated Tubingpump-Plugin Added documentation for tubingpump script functions
20130718	Added documentation for synchronous neMESYS pump start / stop
20130821	Added documentation for LED array plugin
20130828	Added documentation for neMESYS continuous overlap flow
20131014	Added documentation for neMESYS change flow rate functionality
20131119	Added documentation for neMESYS I/O channels
20140127	Updated documentation of Qmix Controller Plugin Added documentation for user-defined controller channels
20140214	Added documentation for device process data identifiers and direct device property access to section Script System Updated documentation of PID controller script function
20140909	Inserted documentation of the analog channel monitoring functionality of the neMESYS plugin and a section about predefined analog channel confiugurations in the qmixio plugin chapter.
20141104	Documented functionality linking neMESYS pumps and scripting system. Added section about creating device configurations with simulated

	devices.
20150105	Updated LED array documentation for new LED array V2 devices
20150629	Added documentation for spectroscopy plugin
20150723	Updated rotAXYS plugin documentation
20151021	Updated documentation for new syringe configuration Updated a number of neMESYS screenshots Added documentation for virtual I/O channels (Virtual Channels) New script function *Write Device Property* documented

2 Introduction

2.1 Foreword

Thank you for deciding to purchase a cetoni product. We would like to support you with this handbook insofar as possible in your interaction with the software. We are directly available for any questions or suggestions that you may have.

2.2 Symbols and Signal Words Used

The following symbols are used in this manual and are designed to aid your navigation through this document:

HINT

Describes practical tips and useful information to facilitate the handling of the software.

IMPORTANT

Describes important information and other especially useful notes, in which no dangerous or damaging situations can arise.

ATTENTION

Indicates a potentially damaging situation. Failure to avoid this situation may result in damage to the product or anything nearby.

CAUTION

Describes a situation that may be dangerous. If this aspect is not avoided, light or minor injuries as well as damage to property could result.

3 Installation

3.1 Installing the Software

IMPORTANT

In order not to impair time-critical control processes of the QmixElements software, no other computation-intensive applications should be executed on the controller PC.

IMPORTANT

Install the QmixElements software + device drivers before connecting your device via USB to the PC.

To install the software, insert the QmixElements CD ROM into your CD/DVD drive, or plug the QmixElements USB stick into a free USB port. Then start the file QmixElements_Setup.exe from the CD or USB stick. The Installation Wizard then leads you through the installation of the QmixElements software and the hardware drivers.

IMPORTANT

Under Windows, you need to be logged in with administrator rights in order to be able to install the hardware drivers.

The Installation Wizard now leads you through the installation of the software.

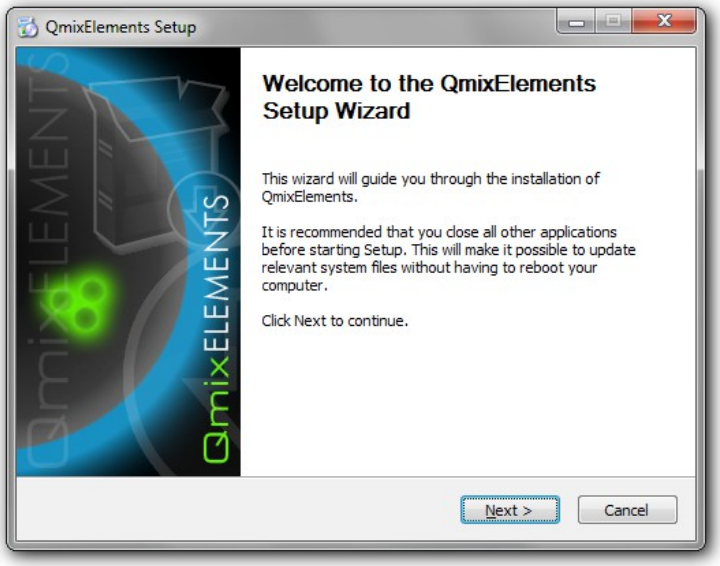


Figure 1: Setup Wizard Welcome Page

The hardware device drivers are installed during the installation. This step is only required if the drivers are not already installed on your computer. If the hardware drivers are already installed, please deactivate the VCI Driver component (Figure below).

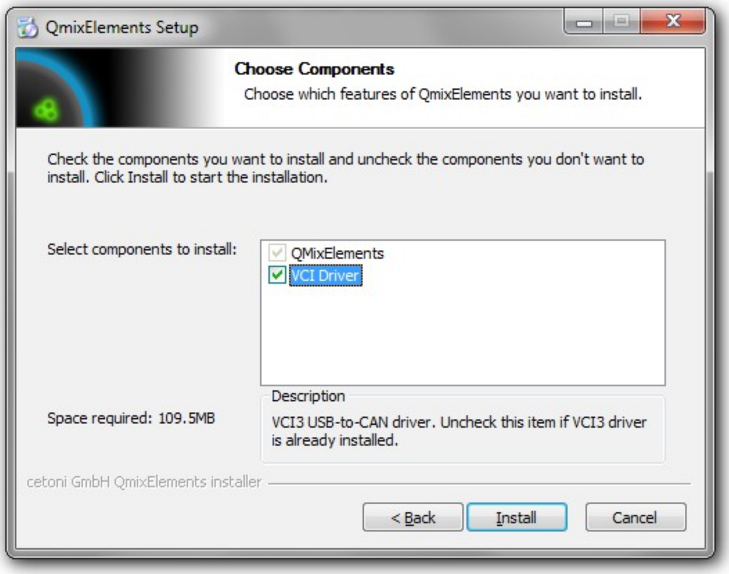


Figure 2: Selection of installation components

Your computer should meet the following system requirements in order to be able to use the software:

- PC with a Pentium processor (or higher) - minimum 1.3 GHz
- at least 1 GB RAM (recommended 2 GByte).
- free hard disk space of approx. 200 MByte
- at least 2 free USB (1.1 or 2.0) interfaces
- Operating systems: Windows 7, Vista or Windows XP (recommended Windows 7)
- wheel mouse

Standby or sleep mode must be deactivated under Windows when the software is used, because activation of the standby or sleep mode can lead to malfunctions of the hardware device driver.

ATTENTION

Danger of standby or sleep mode causing malfunctions and/or data loss. Deactivate standby / sleep mode on your PC or notebook to avoid malfunctioning of the hardware driver.

3.2 Installing VCI USB Device Drivers

The VCI USB device drivers are required for the USB connection to your device. When you connect your device to your PC via USB for the first time or connect the device some time later to another USB slot, Windows starts the Hardware Wizard automatically, this detects a new USB device and installs the requisite drivers.

IMPORTANT

The installation procedure for the device drivers can vary slightly depending on the Windows version used.

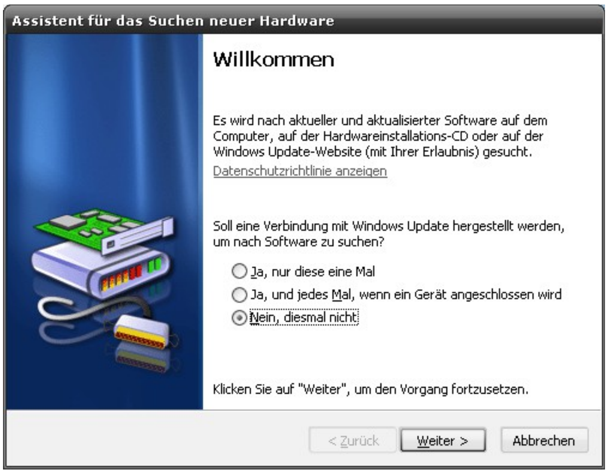
ATTENTION

Danger of data loss as a result of uncontrolled switch off procedures! Always close the software before you switch off your device! Only then are all settings saved correctly, and the configuration data will not be lost.

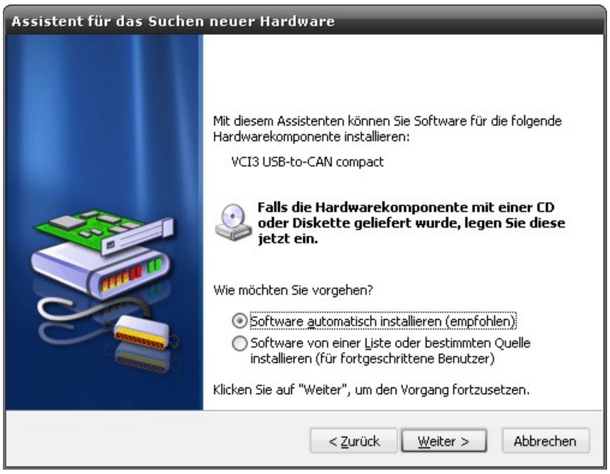
3.2.1 Installing VCI USB drivers for Windows XP

(1) The Hardware Wizard detects a new device at the USB port.

(2) The following dialog appears, which you configure as shown in the diagram, and confirm with *Next*.



(3) In the following dialog, please select an automatic installation, and confirm this dialog with *Next*.



(4) Windows finds a driver for the new device, and the following dialog appears.

(5) Click the *Finish* button to complete the installation.

IMPORTANT

Under Windows XP, you need to be logged in with administrator rights in order to be able to install new hardware.

After the driver has been successfully installed, you can start the software.

3.2.2 Installing VCI USB drivers for Windows 7

When you connect your device under Windows 7 for the first time, the device drivers are installed automatically if you have previously run the software setup from the shipped CD or USB stick.

4 Introduction to QmixElements Software

4.1 Initial Operation

The first time the software is started, a default configuration is loaded: The software loads only its core without any device-specific plugins. The default configuration is active, when you see the following startup screen (figure below).

Figure 3: First start of QmixElements – Startup Screen

You have to install and load a device configuration, in order to be able to control your devices with the software. In the startup screen you can see all ways to load a valid device configuration:

- *Open Configuration* – This allows you to load an existing device configuration that was previously installed on your system.
- *Import Configuration* – Select this menu item to install a device configuration from a disk or USB-stick (eg from QmixElements installation CD) – see chapter 4.2 Import Device Configuration.
- *Create Configuration* – This button starts the *Qmix Device Configurator* that lets you create arbitrary device configurations yourself. More information on creating device configurations can be found in chapter 5 Creating Device Configurations.
- *Start Demo* – If you want to start the application in demo mode, eg for presentations and tests or if you currently have no real equipment, then click this button.

WICHTIG

Also after a system crash or after an update of the software, the default configuration is loaded, and the startup screen appears.

4.2 Import Device Configuration

If you have devices that are not yet supported by the device configurator, you will find a valid device configuration on the installation disk that you received with your device.

Perform the following steps to install an existing device configuration:

HINT

If you create a configuration with the device configurator, the configuration will be installed automatically and you don't need to perform the following steps.

(1) Select the *Device → Import Configuration* menu item from the main menu.

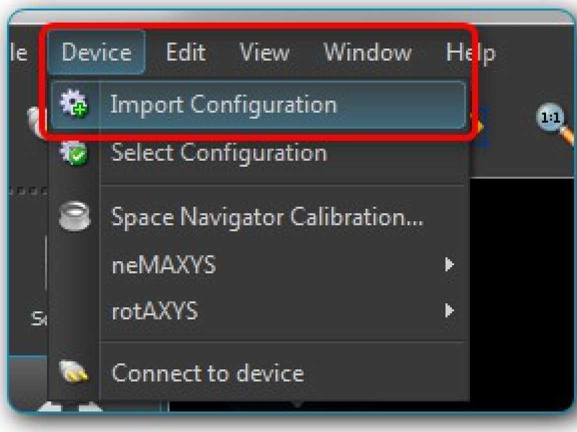


Figure 4: Import device configuration

(2) In the dialog that now opens, select the *DeviceConfiguration* directory on the installation CD or browse to another directory containing a valid device configuration.

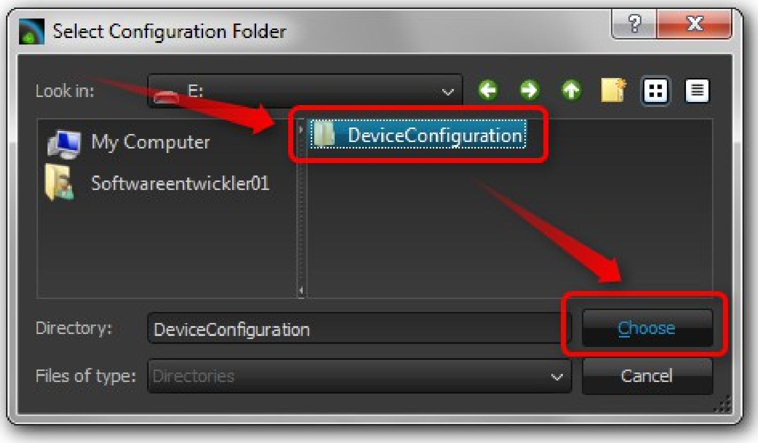


Figure 5: Directory dialog for selecting a configuration

(3) Now enter a unique name for the configuration under which it will be stored on your computer.

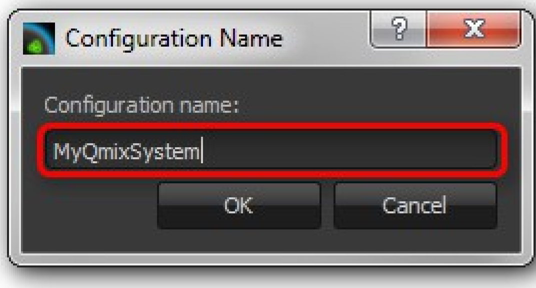


Figure 6: Assign a configuration name

(4) Acknowledge the query to restart the application.

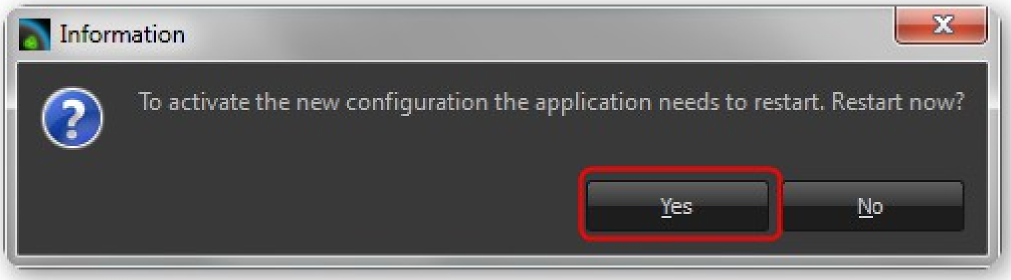


Figure 7: Restart application

After completing these steps, the software with the device-specific plugins will start to load.

4.3 Activation of QmixElements

In order to use the QmixElements software to it's full extend, you have to activate it following these steps:

(1) Start the activation dialog by left-clicking *Edit → Activate Product Key*.

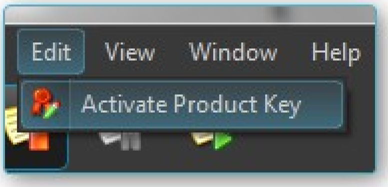


Figure 8: Menu Activate Product Key

(2) Enter the activation code provided with your QmixElements CD into the input field 1 and left-click Ok 2.

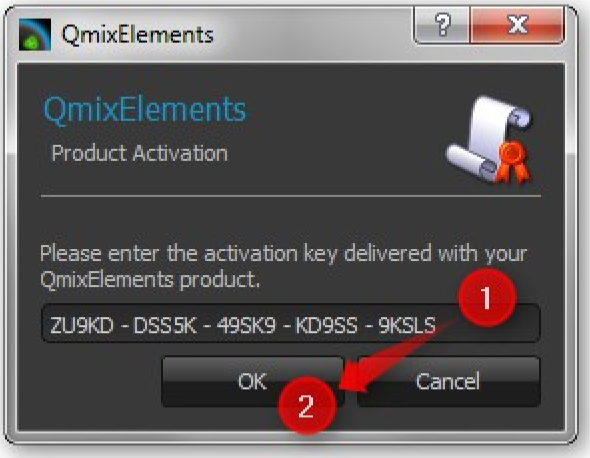


Figure 9: Activation Dialog

(3) QmixElements shows a successful activation.

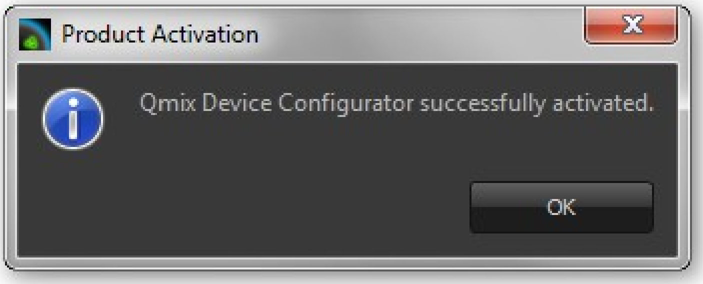


Figure 10: Activated QmixElements

4.4 QmixElements – The Basics

4.4.1 Introduction

The QmixElements application is a plugin-based, modular software solution for controlling a wide range of laboratory automation devices, in particular cetoni's Qmix modules. The software consists of a core, which provides the basic functions and services. This includes the application window, the event log, and the toolbar.

This core system is then expanded by device- and application-specific software modules, referred to as plugins. They provide the user with specific software solutions appropriate for virtually any device configuration within the standardized user interface.

4.4.2 Show User Manual

The software manual can be accessed via the application's help menu (*Help* → *Show Manual*).

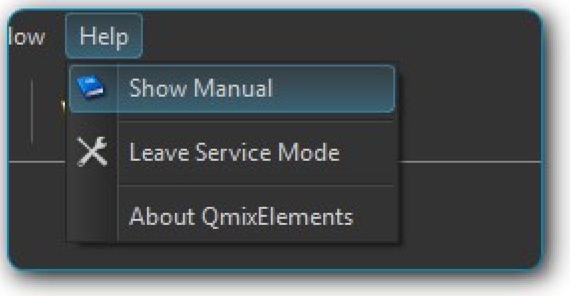


Figure 11: Access software manual

WICHTIG

The software manual is provided in the pdf file format. A pdf-viewer must be installed on your system to be able to view this document.

4.4.3 Show Software Version

If you need to know the version of the installed QmixElements, e.g., for service purposes, select *Help → About QmixElements* from the main menu (see figure below).

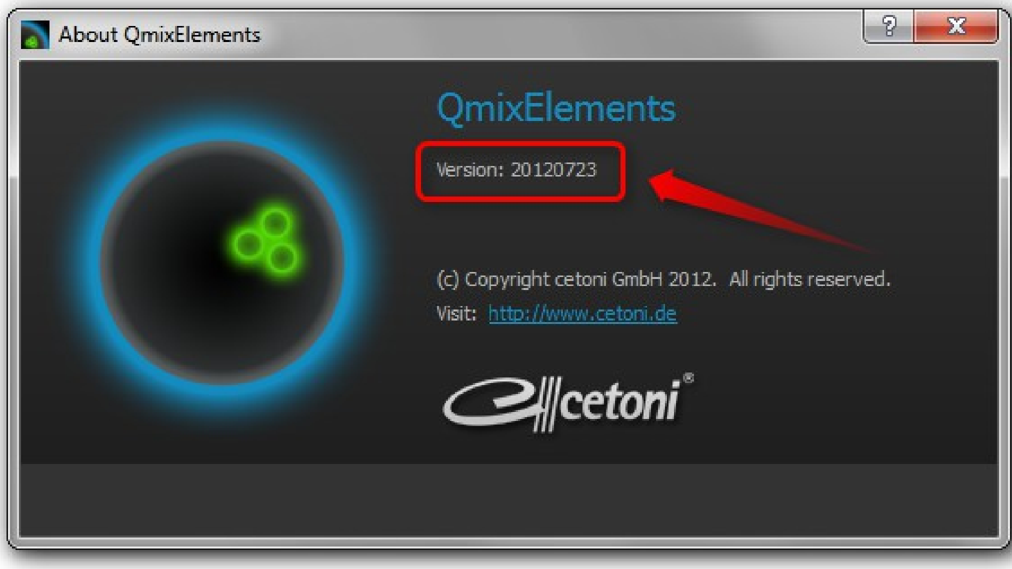


Figure 12: Displaying the installed software version

4.5 Overview of the Main Application Window

The application window of the software consists of the following central components:

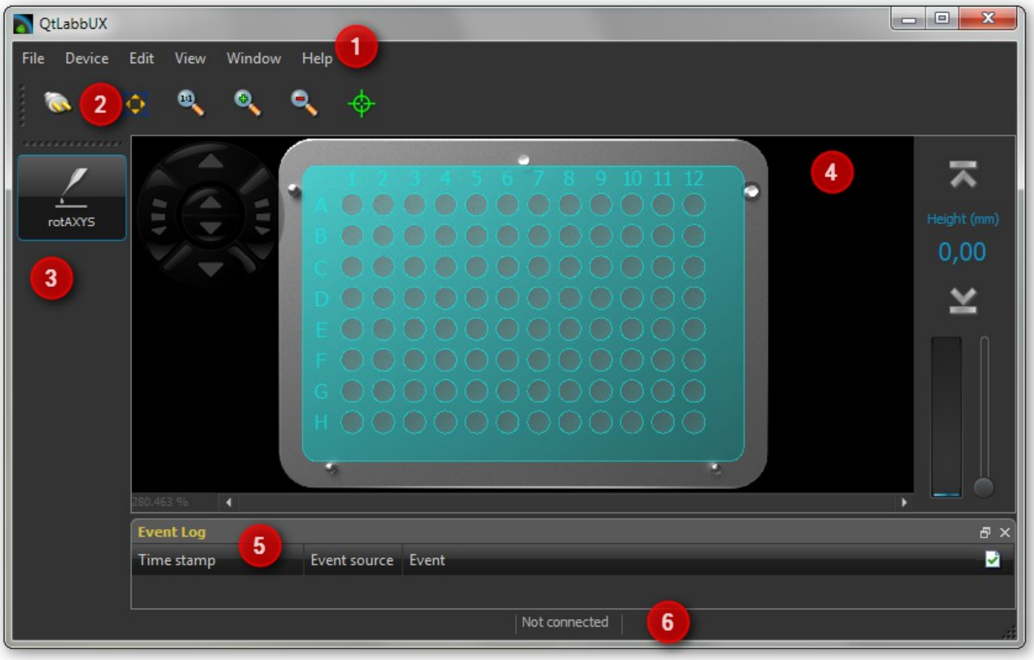


Figure 13: Overview of application Main window

1 Main menu

2 Toolbars (can be freely positioned)

3 Sidebar for selecting workspaces (can be freely positioned)

4 Controls of the selected plugin-specific workspace

5 Event Log

6 Status bar

4.6 Sidebar

There is a button in the sidebar for activating each registered workspace. Each plugin can register any number of workspaces. However, in most cases, plugins register only one specific workspace. Click the sidebar buttons to switch between the available workspaces.

HINT

The sidebar and all toolbars can be freely positioned and arranged within the periphery of the main window.

4.7 Event Log

Important events are shown in an event log. Such events include instructions, warning messages, and error messages. You can delete the event log by clicking the *Clear Event Log* button in the top right-hand corner (see figure).

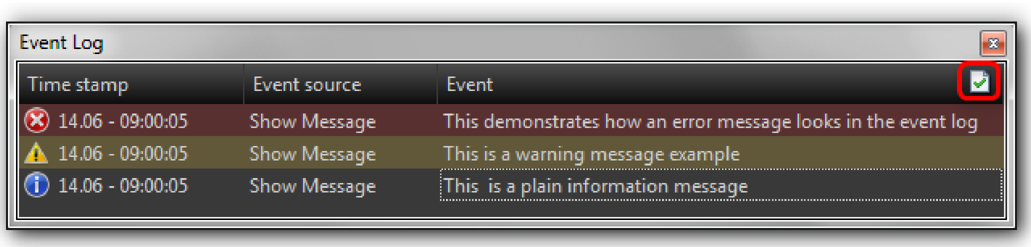


Figure 14: Event log

4.8 Status Bar

The status bar displays a variety of information, such as the connection status to the device.

4.9 Connecting to the Device

In the toolbar, you see the *Connect to device* button signified by the “plug” icon (see figure below). Press this button to connect QmixElements with your device(s).

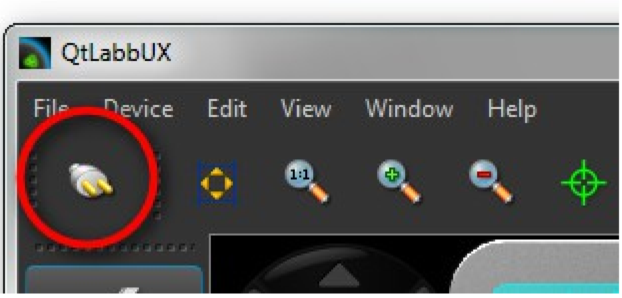


Figure 15: Connecting your devices

IMPORTANT

The device must be connected to the PC via USB and the power supply to the device must be switched on to connect to the device.

4.10 Style Selection

The design of the graphical user interface of the QmixElements software can be customized via Styles. The software installation already contains a number of ready-made styles. To change the current style, select the main menu item *Window → Style Selection*.

The style selection dialog will be shown (figure below) and you can select from a number of available styles 1. In the area *Preview* 2 you will get a small preview of the style. After clicking *OK* and restart the software, the graphical user interface appears with the selected style.

Figure 16: Style Selection Dialog

5 Creating Device Configurations

5.1 Introduction

In order to combine single devices according to your requirements and use them in QmixElements, you have to create a device configuration. Creating device configurations can be done using the *Device Configurator* which is integrated in the QmixElements software. The Device Configurator features the *Device Configuration Wizard* that guides you through the integration of your devices into your device configuration.

5.2 Starting the Device Configurator

You can start the Device Configurator by selecting the menu item *Device* → *Create* Configuration from the main menu.

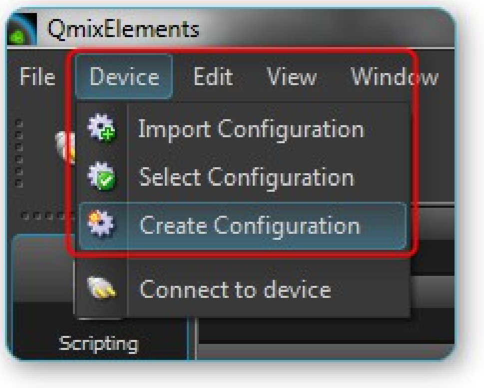


Figure 17: Opening the Device Configurator

5.3 Import a device database

The device configurator and the configuration wizard use a device database (*devices.db*) to read device-specifc parameters and settings for configuration of the devices.

During the installation of the QmixElements software this database is already installed. If you use devices that are not yet listed in the device database (e.g. if your devices are newer than the latest QmixElements software version), you may need to import an updated version of the device database. This database can be found either on your installation media or your get it directly from cetoni.

When you start the device configurator, you will be asked if you want to import a new device database. If your installation media contains the file *devices.db*, you should import it now.

HINT

If your installation media does not contain a device database, your devices are already supported by the QmixElements software and you can skip the device database import.

(1) Confirm the query by left-clicking the *Yes* button.

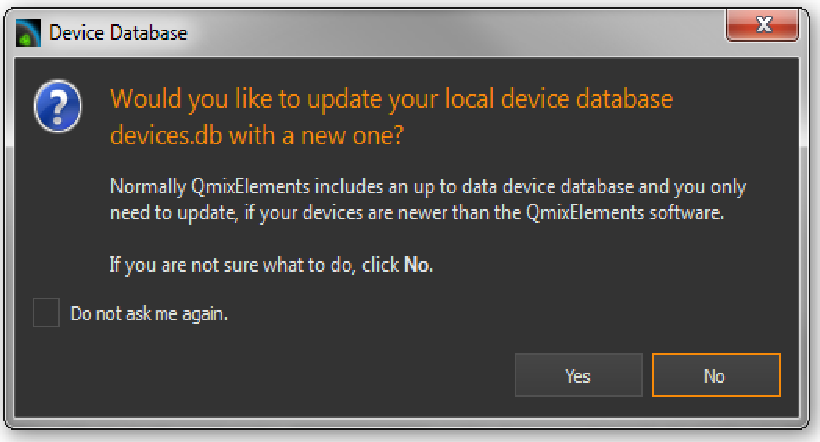


Figure 18: Query for device database import

(2) A file selection dialog opens. Select the database file *devices.db* located in the root directory of the QmixElements CD (see figure below).

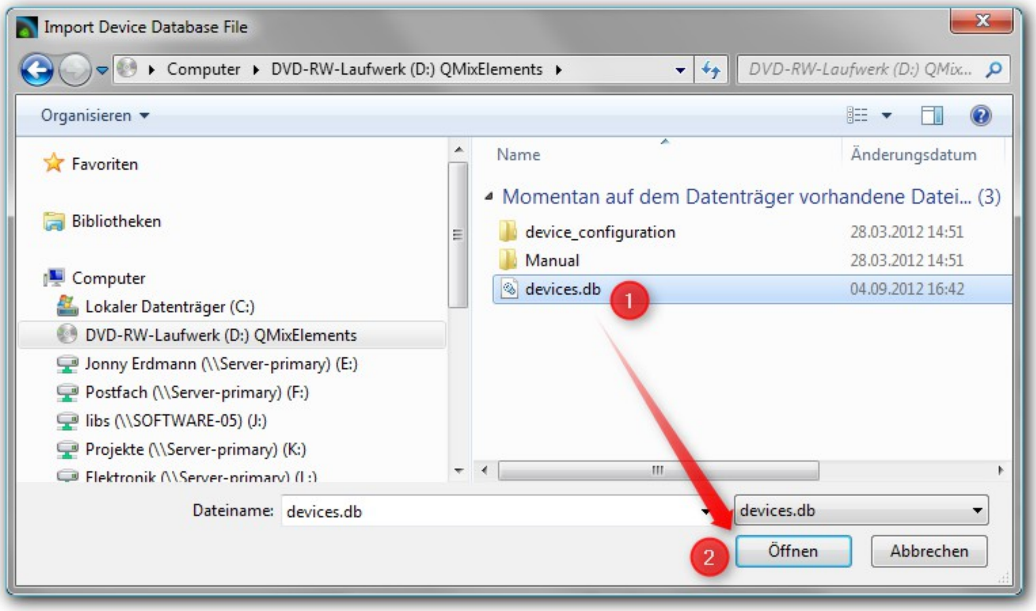


Figure 19: Device database import dialog

(3) The device configurator now has an actual device database available.

IMPORTANT

The device database contains important configuration information about your devices. You always have to import an actual device database if you purchase new devices from the cetoni GmbH.

As long as you don't purchase new devices you needn't import a device database on start of the device configurator. If you don't want QmixElements to further ask for database import just check *Do not ask again if import is not requested* and confirm by left-clicking *No* (see figure below).

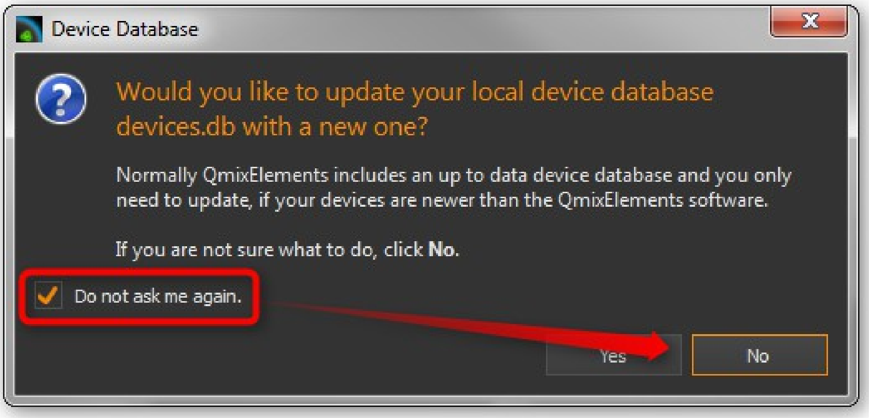


Figure 20: Deselect device database import

If you want to import a device database in the future you can reach the import

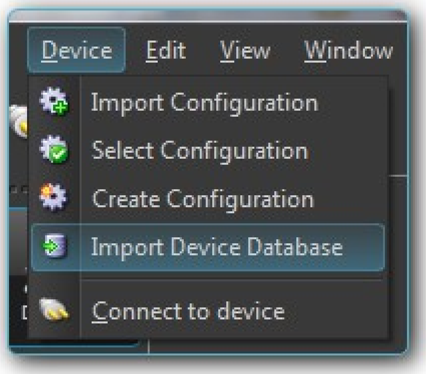


Figure 21: Import device database menu item

dialog by selecting *Device → Import Device Database* from the main menu (see figure below).

HINT

You dont't have to import a device database on each start of the device configurator. As long as you don't purchase any new devices, you dont't need to import a new device database.

5.4 Activating the Device Configurator

The Device Configurator is a liable to activation product within QmixElements. In a not activated software version you can test the creation of device configurations, but you are not allowed to configure devices using the Device Configuration Wizard 1. You recognize a not activated software version at a warn message shown in the status bar of the Device Configurator 2. The activation of QmixElements is discussed in chapter 4.3.

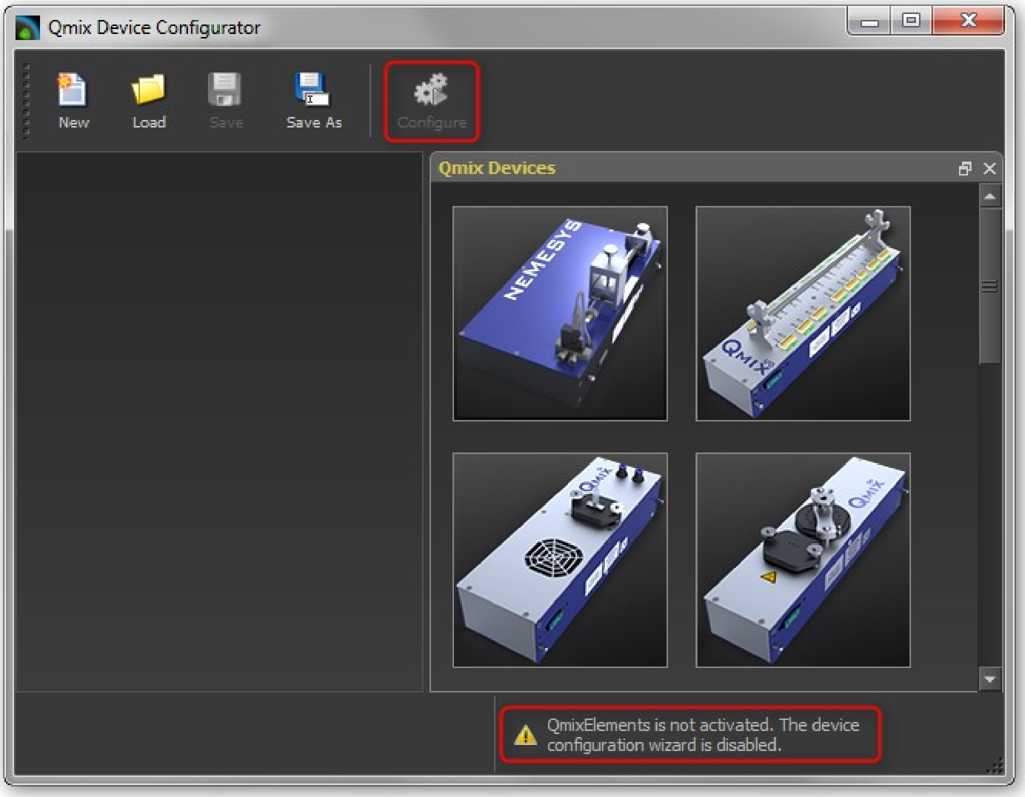


Figure 22: Not activated version of QmixElements

5.5 Overview

The Device Configurator is divided into the following three parts:

Figure 23: Device Configurator overview

1 Toolbar – Functions for creating, loading and saving device configurations, as well as starting the Device Configuration Wizard.

2 Device List – Selection of available devices.

3 Device configuration created by user.

5.6 Toolbar

The toolbar contains the following functions:

Creates a new, empty device configuration.

Loads an existing device configuration.

Saves the created device configuration.

Saves the created device configuration using a new name.

Sets all devices of the created device configuration to simulated.

Starts the Device Configuration Wizard.

5.7 Device Configuration Wizard

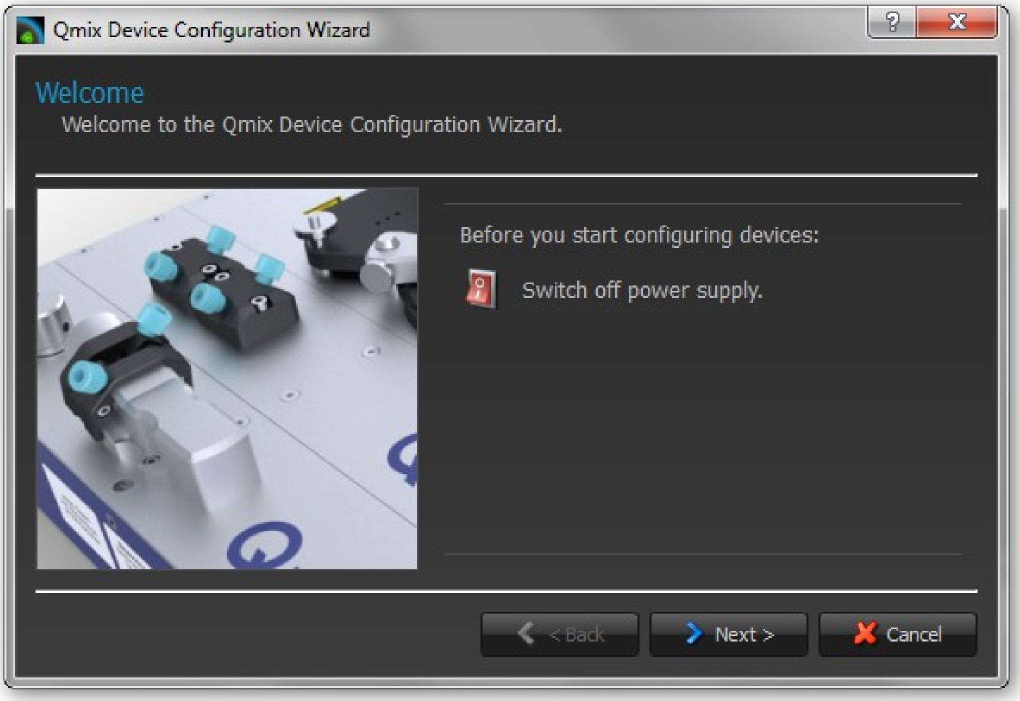


Figure 24: The Device Configuration Wizard

The Device Configuration Wizard guides you through the integration of your devices into your device configuration. You must execute the Device Configuration Wizard after:

- creating a new device configuration or changing an existing one.
- swapping devices or integrating a new device into an existing device configuration.

IMPORTANT

Device configurations and devices belong together. Always execute the Device Configuration Wizard if you want to change a device configuration or integrate a new device into an existing device configuration.

5.8 Creating a sample device configuration

The following example describes the creation of a device configuration containing a neMESYS dosing module and a Qmix Q+ module.

(1) Left-click on the neMESYS dosing module listed in the device list 1 and hold down the mouse button.

(2) Drag the neMESYS dosing module into the device configuration 2 holding the mouse button down. As soon as you release the mouse button the neMESYS dosing module is added to the device configuration.

(3) Start the Device Configurator as described in section 5.2.

HINT

You can remove devices from the device configuration by selecting the device by clicking it and then pressing the Delete key.

(4) Proceed in the same way with the Qmix Q+ module.

(5) Start the Device Configuration Wizard by left-clicking *Configure* 3.

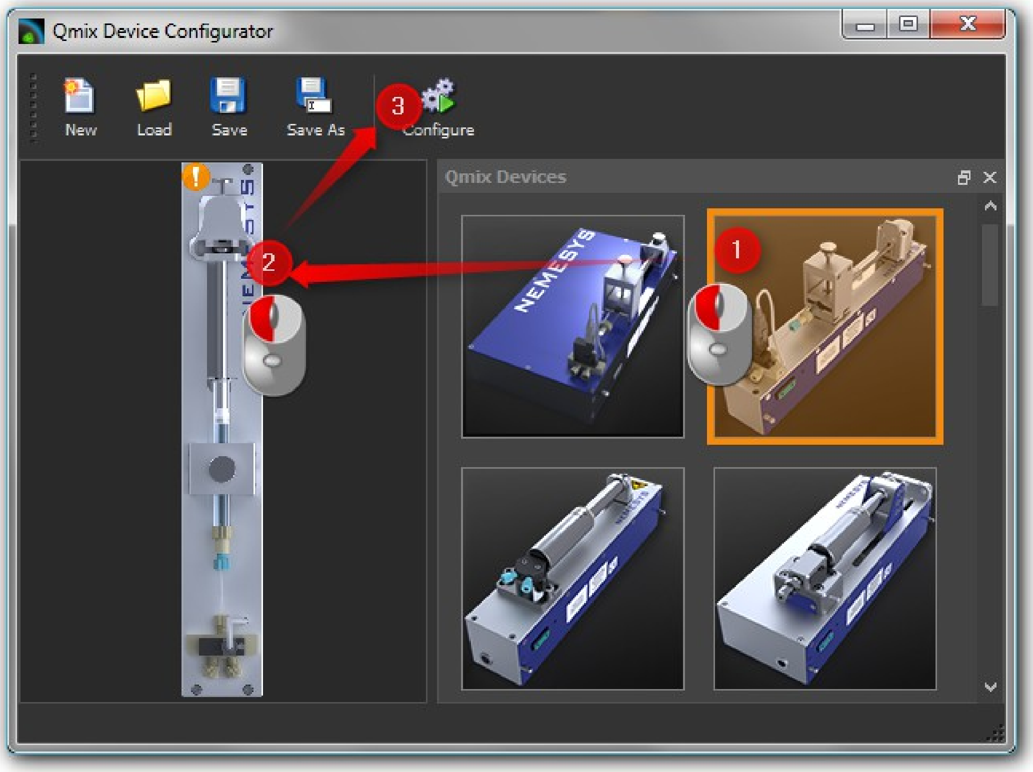


Figure 25: Creating a device configuration

IMPORTANT

Please note that at the beginning of the device configuration process the base module is switched off and no devices are connected to it.

(6) Left-click *Next* in order to start the device configuration process.

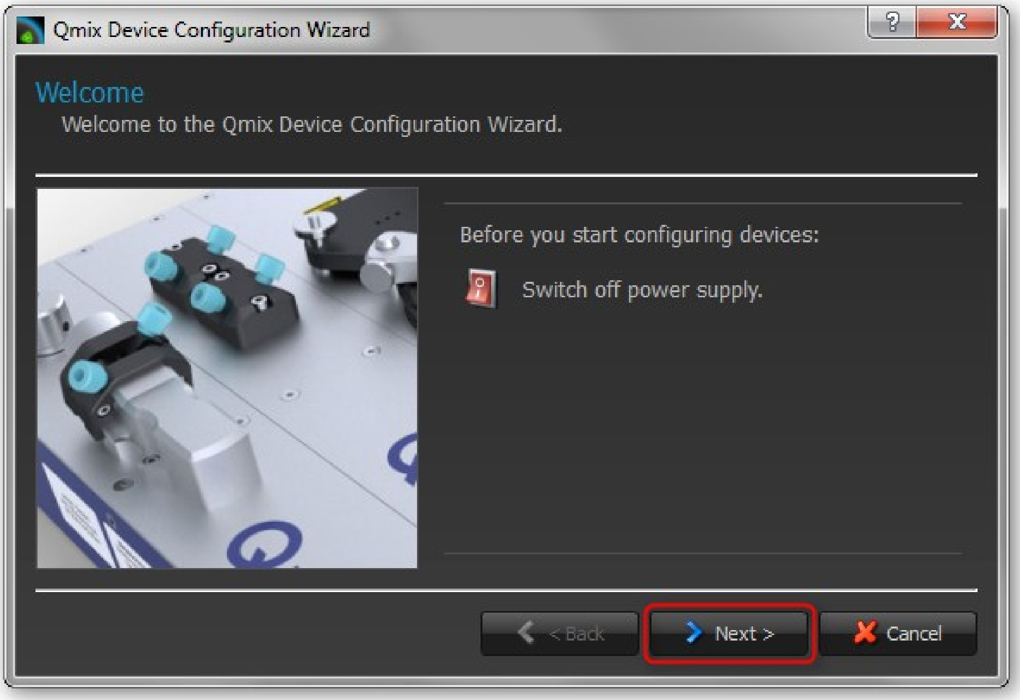


Figure 26: Starting the Device Configuration Wizard

(7) Select the location the device configuration is stored at by left-clicking the button displaying the folder symbol.

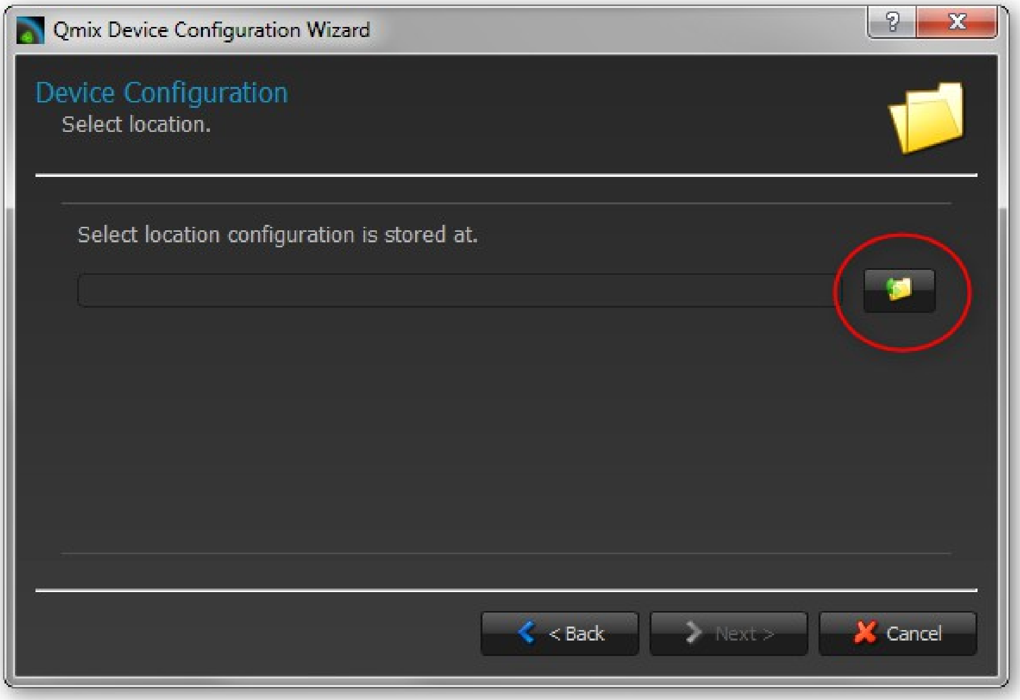


Figure 27: Selecting the device configuration folder

(8) In the folder dialog opening up you can select an existing folder the device configuration is stored at. If the device configuration folder does not exist you can create it by left-clicking *New Folder*.

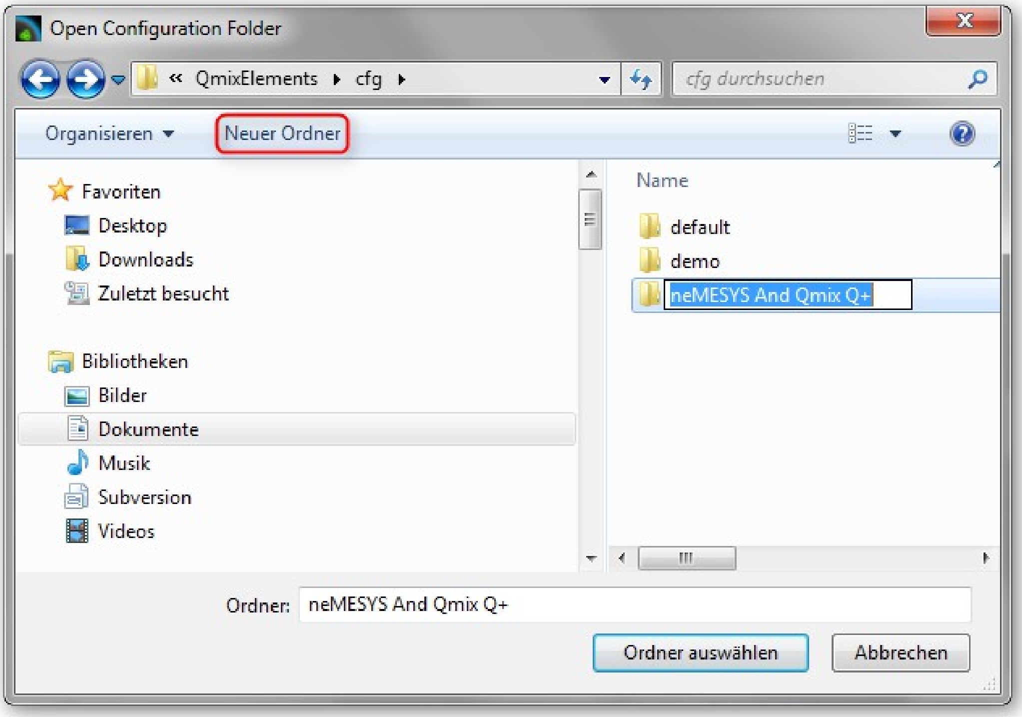


Figure 28: Open Folder Dialog

(9) The Device Configuration Wizard advices you to follow these instructions:

- Plug only the neMESYS dosing module to the base module.
- Plug the terminator.
- Switch on power supply.
- Left-click *Next* to proceed.

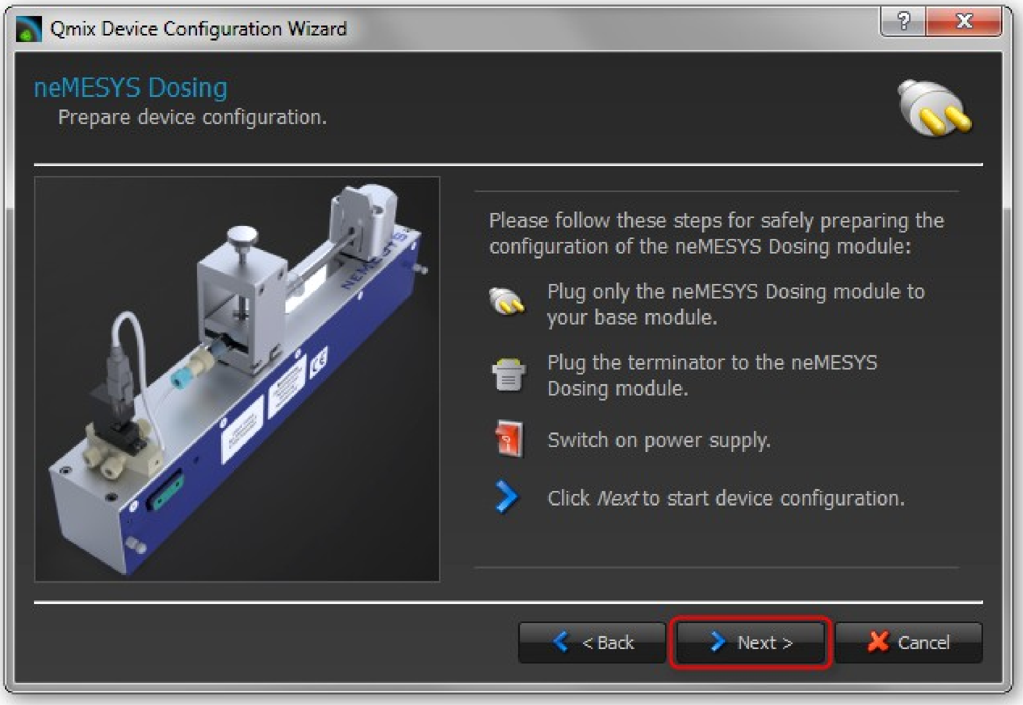


Figure 29: Prepare the device configuration process

(10) The neMESYS dosing module is configured. This may take a few seconds.

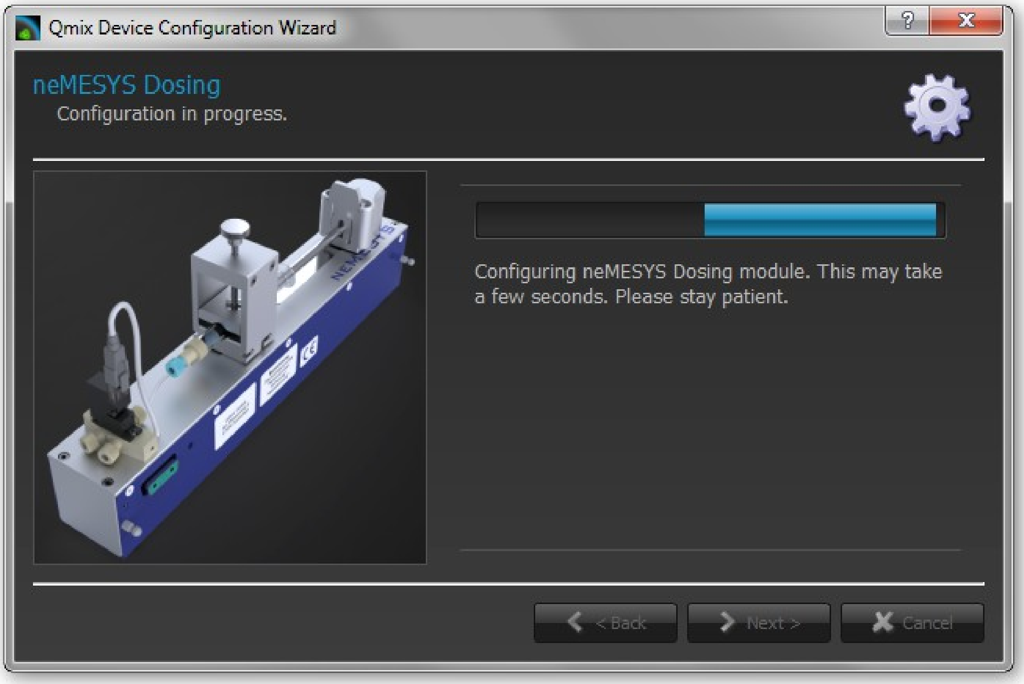


Figure 30: Configuring device

(11) The Device Configuration Wizard displays the neMESYS dosing module configured successfully. Left-click *Next* to proceed.

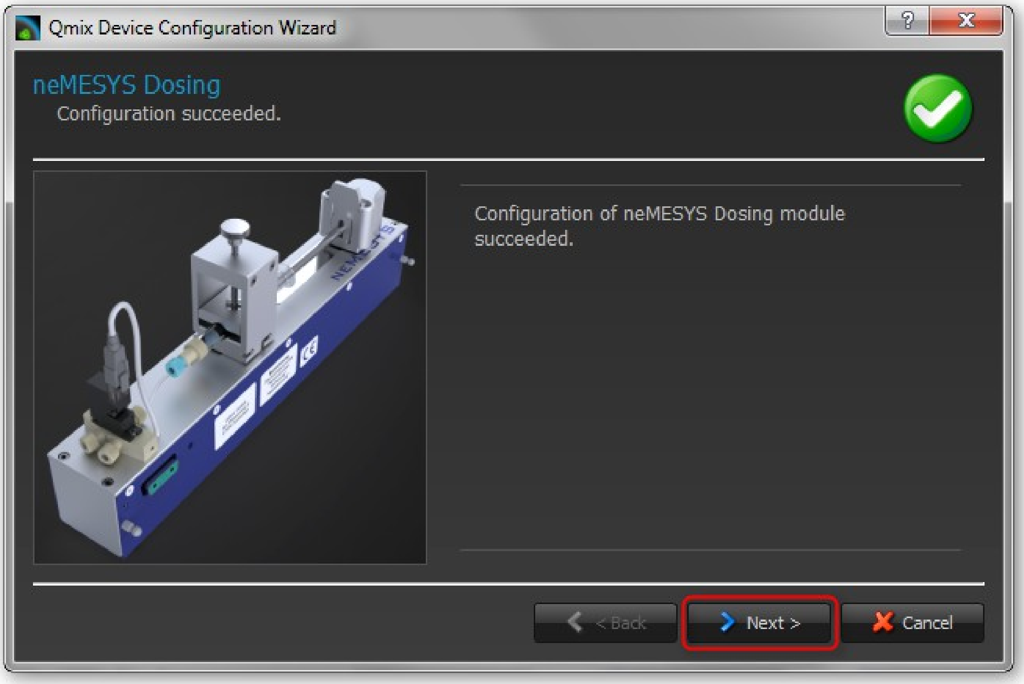


Figure 31: Device configuration successful

(12) The Device Configuration Wizard advices you to follow these instructions:

- Switch off power supply.
- Unplug the neMESYS dosing module from the base module.
- Left-click *Next* to proceed.

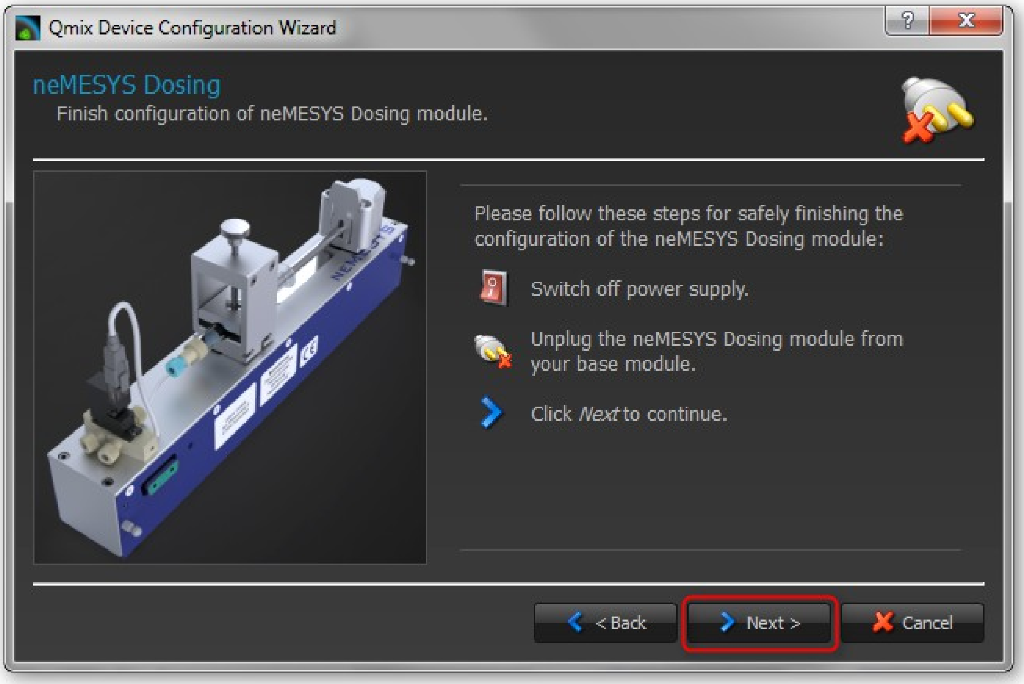


Figure 32: Unplug configured device

(13) Apply steps (9) to (11) to the Qmix Q+ module.

(14) On the next page check *Import and activate new device configuration into QmixElements now*. Doing so QmixElements activates the new device configuration and restarts after finishing the Device Configuration Wizard.

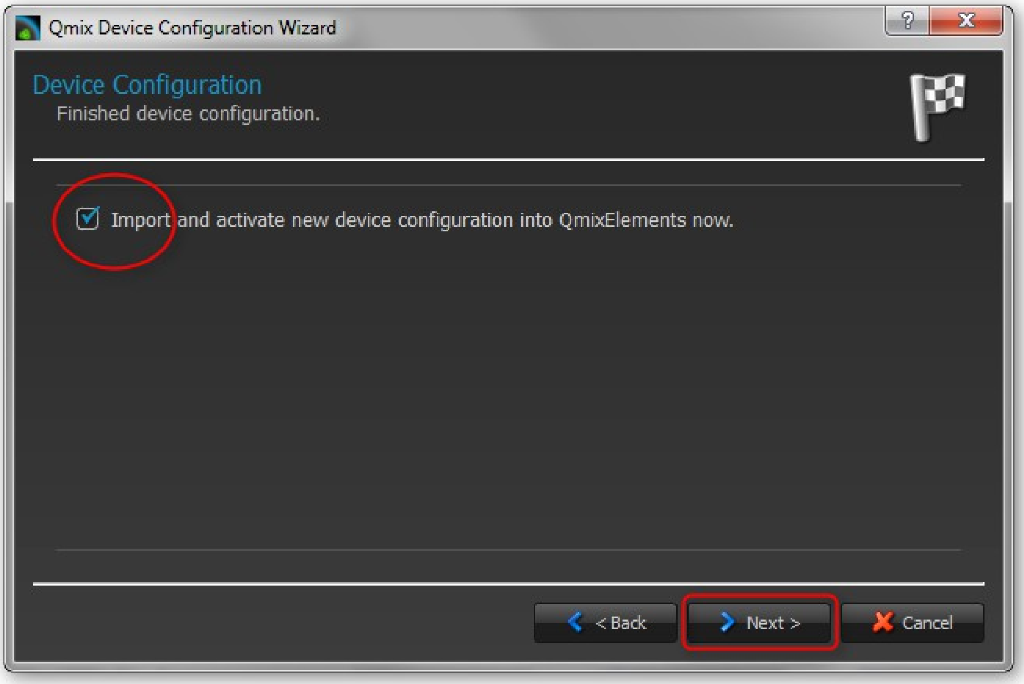


Figure 33: Activating device configuration

(15) Prepare the restart of QmixElements with the created device configuration:

- Switch off the base module.
- Plug the neMESYS dosing module and the Qmix Q+ module to the base module.
- Plug the terminator.
- Switch on the base module again.
- Finish the Device Configuration Wizard by left-clicking *Finish*.

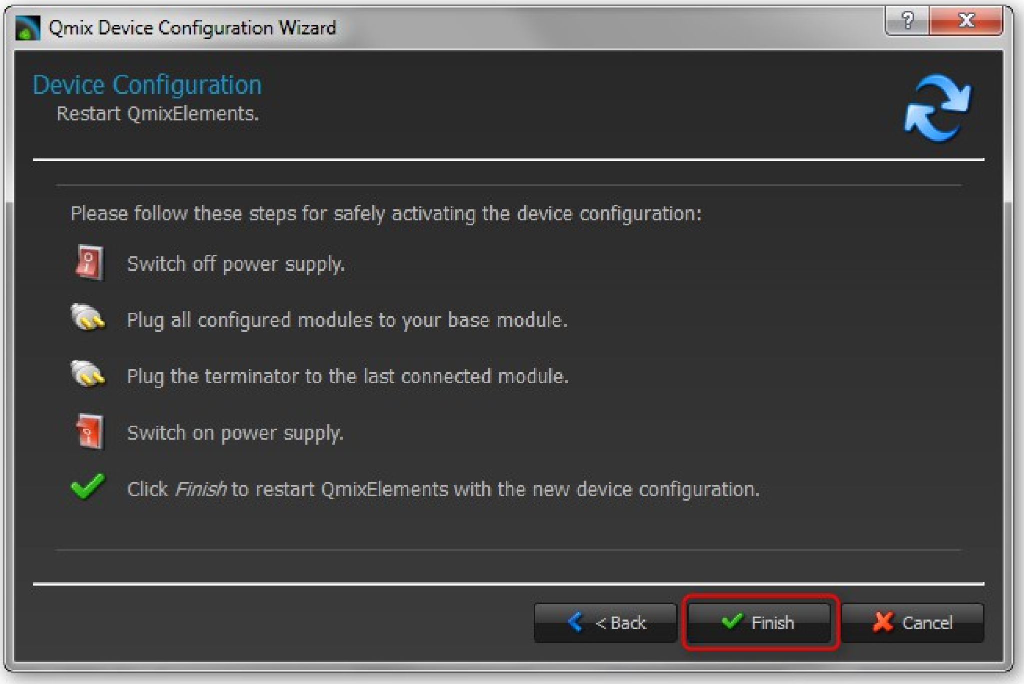


Figure 34: Finishing the Device Configuration Wizard

(16) Confirm the software restart with the new device configuration.

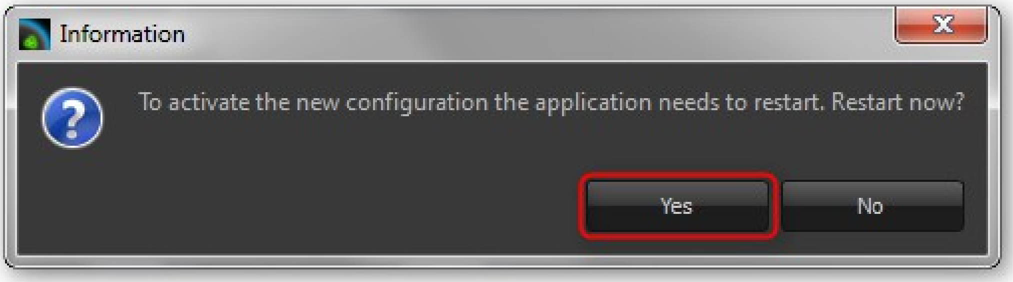


Figure 35: Confirming restart of QmixElements

(17) QmixElements now offers the functions of the neMESYS dosing module and the Qmix Q+ module.

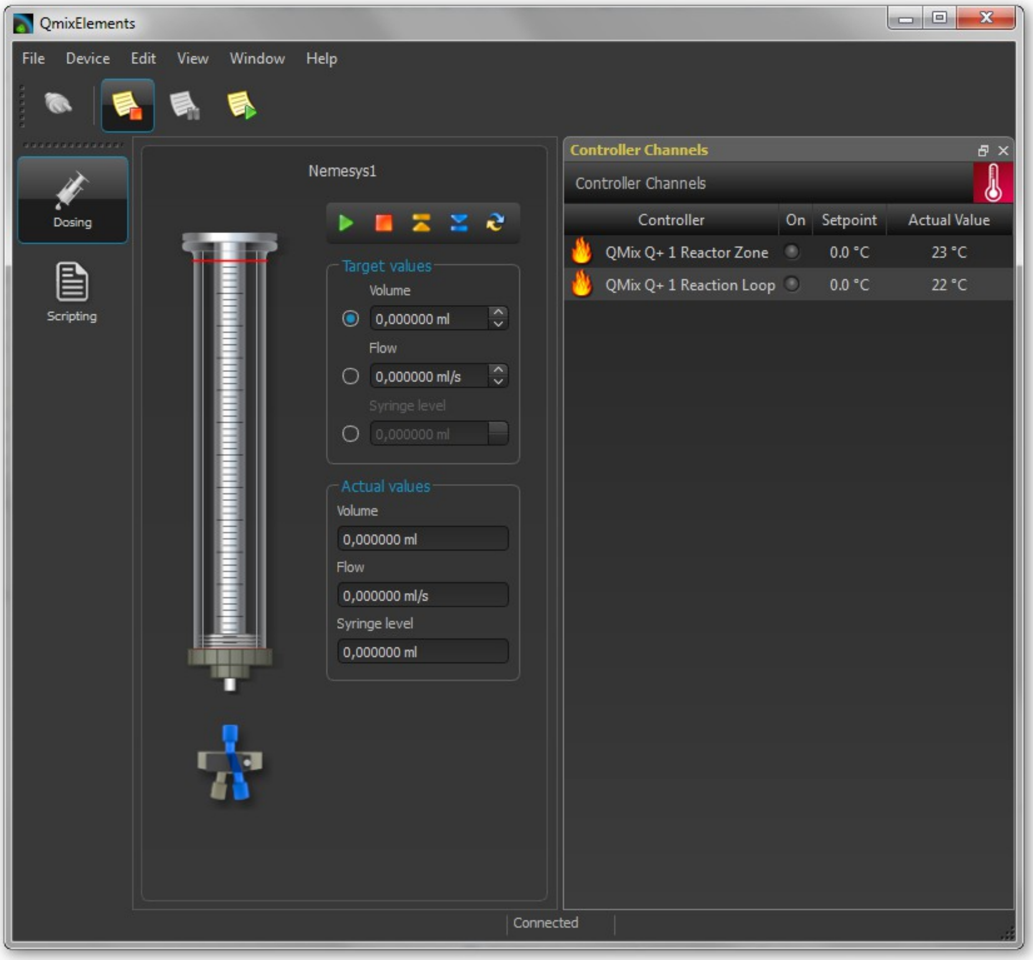


Figure 36: Activated device configuration

5.9 Extending an existing device configuration

The following shows how you can extend the device configuration from the last section consisting of a neMESYS dosing module and a Qmix Q+ module to a further device.

(1) Start the Device Configurator as explained in section 5.2.

(2) After start of the Device Configurator the device configuration currently loaded by the QmixElements software is shown. The green checks displayed on top of the devices indicate that these devices have already been configured.

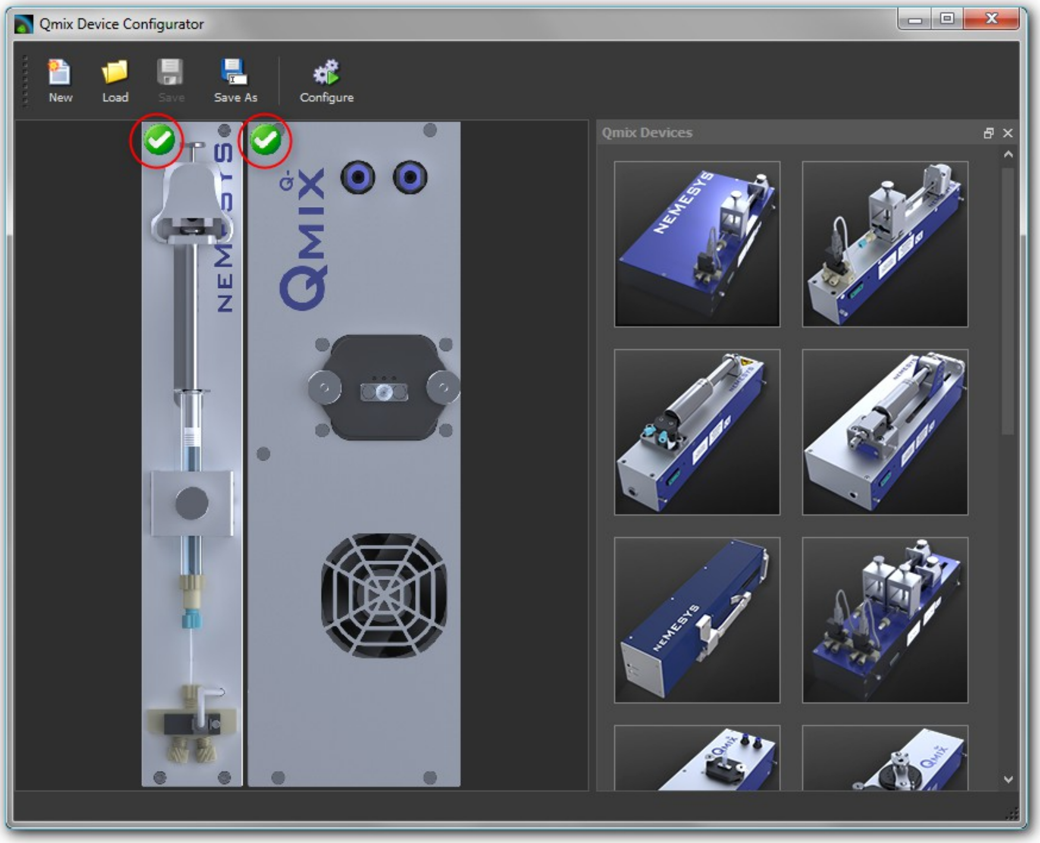


Figure 37: Displaying the currently loaded device configuration

(3) If you want to change another device configuration as the one currently loaded by QmixElements, you have to load the desired device configuration using the *Load* button.

Figure 38: Loading a device configuration

(4) Add a further device as described in the previous section. The exclamation mark above the device points out that it has not been configured yet. The configuration can't be used in this state.

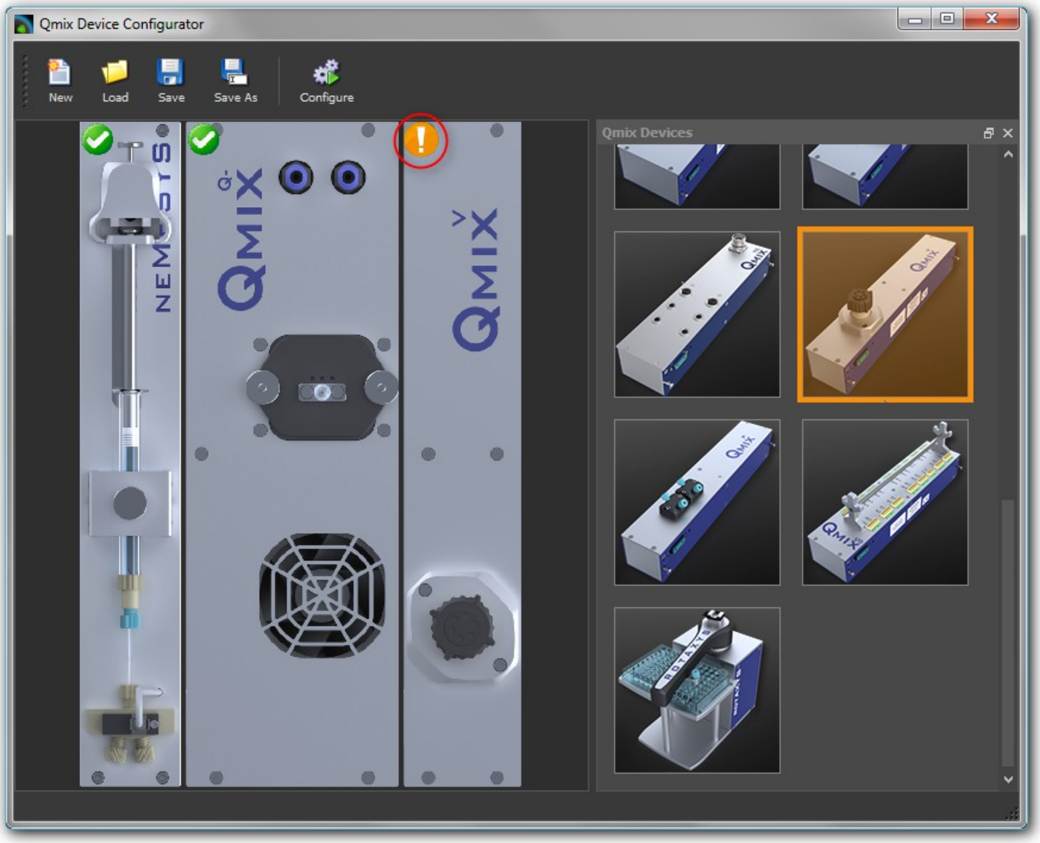


Figure 39: Adding a device to an exisiting device configuration

IMPORTANT

If a device configuration contains not configured devices, it is invalid and can't be used yet.

(5) Start the Device Configuration Wizard by clicking the *Configure* button.

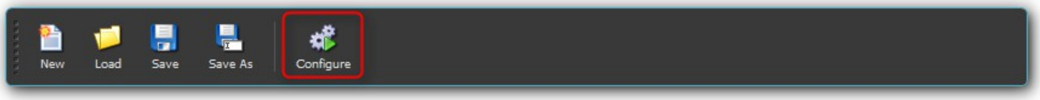


Figure 40: Starting the Device Configuration Wizard

(6) If the software has already been connected to the plugged devices, the device configuration process can't be continued. In this case the Device Configuration Wizard proposes to automatically restart the software and the Device Configurator. Click the *Yes* button if you agree.

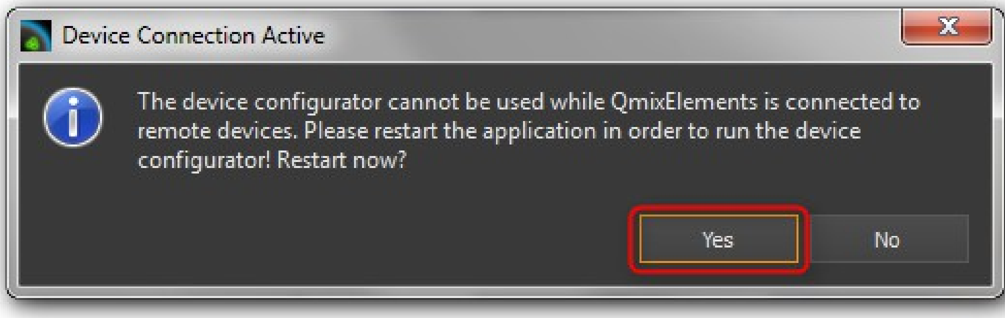


Figure 41: Restart the Device Configurator

(7) The device configuration process known from the previous section starts. But this time the wizard proposes on the second page to exclude devices from the device configuration process that have already been configured. Just check *Skip configured devices*.

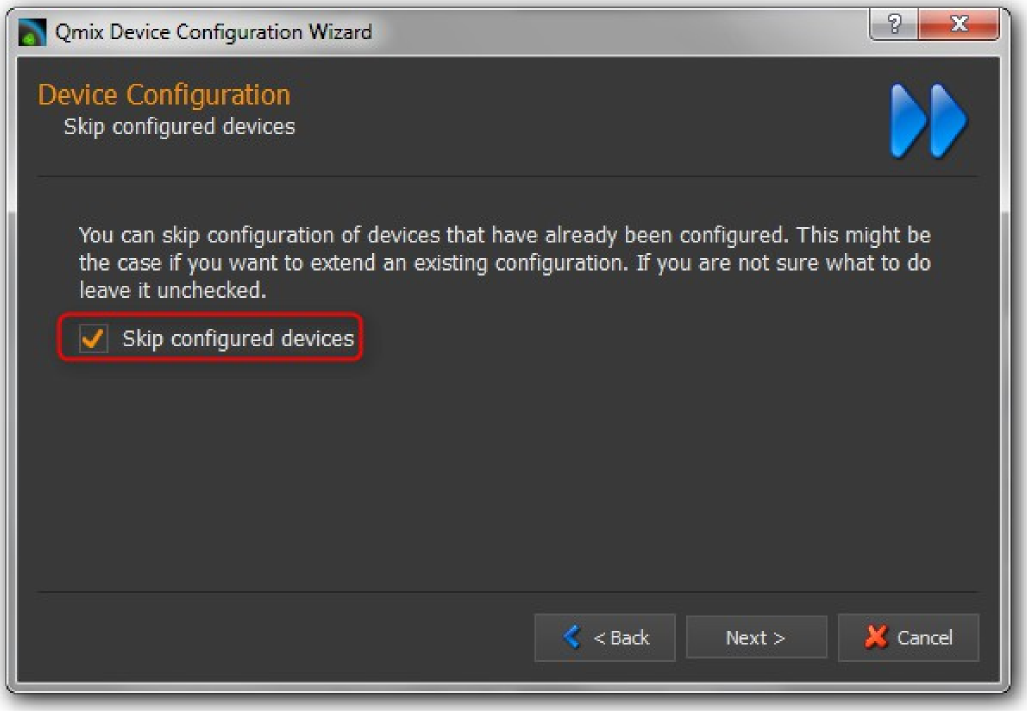


Figure 42: Skip configured devices

(8) The Device Configuration Wizard directly continues with the configuration of the newly added device (Qmix V in this example).

(9) Just finish the Device Configuration Wizard the same way you would have done if you had created a device configuration from scratch. (see section 5.8)

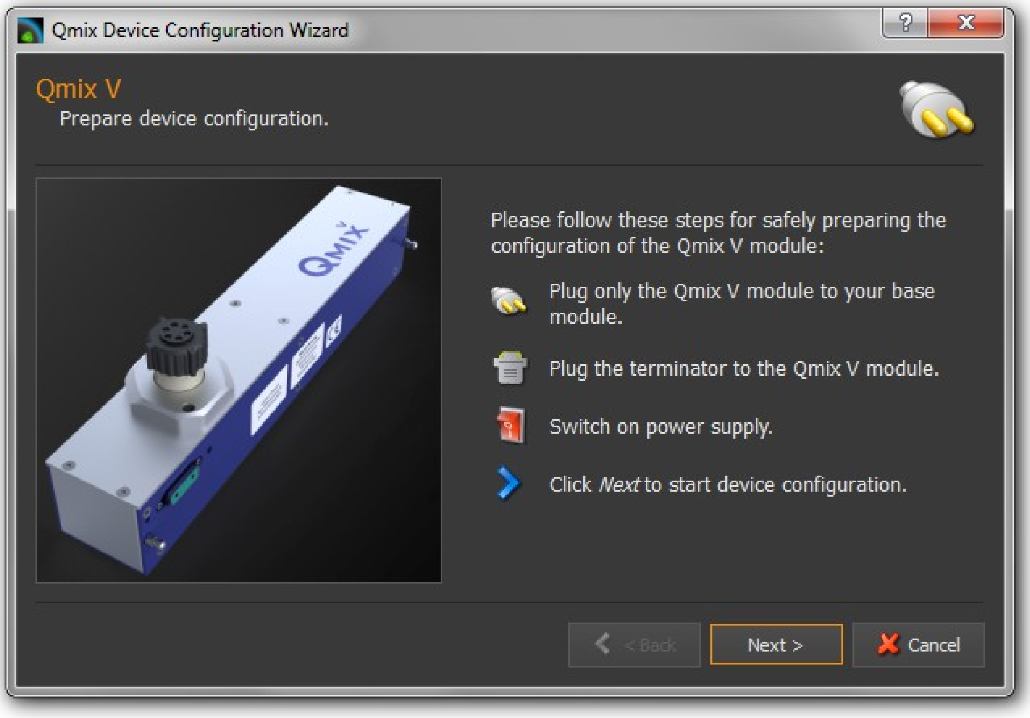


Figure 43: Configuration of the newly added device

(10) After restart the functionality of the new device is available to you.

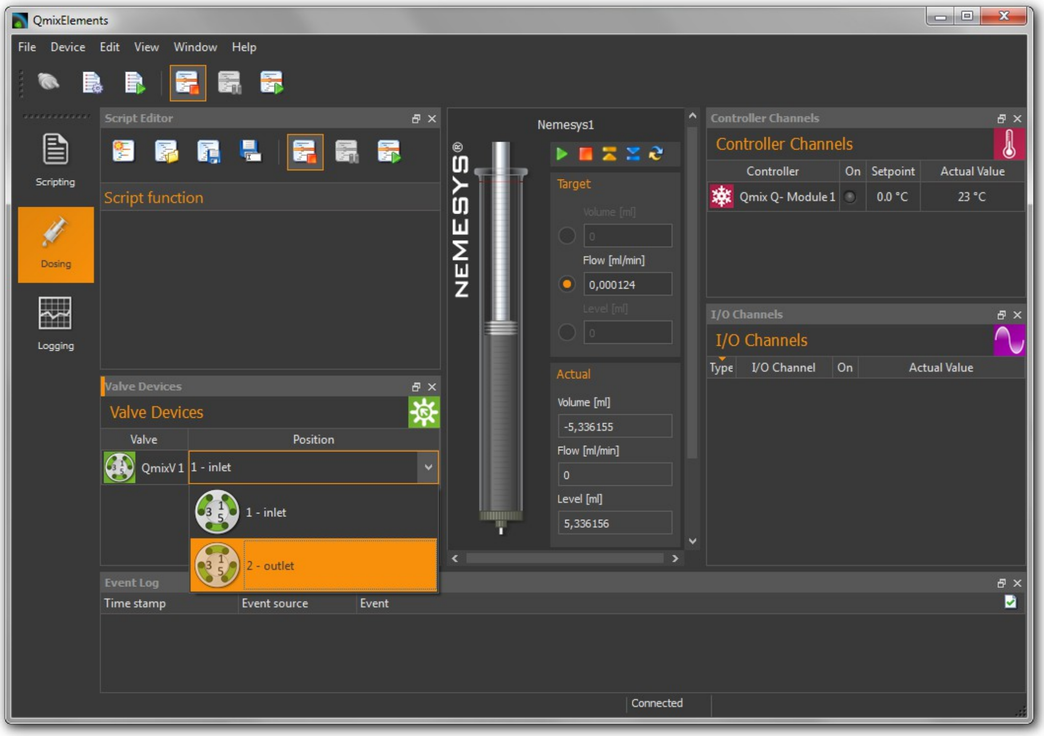


Figure 44: QmixElements with functionality of new device

5.10 Exchanging a device

You can use the configure single device functionality of the Device Configurator to exchange a device of an existing device configuration. This might be useful if you want to exchange a contaminated device against a cleaned device for example. For this purpose, proceed as follows:

(1) Start the Device Configurator as described in section 5.2.

(2) If you want to exchange a device of another device configuration as the one currently loaded by QmixElements, you have to load the desired device configuration using the *Load* button.

Abbildung 5.1: Laden einer Gerätekonfiguration

(3) Right-click the device you are going to exchange and select *Configure*.

Abbildung 5.2: Einzelkonfigurationsfunktion

(4) The Device Configuration Wizard starts. But this time only the selected device is configured.

(5) Plug the replacement device when you are asked to do so.

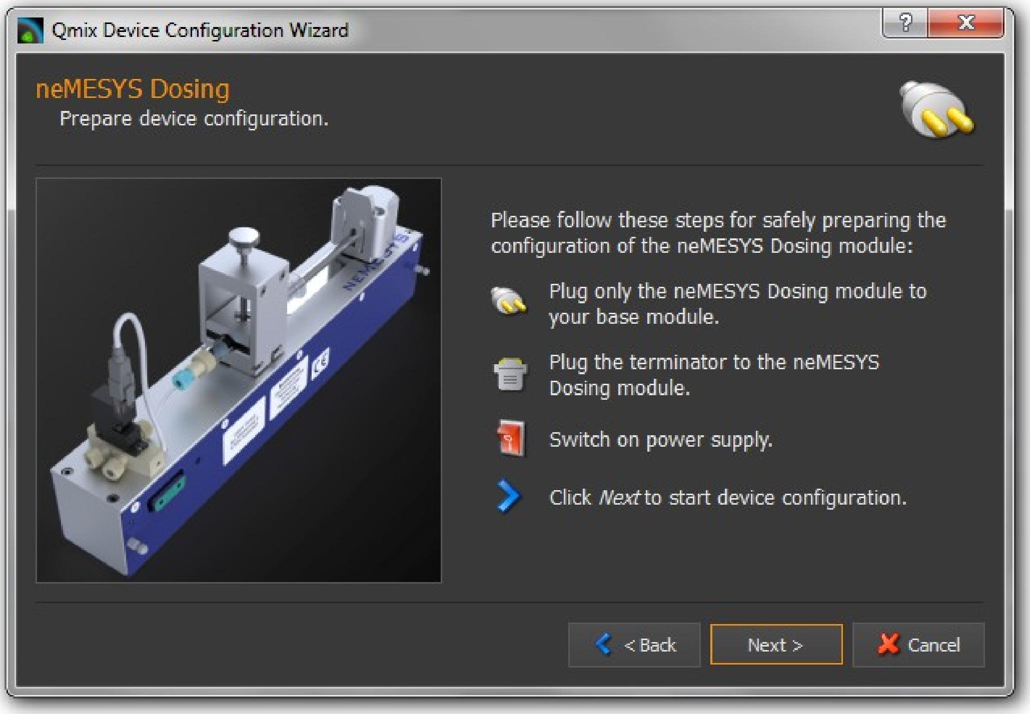


Abbildung 5.3: Konfiguration des Austauschgerätes

(6) Just finish the Device Configuration Wizard the same way you would have done if you had created a device configuration from scratch. (see section 5.8)

(7) Now you can use the replacement device for your device configuration.

5.11 Simulated devices

You can set single devices or a whole device configuration to simulated. This is useful if you want, as an example, create QmixElements scripts using devices not available at the moment. The QmixElements demo configuration for example completely consists of simulated devices. Proceed as follows in order to simulate a single device:

(1) In the device configuration right-click the device you want to simulate.

(2) Left-click the button *Simulate*

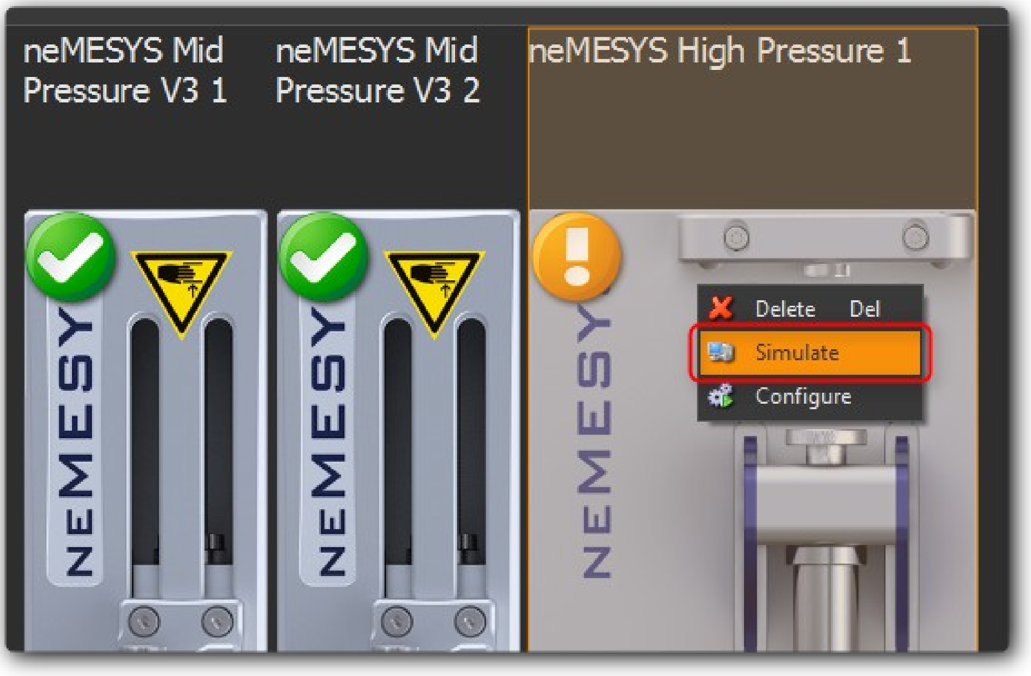


Figure 45: Simulating a single device

(3) The device is marked as simulated. Save the device configuration. The next time you load this device configuration in QmixElements the simulated device will be available to you.

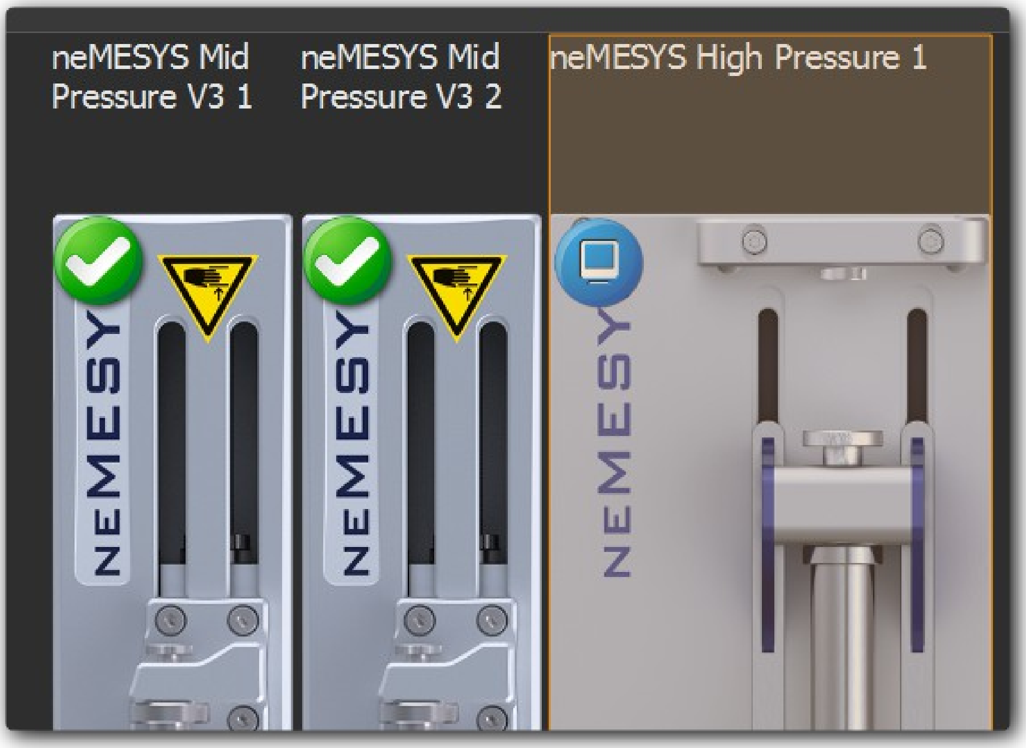


Figure 46: Simulated device

You can simulate a whole device configuration by left-clicking *Simulate All* in the main toolbar.

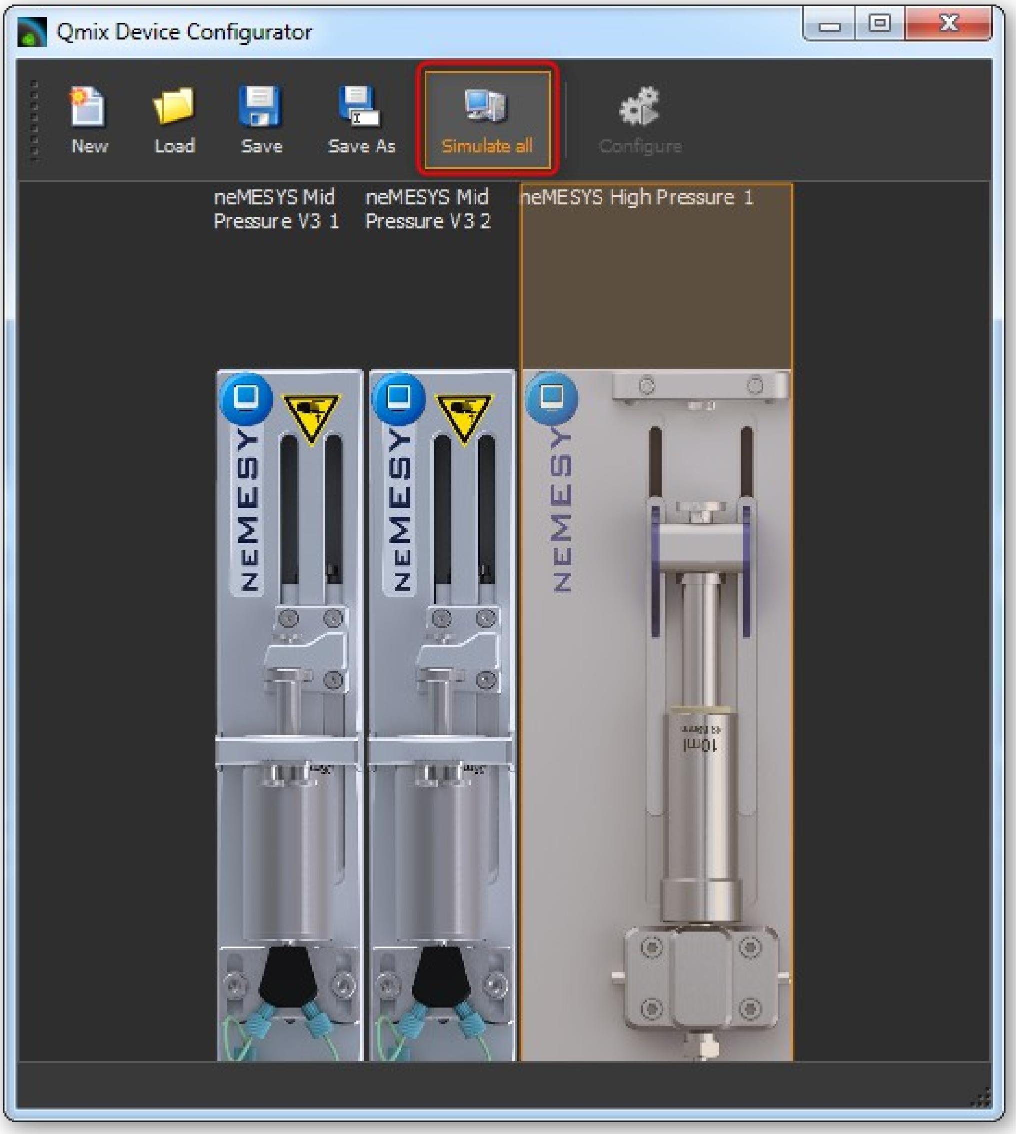


Figure 47: Simulating a whole device configuration

6 Script System

6.1 Introduction

The software provides a powerful scripting system to set up automated process sequences.

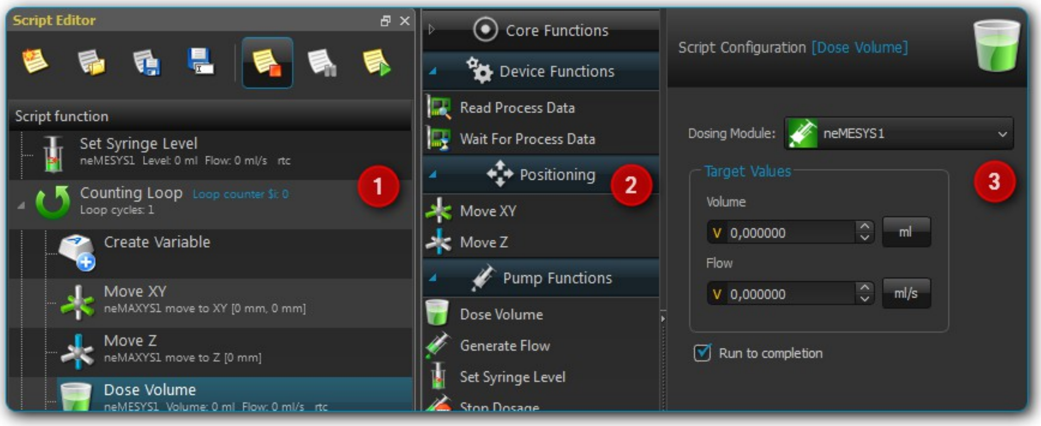


Figure 48: Script system overview

The script system work bench consists of the following three main parts:

- 1 *Script Editor* – shows the script programmed by the user as a function tree. It also features buttons for controlling script files and their execution.

- 2 *Script Pool* – contains all available script functions ordered in device categories.

- 3 *Script Configuration* – is used to configure the parameters of individual script functions.

6.2 Script Pool

Activate the *Scripting* button in the side bar to show the Script Pool.

Figure 49: Show Script Pool

In addition to the *Script Pool*, the *Script Configuration* area is also shown. The *Script Pool* contains all script functions that are available for programming scripts. The script functions are grouped into categories. In addition to a set of core functions, each device and each plug-in registers its own specific script functions in a separate category.

The user can open or close the categories in the *Script Pool* at any time. To open or close the function list of a category, simply double-click on the category name (figure below).

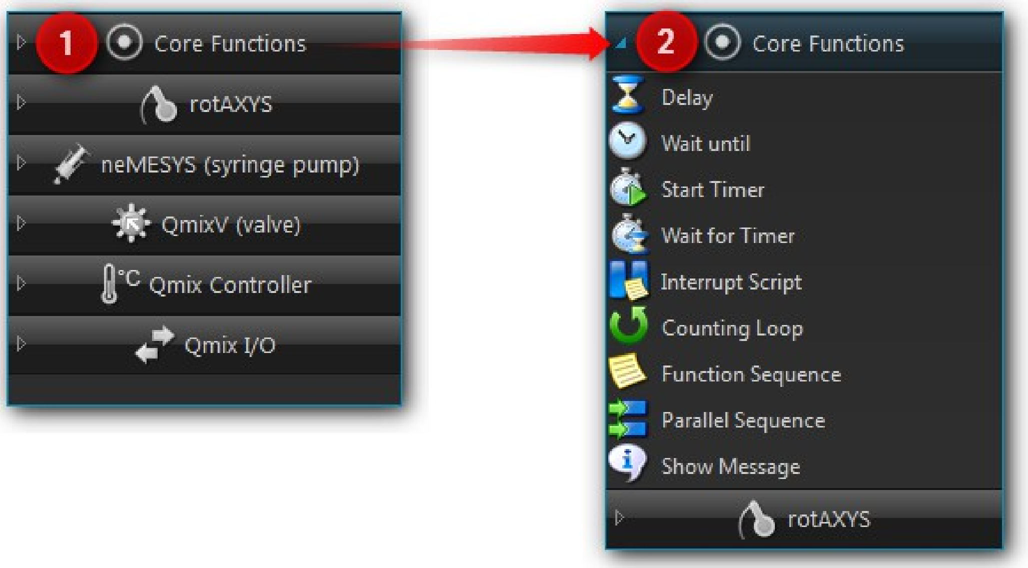


Figure 50: Script Pool – Expanded view (2) of individual category functions.

6.3 Script Editor

6.3.1 Introduction

The *Script Editor* is used for the graphic programming of scripts. The following items are numbered in the figure below:

1 *Toolbar* – is for loading, saving, and controlling the sequence of scripts.

2 *Function tree* – shows a treelike program structure.

3 The currently executed function is highlighted in green.

4 When you select a function by clicking, it is highlighted blue.

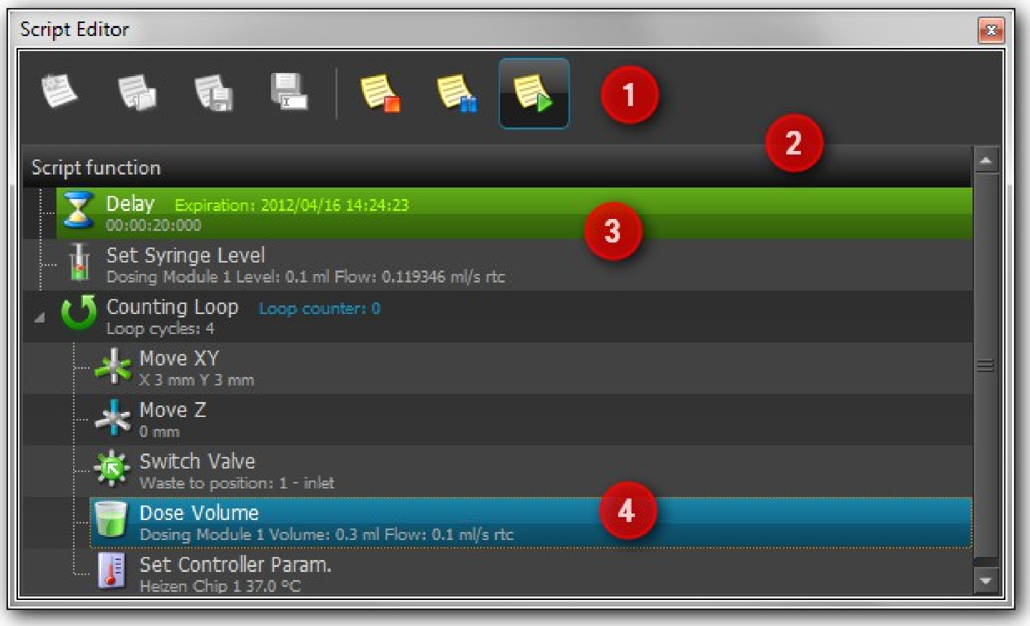


Figure 51: Script editor panel

Each single function is displayed in the function tree in a separate line. In this line all the important function parameters are visible for you (see figure below):

Figure 52: Single function in the Script Editor

On the left side you will find the graphic icon of the function 1. Immediate right of the icon at the top 2 you will find the function name. Status information can be found to the right of the function name 3. For many functions, these status information are visible only during the execution and are subject to change. Directly below the function name is a summary of all important function parameters 4, that you have configured in the configuration area.

The *Script Editor* is a movable and dockable window: you may move and dock the *Editor* to another position within the main software window. To do this, drag-&-drop the window via the title bar to its new location using the computer mouse. If the *Editor* window is not visible, you may first have to activate it via *Window* → *Script Editor* in the main menu (figure below).

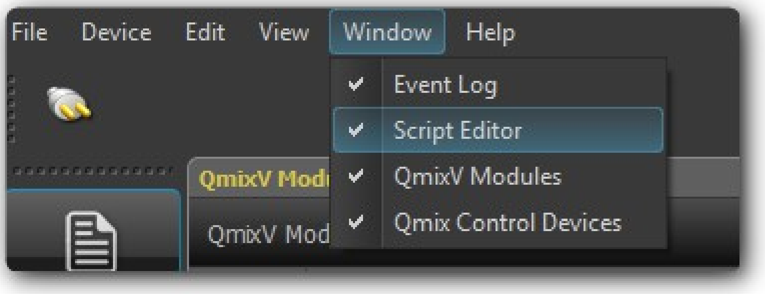


Figure 53: Activating the Script Editor

6.3.2 Toolbar

Generates a new, empty script.

Loads an existing script file into the *Script Editor*.

Saves the currently active script.

Saves the currently active script into a new script file.

Stops the execution of the current script. By clicking the start button, the complete program will be restarted from the beginning.

Pauses the execution of the current script. By clicking the start button, the execution will resume from its current position.

Starts the execution of a script or resumes a script after an interruption.

HINT

You can also load script files easily via drag & drop. Simply drag a script file from your file system over the script editor and drop it there.

6.3.3 Context Menu

If you click with the right mouse button on any function within the *Script Editor*, a context menu appears to quickly access additional actions (figure below).

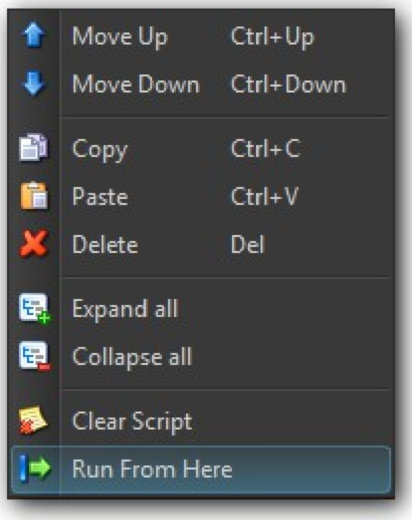


Figure 54: Script Editor context menu

TIP

To start script execution at a specific function from within a script, click the right mouse button on the function of choice and select ***Run From Here*** *from the context menu.*

6.4 Script Configuration Area

6.4.1 Overview

The configuration area contains all controls for configuring the script function that is currently selected in the *Script Editor.*

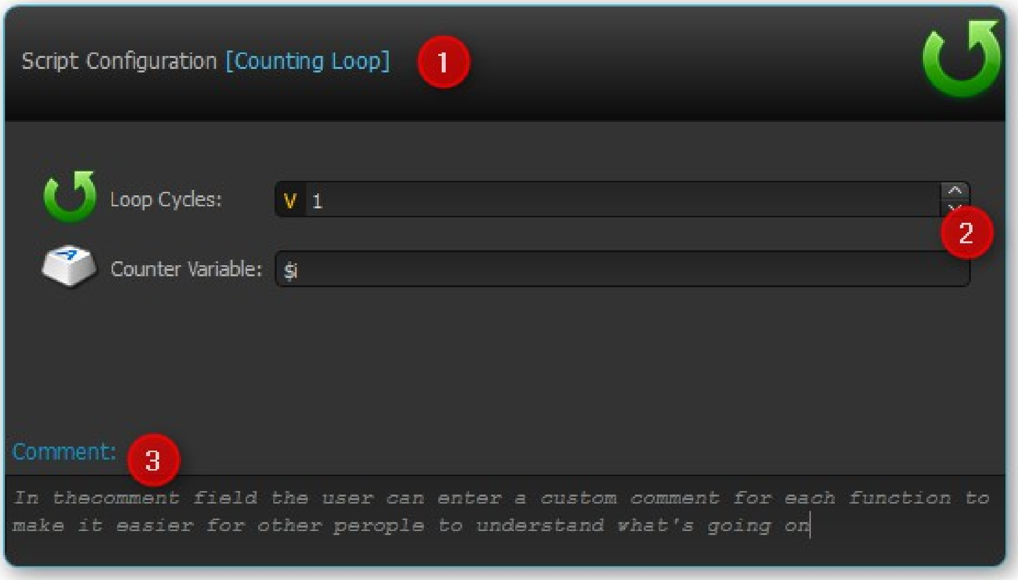


Figure 55: Script Configuration Area

The configuration area consists of:

1 header with the name of the currently selected function

2 input- and control elements of the function

3 comment box to enter any commentary

The input- and control elements 2 are different for each script function. But all functions provide some common controls like the function caption 1 in the header and a comment box 3 at the bottom of the configuration area.

6.4.2 Changing Function Caption

In the header of the configuration area you can change the caption of the function. It allows you to use "talking" function names that make it a lot easier for you or third parties to read and understand your scripts later.

To change the function name, you can either click with the left mouse button on the function name in the header or you can click on the name with the right mouse button and select the context menu item, *Edit Function Caption* (see figure below).

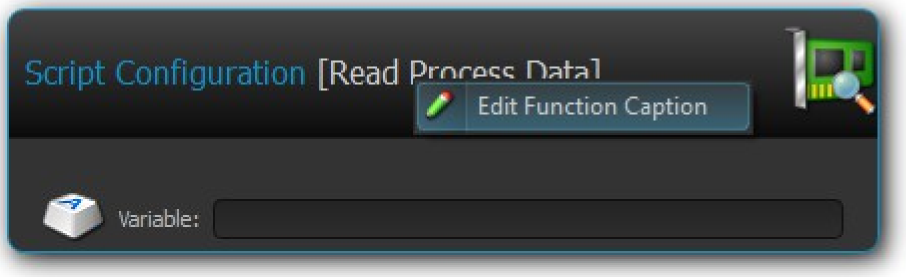


Figure 56: Showing Caption input dialog

In the input dialog that now appears, you can enter a new name for the function.

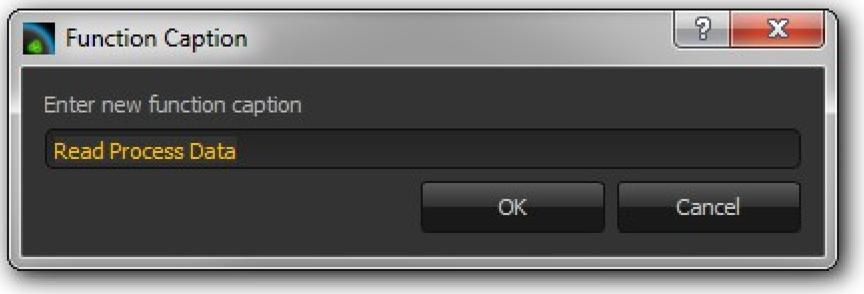
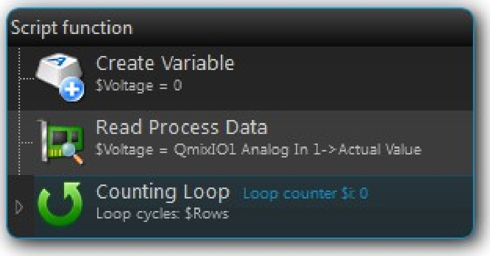


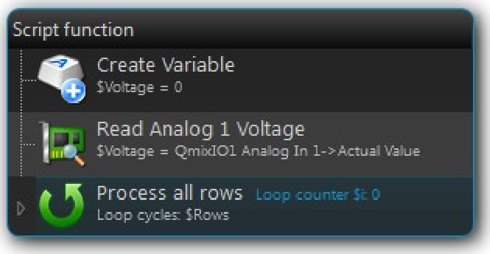
Figure 57: Function Caption input dialog

The following example shows a short program with the default function names on the left side and the same program with own function names on the right side:

Default function names



Application-specific names



HINT

Enhance the readability, understandability and maintainability of your scripts through the use of speaking, application-specific function names.

6.4.3 Enter Comment

In the comment field you can enter a comment that will allow other users to understand your scripts better and to follow the flow of execution easier.

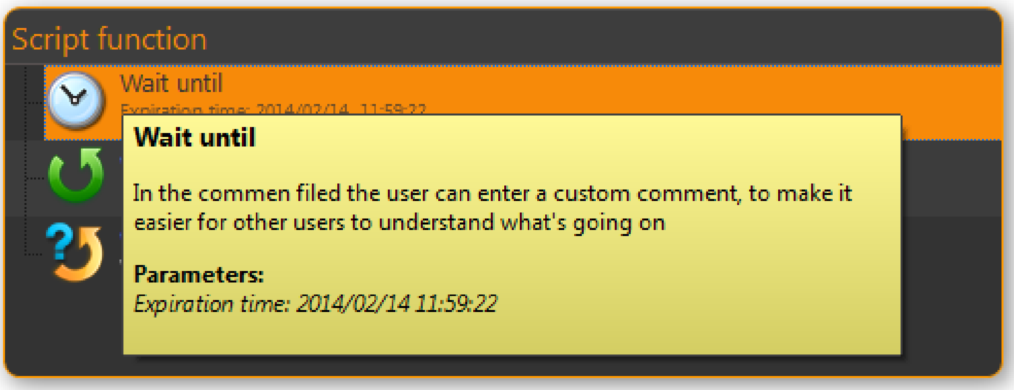


Figure 58: Script function comment

When you move the mouse pointer over a function in the *Script Editor* the comment of this function will be shown in a message box (see figure above). So you can read the comment of a function without having to open the configuration area of that function.

6.5 Programming

6.5.1 Adding Functions

Functions are activated via drag-&-drop from the *Script Pool* to the *Script Editor*. To do this, proceed as follows:

(1) In the Script Pool, left-click on the function that you want to insert 1 and hold down the mouse button.

(2) Move the pointer to the desired position within the *Script Editor* window.

(3) As soon as you release the mouse button 2, the selected function will be inserted into the *Script Editor*.

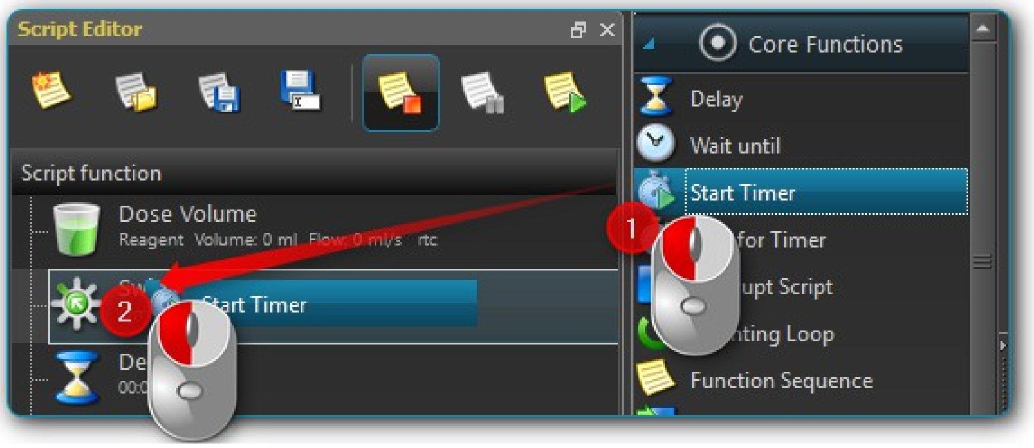
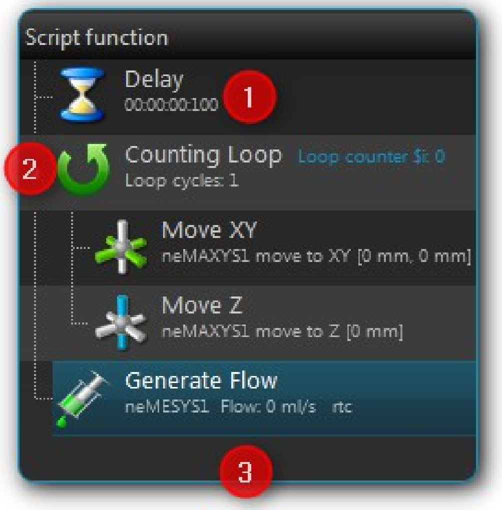


Figure 59: Inserting a function into a script via drag-&-drop

The function is inserted according to where the mouse pointer is positioned when you release the mouse button. The following scenarios are possible (figure below):

1 If you release the mouse button atop an existing function, the new function will be inserted immediately before that existing function.

2 If the mouse button is released atop a function sequence (for example, a loop), the new function will be inserted at the end of that sequence.



3 If the mouse button is released on an empty area at the end of the function tree, the function is added at the end.

6.5.2 Moving Functions

Analogous to inserting a new function from the *Script Pool*, you can move the functions to new positions within the function tree via drag-&-drop. Again, the same insertion rules apply as above.

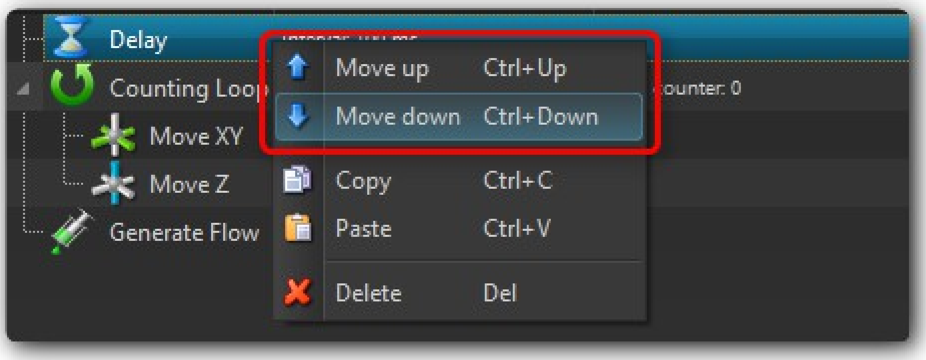


Figure 60: Moving functions within a function tree

There is alternative way for moving functions up and down the list: First, make a right mouse click on the function that is to be moved. This will open a context menu from which you may then select *Move up* or *Move down*, respectively (figure above). Alternatively, you can use the keyboard with the *Ctrl+Cursor up* or

Ctrl+Cursor down shortcuts.

This latter method may only be used to move functions up or down within the current sequence. If you want to move a function to a completely different position within the function tree, this can only be done via drag-&-drop.

6.5.3 Deleting Functions

There are two ways to delete functions:

1. Make a right mouse click on the function that is to be deleted and then select *Delete* from the context menu.

2. Select the function to be deleted by left-clicking it and then press the *Delete* key of your keyboard.

6.5.4 Copying Functions

Similarly, functions can be copied either by using the context menu via the mouse or using key combinations via the keyboard. If you work with the context menu, simply select *Copy* and then *Paste* from the menu (figure below); with the keyboard, use the *Ctrl+C* shortcut to copy and *Ctrl+V* to paste.

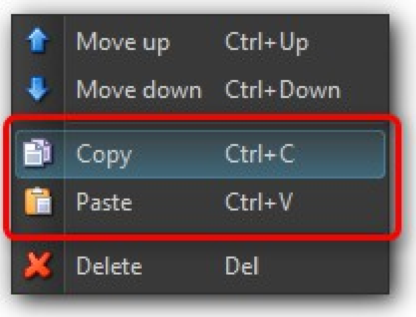


Figure 61: Copying a function

This is how you copy a function to a new position:

1. Select the function you want to move by left-clicking it.

2. Copy the function via *Copy* of the context menu or via the keyboard and *Ctrl+C*.

3. Select the function before which you want to insert the copied function by left-clicking it.

4. Paste the copied function via *Paste* of the context menu or via the keyboard and *Ctrl+V*.

To insert the same function at multiple points of the function tree simply repeat steps 3 and 4 (above).

6.5.5 Editing Function Parameters

As soon as you select a function from the function tree, the operating elements appear in the *Script Configuration* area that allow you to configure the selected function. Edit the function parameters as required.

6.5.6 Showing Function Tooltip

If you move the mouse cursor over a function, a tooltip window is displayed for this function after a short time (see figure below). In this window, you will get all information about this function at a glance: function name 1, user comment or function description 2 and function parameters 3.

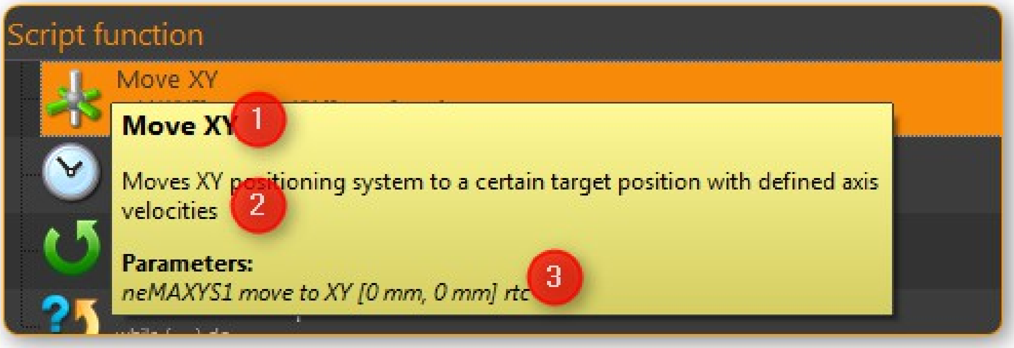


Figure 62: Showing script function tooltip

6.6 Variables

6.6.1 Introduction

The script system of supports the use of variables. Within script programs, variables serve as containers for calculated and device values. Their values are generated during program execution from, for example, loop counts or sensor data. Each variable is defined by a name. Script functions that support the use of variables may then use the values stored in these containers, *e.g.*, to trigger value-dependent events.

6.6.2 Setting Variables

Before variables may be used by a script, such variables need to be defined. There are two possibilities to setup variables:

1. Explicit: Variables are defined explicitly by the user, *e.g.*, via the function *Create Variable* (see section 6.8)

2. Implicit: Implicit variables are created via functions that offer variables by default, such as the counter of the *Counting Loop* function.

6.6.3 Naming Variables

There are some important requirements to keep in mind when naming variables: Every variable is called within a program script via an essentially freely definable name. This name serves to unequivocally identify that variable; different names signify different variables. The $-prefix clearly identifies a name and its use as a variable. The scripting system is case sensitive: *$Var* is different variable than *$var*.

Additionally, the following rules apply when naming variables:

- Variable names have to start with the Dollar symbol ($) and must not themselves contain a $-symbol.
- Only alphanumeric characters are allowed (a-Z, 0-9).
- Special characters (such as, $, &, /, -, ...) are not allowed.
- Variable names must not start with a number.

HINT

You can display the contents of variables using the Show Message function, e.g. to check the results of calculations.

6.6.4 Visibility Range of Variables (Scope)

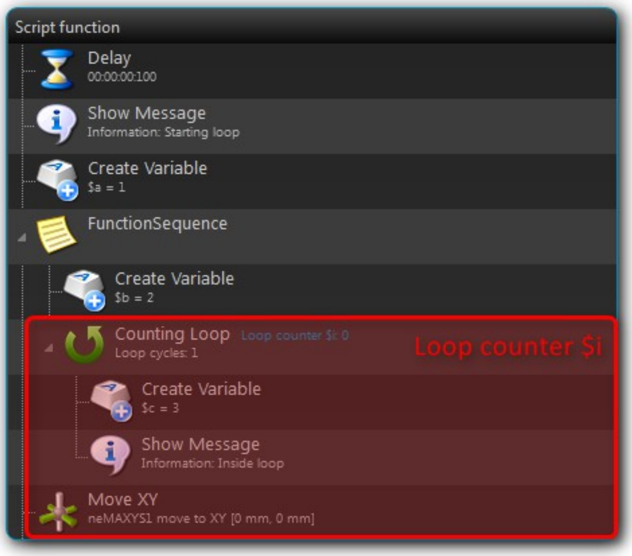
The visibility range (scope) of a variable is that part of the program within which that variable is visible, *i.e.*, available. Qmix scripts are trees with an essentially unlimited number of branches and levels; a variable, *i.e.*, the value it returns, is only visible, *i.e.*, available to be used, at that level at which it has been defined plus all its sub-levels.

The following example illustrates the scope of variables:

The scope of variable *$a* is highlighted in red: It is available throughout the entire script:

Variable *$b*, on the other hand, is only visible within a specific function sequence.

Further down still within the function tree, the counter variable *$i* of the counting loop is only available to those functions that are added to within that loop:



The scope of variable *$c*, which has been setup within the counting loop, is only available within that counting loop, too, as no other sublevel has been added at this point:

It is important to note that, if two variables have the same name, the variable that has been defined later (*i.e.*, at a lower level) will overwrite the variable defined earlier (*i.e.*, at a higher level). In the example above, if *$c* would have been named again as *$b*, the later-defined value (*i.e.*, 2) would replace the earlier-defined value (*i.e.*, 1).

IMPORTANT

If a lower-level variable has the same name as a higher-level variable, the lower-level variable will supersede the higher-level variable. That is, functions at the lower level cannot access the value of the higher-level variable of the same name and will use „newer“ value instead.

6.6.5 Using Variables

Variables can be used with all functions that support them. Calling a variable to, *e.g.*, set or calculate a value, requires the use of the dollar symbol ($) as a prefix: To use (call write to) the variable *a*, the required syntax is: *$a*.

IMPORTANT

*Variables get assigned a valid value only after they have been assigned a value via being run through a relevant function (*e.g., Create Variable*). If you are using the action* ***Run From Here*** *to start a script, variables may not have been assigned a valid value yet if the respective assignment function follows later in the sequence or has been skipped.*

Functions that support the use of variables have the relevant input boxes highlighted by a yellow “**V**” (see figure below). Here you can insert the name of a variable instead of a numeric value that is to be used subsequently within the relevant section of the program script.

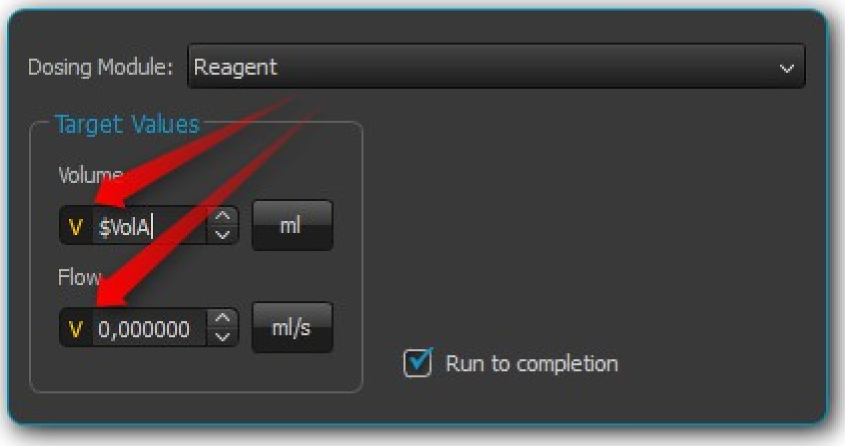


Figure 63: A yellow „V“ signifies a function that supports variables

HINT

Nearly all input fields that support variables allow for direct access of device process data via device property identifiers. (see section 6.7)

6.6.6 Auto-Completion of Variable Names

Input boxes that support the use of variables feature auto-completion to aid the selection and input of available variable names: Upon inserting the $-symbol, a list will appear that contains all variable names defined so far (see figure above).

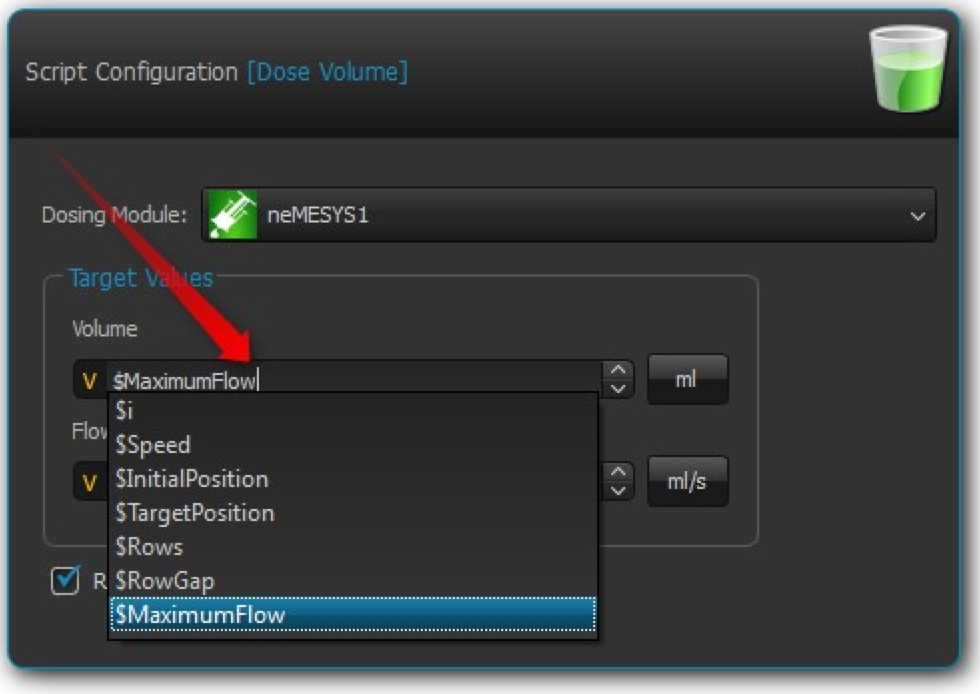


Figure 64: Auto-completion of variable names

Every additional character that you enter will cause a filtering of that list according to the character sequence inserted thus far. You may use either the *up* or *down buttons* of your keyboard or the mouse to select a name from that pre-filtered list. Accept the selection by pressing the *Enter* key.

6.7 Device Property Identifiers

Nearly all input fields that support variables (see section 6.6.5) allow for direct access of device process data via device property identifiers. Just click with the right mouse button in the input field and select the menu item *Insert device property* (see figure below).

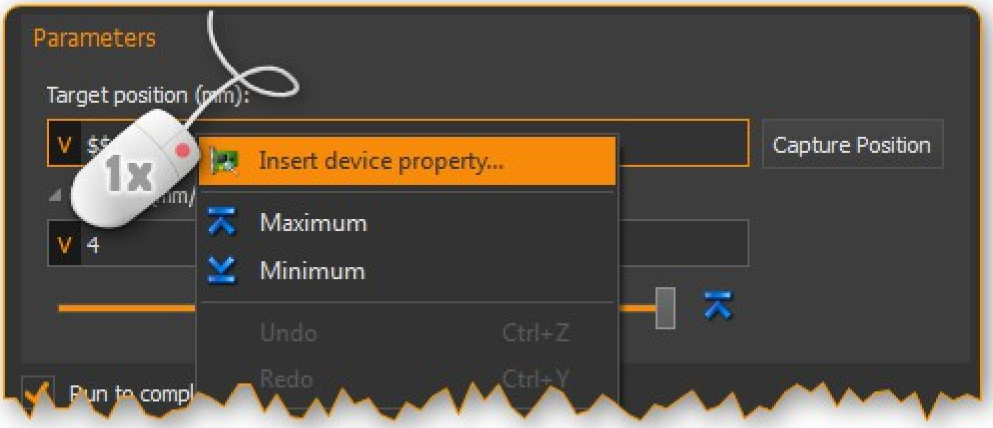


Figure 65: Inserting device properties into input fields

A dialog for selecting the process data is displayed (see figure below).

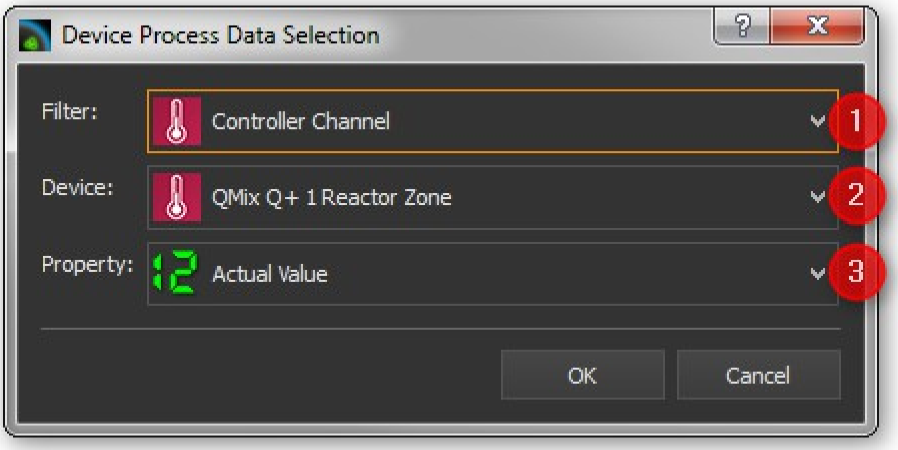


Figure 66: Selection dialog for device process data

In this dialog you can select which type of device you want to access in the selection box *Filter* 1. Select a specific device in the selection box *Device* 2 which contains the filtered list of devices. Finally select the process data to be accessed in the *Property* 3 field.

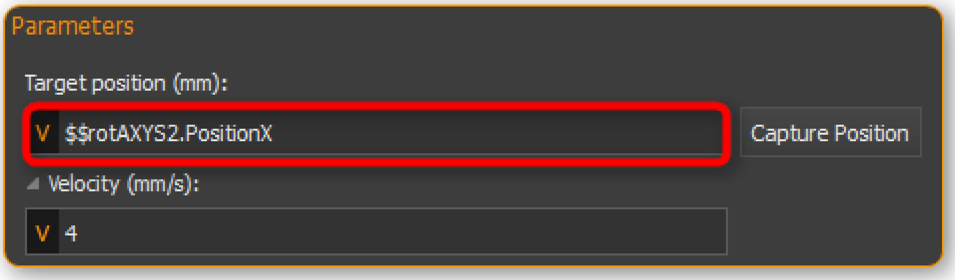


Figure 67: Using process data identifiers in input fields

The selected process data identifier will be entered into the input field. Similar to variable names, the process data identifiers have a particular form:

$$DeviceName.DeviceProperty

Each identifier starts with two dollar signs. A point separates the device name from the device property name. The entire process data identifier must not contain spaces or other special characters.

IMPORTANT

The device name and the name of the process data have a normalized form. All spaces are removed and replaced by underscores. The device name is the unique name of each device and may be different from the device caption that can configured by the user.

When the script function is executed, the process data is read from the device and used as function parameter for the script function.

6.8 Generic Script Functions

The *Script System* comes with a set of generic, or core, functions that control the program flow. These functions are always available irrespective of the activated plug-ins. You will find these functions in the categories *Core Functions*, *Function Sequences*, and *Time Functions* (figure below).

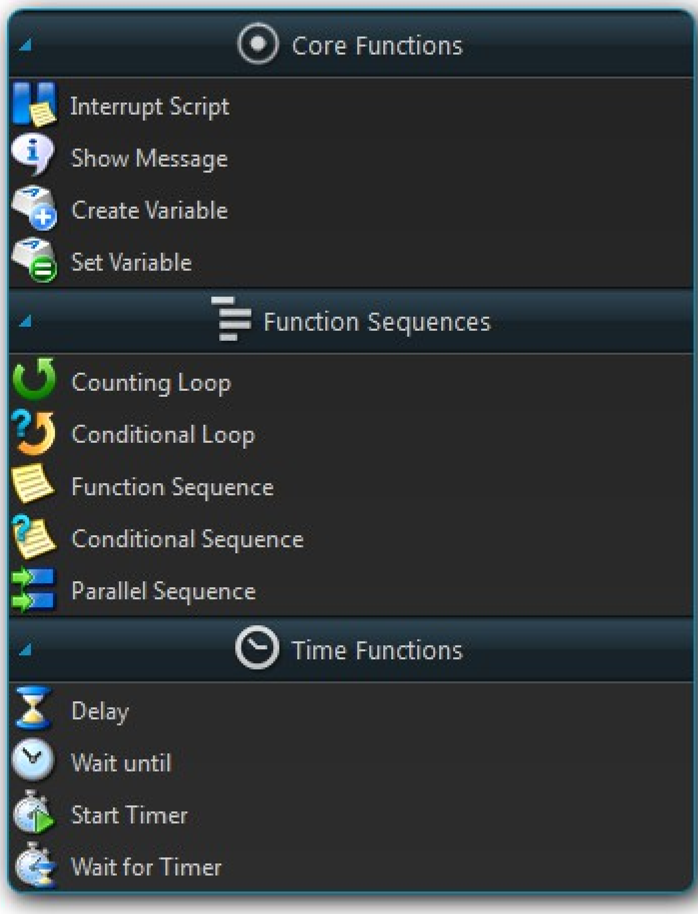


Figure 68: Generic script functions

6.9 Core Functions

6.9.1 Overview

The category Core Functions contains functions that control program flow and the use of variables.

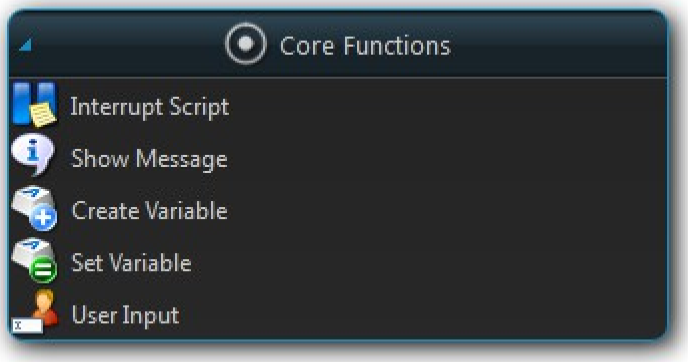
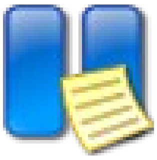


Figure 69: List of core functions

6.9.2 Interrupt Script



This function allows you to pause script execution. The user may then continue the script at a later time.

6.9.3 Show Message

The *Show Message* function allows you to display a message to the user and optionally interrupt script execution. It is well suited for applications that require an intervention by the user at certain times during the execution of a script.

When you run this script function, the message is shown in a message window …

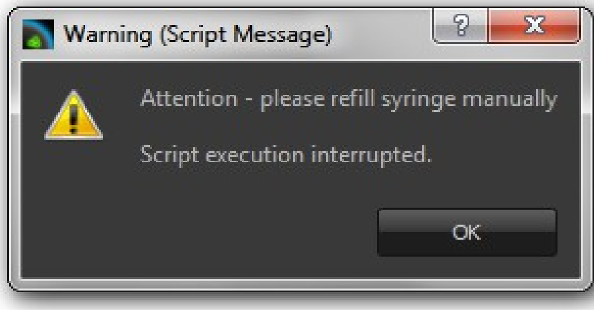


Figure 70: Message window

… and in the event log of the QmixElements software window:

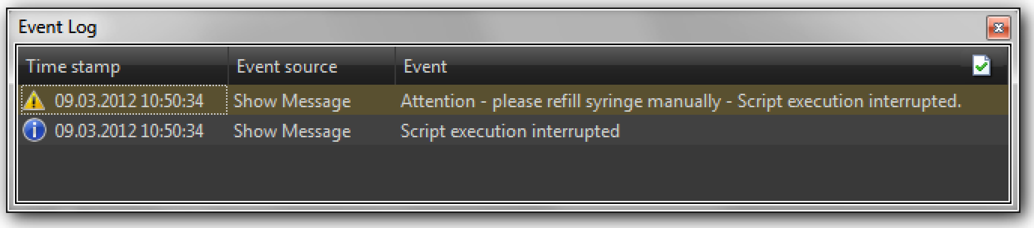


Figure 71: Event log

If several message functions have been executed, the message window will always show the message of the last function. All messages are displayed in chronological order in the *Event Log*.

6.9.3.1 Show Message Configuration

To configure the messaging function proceed as follows:

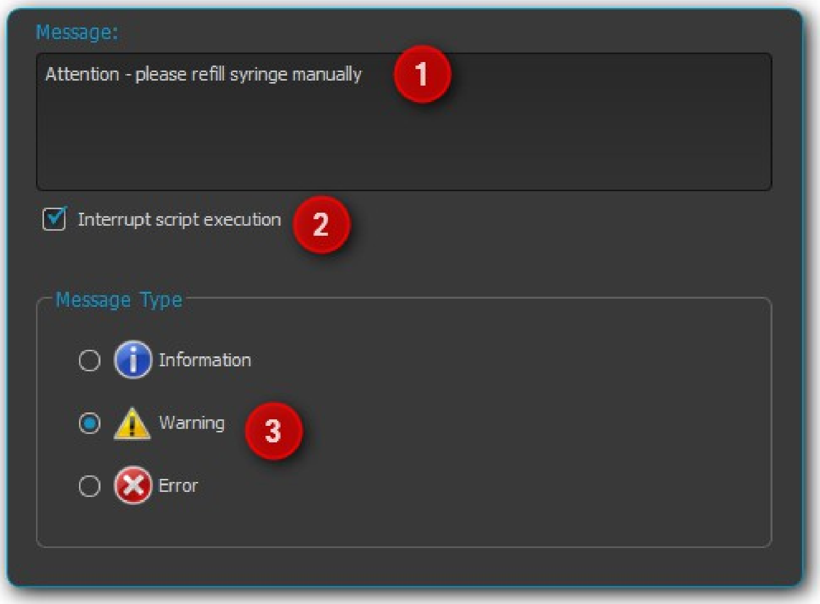


Figure 72: Message configuration panel

1 Enter your message text into the *Message* field.

2 Select whether the message should pause script execution. If the option *Interrupt script execution* is checked, the script pauses at the message function until the user continues script execution.

3 Select the type of message you want to display in the *Message Type* group. Depending on the message type, the relevant icon appears in the event log and in the message window and the message is highlighted with a type-specific color in the event log.

6.9.3.2 Displaying variables content or process data

In the *Message* field of the function, you can also use variables (like *$Volume*) or process data identifiers (like *$$neMESYS1.SyringeFillLevel*). These serve as place holders and are replaced at script runtime by the contents of the variables or the process data of a certain device. As soon as you enter a dollar sign *$* in the message input field, a list of available variables is displayed, from which you can select a variable (see figure below).

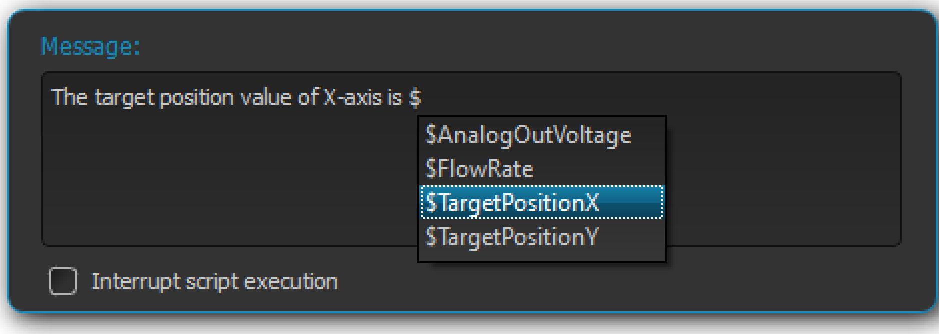


Figure 73: Using variables in Show Message function

E.g. you can enter the following text in the message input field:

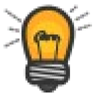
```
The contents of variable A is $A.
```

If the variable *$A* contains the value 25.3 at runtime, then the function will display the following message:

```
The contents of variable A is 25.3.
```

HINT

You can display the contents of variables using the Show Message function, e.g. to check the results of calculations.



HINT

*Click with the right mouse button in the input field and then select the context menu item **Insert device property** to insert a process data identifier of device process data you would like to print out, when Show Message function is executed.*

6.9.4 Creating a Variable

With *Create Variable* you can define a new variable and assign a name, a type, and a value to that variable (see figure below):

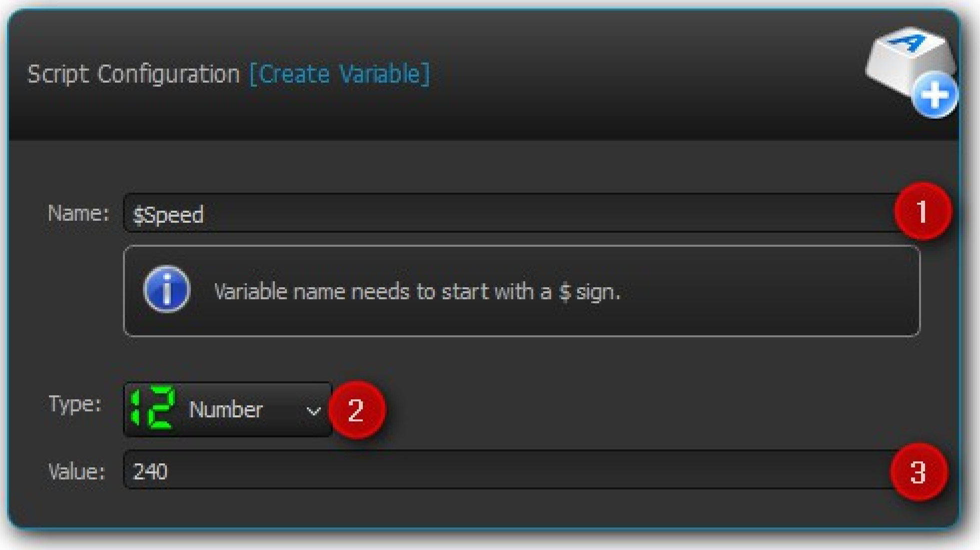


Figure 74: Configuration of a variable

1 *Name* – set a unique name for your variable. For details concerning the naming of variables, please refer to section 6.6.3.

2 *Type* – choose the kind of variable you want to set.

3 *Value* – give the variable a value or, if the variable is of the type *Calculation*, enter the mathematical expression to calculate the variable.

TIP

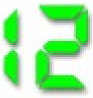
To prevent an incorrect variable name, the ***Name*** *field accepts only names that start with a $-symbol.*

6.9.4.1 Types of Variables

The QmixElements script system supports the following four types of variables:

Boolean

Boolean variables use one of the two values *true (1)* and *false (0).*



Number

Number variables use numeric values.

Text

Text variables use alphanumeric strings, including words or texts.

Calculation

Calculation variables save values that are calculated during operation

6.9.4.2 Calculating using Variables

If you set the variable type to Calculation, you can define an expression in the field *Value* that is to be calculated during run time. This expression may include other variables to use their respective values.

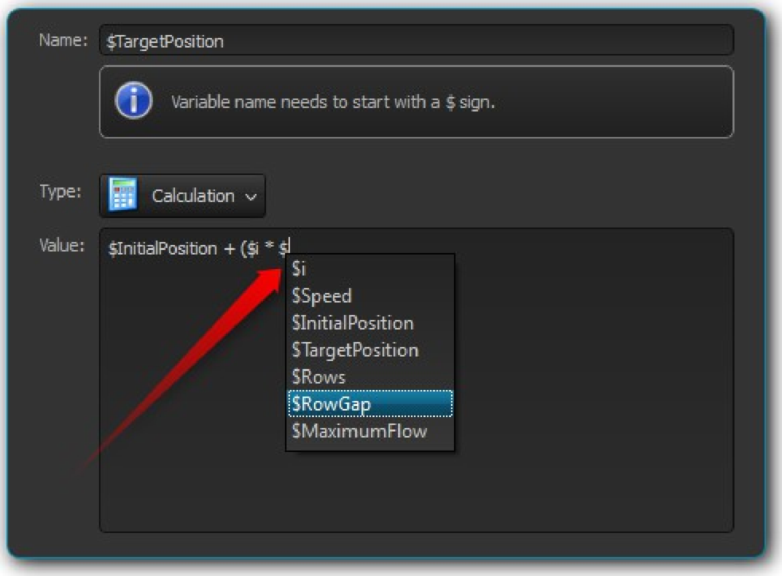


Figure 75: Defining calculation variables

HINT

The auto-complete feature will support you during entering a new expression by listing all previously defined variables.

HINT

*Click with the right mouse button in the input field and then select the context menu item **Insert device property** to directly access device process data values in the calculation.*

6.9.4.3 Operators

The following tables summarize available operators and their syntax:

Arithmetic Operators		Example
+	Addition	`$a + $b`
-	Subtraction	`$a - 1`
*	Multiplication	`5 * $b`
/	Division	`$a / 10`
%	Division and returning the remainder	`$a % 5`

<table>
<tr><th colspan="2">Logical Operators</th><th>Example</th></tr>
<tr><td>==</td><td>Checks for equivalence</td><td>1 == 2</td></tr>
<tr><td>!=</td><td>Checks for difference</td><td>5 != 6</td></tr>
<tr><td>></td><td>Checks for Larger Than</td><td>7 > 2</td></tr>
<tr><td><</td><td>Checks for Smaller Than</td><td>3 < $a</td></tr>
<tr><td>≥</td><td>Checks for Larger or Equivalent</td><td>$b ≥ 3</td></tr>
<tr><td>≤</td><td>Checks for Smaller or Equivalent</td><td>$a ≤ $b</td></tr>
<tr><td>&&</td><td>Links two values with the logical AND</td><td>true && false</td></tr>
<tr><td>||</td><td>Links two values with the logical OR</td><td>$a || (b < 5)</td></tr>
<tr><td>!</td><td>Links a value with the logical NOT</td><td>!true == false</td></tr>
</table>

String Operators		Example
+	Assembles two strings or texts	`text + $a`

6.9.4.4 Operator Hierarchy

The operators are ranked by a predefined hierarchy. When complex calculations are to be carried out, which combine a number of different operators, these operators will be resolved according to the following order:

1. rank: `||`
2. rank: `&&`

3. rank: `==` `!=`

4. rank: `<` `≤` `>` `≥`

5. rank: `+` `-`

6. rank: `*` `/` `%`

7. rank: `!`

8. rank: `()`

Parentheses allow you to influence the hierarchical order in which the various operators are executed according to your needs.

6.9.4.5 Examples

...for the use of *Calculation* variables:

Example 1

Adding two variables *a* and *b* and storing the result as a third variable *c*:

Name: `$c`

Value: `$a + $b`

Example 2

Multiplication of the variable *Speed* with 5 and storing the result as the same variable:

Name: `$Speed`

Value: `$Speed * 5`

Example 3

Connecting the two Boolean variables *SwitchedOn* and *CheckOk* with the logical AND and save the result as the variable *TurnOn*:

Name: `$TurnOn`

Value: `$SwitchedOn && $CheckOk`

6.9.5 Set the Value of a Variable

With *Set Variable* you can assign a new value to an already existing variable. The configuration of the variable is equivalent to the function *Create Variable.*

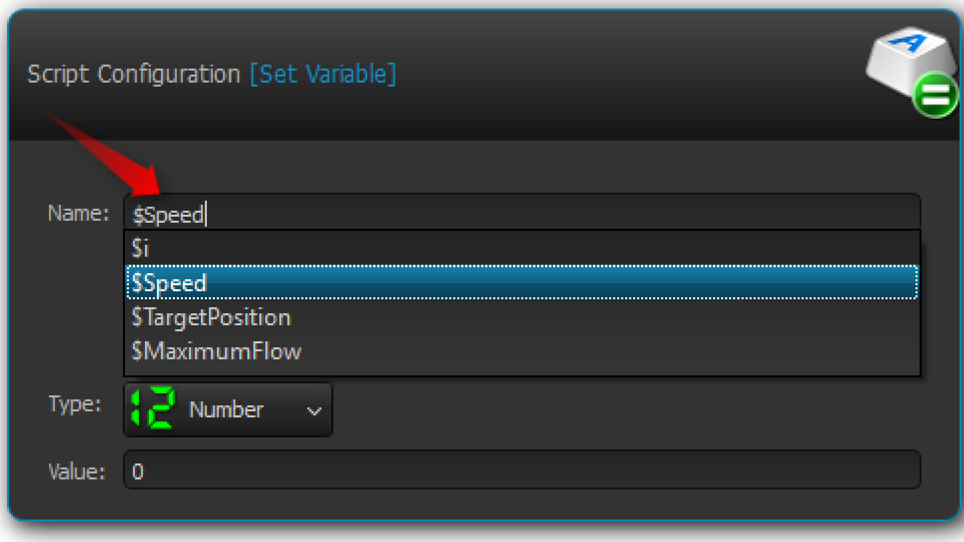


Figure 76: Entering a variable name using the auto-complete feature

HINT

When entering a name, the auto-complete feature will provide you with a selection of all previously defined variable names.

IMPORTANT

To be able to set the value of a variable using the ***Set Variable*** *function, the variable has to be defined beforehand, e.g., by using the function* ***Create Variable****.*

6.9.6 Create Property Variable

Use this function to create a special variable that is associated with a certain property of a certain device. Via this variable you can directly access the assigned device property from within a script, without any additional function calls.

If you e.g. want to read an analog input value using normal variables, you first need to create a variable via *Create Variable* function and then use the *Read Process Data* function to store the analog input value into this variable. Only then you can use this variable for further calculations and tests in script functions. Whenever you need a new analog input value, you must call the *Read Process Data* function again.

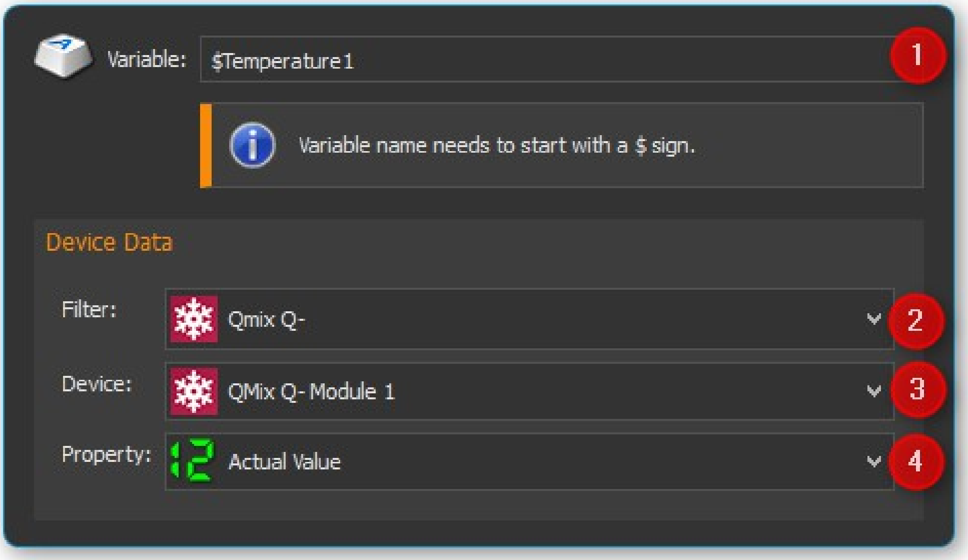


Figure 77: Create Property Variable configuration panel

With property variables that access to device properties (e.g. analog input value) is considerably simplified. If you create a property variable, you use the configuration panel to connect a device property permanently with a variable. Whenever you read the value of this variable from within a script function, automatically the current value of the connected device property is read.

You can set the name of the variable in the input box *Variable* 1 (figure above). Select the device you want to access from the drop-down list *Device* 3, which lists all modules that are part of the current configuration. To simplify the search for a device, you can filter the device list by selecting an appropriate filter (device type) from the *Filter* drop-down field 2.

After the device has been selected, use the last drop-down list *Property* 4 to select the device property, the process data value that you want to access.

In summary, this function can be described as follows:
Assign the device property 4 *of the device* 3 *to the variable* 1 *and return the device property value if a read access on the variable takes place.*

IMPORTANT

Property variables can not be used to store values during calculations — use normal variables for calculations and value storage.

6.9.7 User Input

This function allows the reading of a single input value (number or text) into an existing variable. In the configuration area of the *User Input* function (see figure below), you can configure the following parameters:

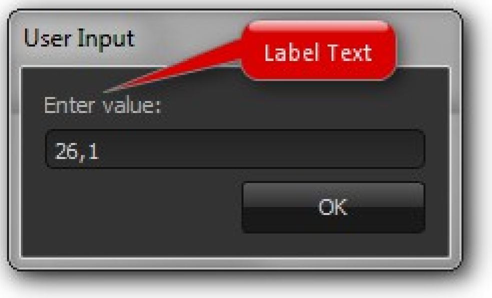
Figure 78: Configuration User Input function

1 *Label Text* – Here you specify a text that will be displayed to the user during the execution of the function above the input field as a label and explanatory text.

2 *Variable* – In this field you specify the variable to store the value entered by the user.

3 *Type* – You can specify the input type, number or text, here. Depending on the selected input type the user input dialog accepts only certain values.

4 *Test Input Dialog* – Clicking this button allows you to test the input dialog to check how the user will see the input dialog later during execution of the script function (see figure below).



6.10 Function Sequences

6.10.1 Overview

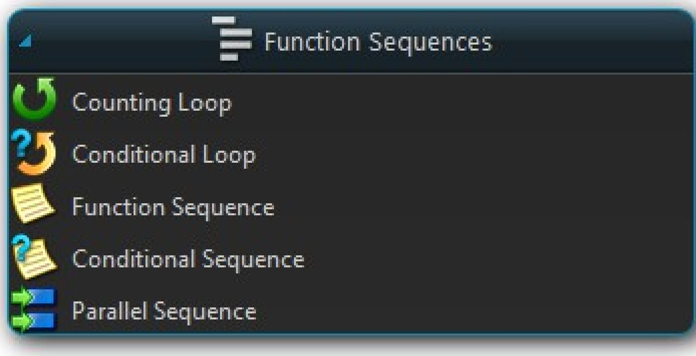


Figure 79: List of function sequences

The *Function Sequences* category contains various types of function sequences. Function sequences are script functions that may contain a number of different functions. When a *Function Sequence* is called, all script functions contained within this sequence are executed.

6.10.2 Counting Loop

The *Counting Loop* function allows for multiple sequential executions of the same function(s) within that loop. The number of times the loop is to be run is defined in the configuration area (*Loop Cycles*).

In addition, you may set a name for the count variable (*Loop Counter*) of that loop. Within that loop, functions that support the use of variables may then use this variable to execute specific tasks or to perform additional calculations.

The number of loops may be defined explicitly, by setting a number, or implicitly, via a variable that returns a number (figure below, 1).

Additionally, you can assign a custom name to the Counter *Variable* of the present loop function (figure above, 2). Within this loop, other functions may access the current loop count via this variable and use it for further calculations. This may be used, *e.g.*, to control incremental moves of positioning systems.

IMPORTANT

The counting variable of a counting loop always starts with the value 0 (zero). This is, a loop count set to 10 contains the values 0 to 9 for the successive loops.

IMPORTANT

You should always insert a Delay Function in a loop to prevent the loop from blocking the application interface by consuming 100% processor power. A small delay of 10ms – 100ms is sufficient.

6.10.3 Conditional Loop

6.10.3.1 Introduction

This conditional loop function is used for multiple execution of functions within the loop as long as the loop condition set at its beginning is met. The termination condition in the loop header is checked before each iteration of the loop. The loop execution is stopped or skipped if the termination condition is false.

Spelled out, the *Conditional Loop* function reads as follows:

Repeat the loop and execute the function(s) contained within that loop as long as *the loop condition holds true.*

QmixElements provides two alternatives to define a *Conditional Loop* – a simple and an advanced mode that are to be selected in the configuration panel.

6.10.3.2 Configuration - Simple Mode

In the configuration panel, you may choose the simple mode to comfortably set the loop conditions directly via the Graphic User Interface – simply uncheck *Advanced Mode* 1. The following parameters may be set:

Figure 80: Configuration panel for conditional loops – basic mode

1 *Advanced Mode* – this button is to toggle between the simple and advanced mode (see section below).

2 *Variable Name* – here you insert the name of the variable the value of which is to be tested against. The variable must start with the Dollar sign ($) that is required to signify the name as a variable within QmixElements.

3 *Variable Type* – select the required variable type. The selection will determine the kind of condition to be selected next.

4 *Conditional operator* – this drop-down menu lists all operators available for the set function type.

5 *Conditional value* – the value or a variable that the variable from 2 is to be compared with.

Sample Configuration: Basic Mode

The loop condition to be checked against is whether a the variable `$Position` is smaller than or the same as `25.000,31`.

2 Variable Name	3 Type	4 Operator	5 Condition
`$Position`	`Number`	≤	`25.000,31`

The script editor shows the complete mathematical expression:

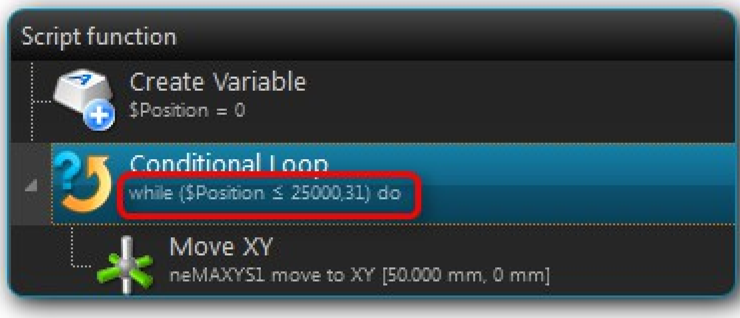


Figure 81: Example for a Conditional Loop

The resulting expression effects that the loop will be rerun as long as the value of the variable `$Position` is smaller than or the same as `25.000,31`.

6.10.3.3 Configuration - Advanced Mode

Whereas the simple mode provides the user with a graphical interface, the advanced mode allows to directly insert an equation with variable names and mathematical and logical expression.

Figure 82: Configuration panel for conditional loops – advanced mode

The advanced mode provides a simple input box. Here you can set the loop condition using Java Script expressions. As shown in the example above, you can insert rather complex mathematical expressions or multiple variables to set conditions.

IMPORTANT

You should always insert a Delay Function in a loop to prevent the loop from blocking the application interface by consuming 100% processor power. A small delay of 10ms – 100ms is sufficient.

6.10.4 Function Sequence

A *Function Sequence* may be used to to group multiple successive functions. For one, this simplifies navigation within the function tree: The content of a function sequence may be hidden if required, thus making more of the entire tree visible within the *Script Function* panel. Also, a function sequence simplifies the setup of program scripts. Function sequences that consist of several functions may be copied to other positions within a script. This means, if you want to copy a number of successively executed functions, for instance to use them at various places in the program either sequentially or in parallel, then move them to the relevant section of your function sequence.

6.10.4.1 Naming a Function Sequence

A function sequence may be given a name; that name will then be shown in the function tree. This simplifies navigation in particular with larger numbers of function sequences. To do this, simply enter the name in the *Caption Field* (Figure below, 1).

6.10.4.2 Loading a Function Sequence

The *Function Sequence* also allows the loading of pre-stored scripts: You may load a script, which you have created and saved previously, into your current function sequence. To do this, just click on the *Load from script file* button (figure below, 2) and select the desired file.

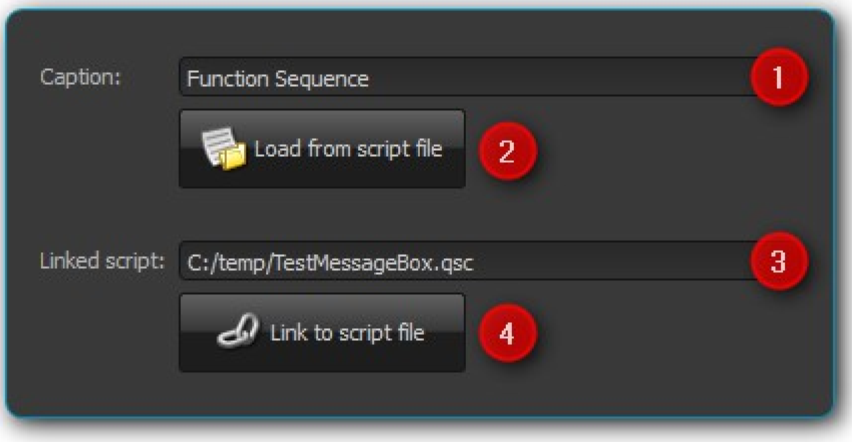


Figure 83: Function sequence configuration

6.10.4.3 Linking to a Functional Sequence

Instead of loading a function sequence, which essentially means to copy the content of an existing script into the current script, you can also set a link to a function sequence. Setting a link means that the content of the function sequence is not itself inserted or copied into the current script. Rather, its content is always re-loaded from the linked script file whenever it is called. That is, whenever called, all linked instances are updated with the current content of that linked script: Any change to your source script will be executed in your current script accordingly.

To setup a script link, click on the *Link to script file* button (figure above, 4) and then select the external script that you want to link into your current script.

Importantly, changes applied to the linked script are not automatically loaded into your current script. To update the current script with the changed content of a linked script, you must reload the current script.

ATTENTION

The position of the linked script within the file system is stored as a relative path in reference to your current script: If you move the current or the original script to another folder, you need to adjust the script link accordingly.

6.10.5 Conditional Sequence

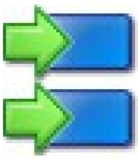
A *Conditional Sequence* provides the user with a number of script functions the contents of which will only be carried out if the condition set in its header is true. If the condition is not true at the time when it is tested, the function(s) contained within that conditional sequence will be skipped.

Spelled out, the Conditional Loop function reads as follows:

If the condition is true, carry out the function(s) contained within that sequence; else continue after that sequence.

The configuration of the Conditional Sequence is analogous to the configuration of the Conditional Loop as described above in Section6.10.3.2.

6.10.6 Parallel Sequence



The script function *Parallel Sequence* allows the concomitant execution of functions or function sequences. The configuration of this script function is identical to the configuration of a normal function sequence. In contrast, however, to a standard function sequence, the individual functions are not executed sequentially but in parallel. In other words, when executing this function, all direct child functions start simultaneously.

The parallel sequence finishes only after all parallel child functions have been completed. Thus, the function is suitable for execution of parallel processes in which it is important to wait for the end of the process that takes the longest before proceeding to the next step.

6.11 Time Functions

6.11.1 Overview Time Functions

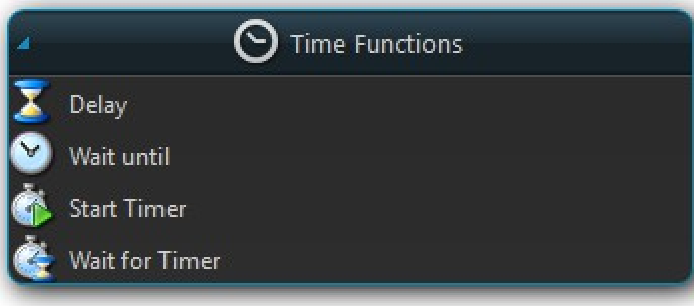


Figure 84: List of timing functions

This category contains functions to enable a time-controlled program flow. The execution of scripts may be suspended for a certain amount of time or can be interrupted to wait for certain timer events.

6.11.2 Delay

The *Delay* function interrupts program execution for a configurable length of time. You can set the delay time (hours, minutes, seconds, or milliseconds) in the configuration area (see figure below).

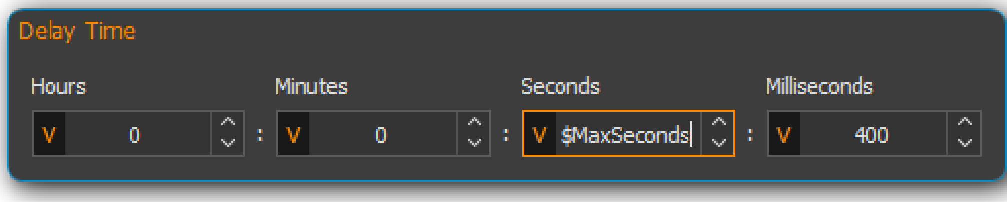


Figure 85: Configuration of Delay function

All input fields support the use of variables. You can freely and flexibly assign any values to the input fields and you can freely mix variables and values with each other.

6.11.2.1 Sample Configuration

If you e.g. would like to set a delay of10 minutes and 15 seconds, you can do so by very different assignments of the input fields:

Hours	Minutes	Seconds	Milliseconds
0	10	15	0
0	0	615	0
0	0	0	615000
0	10	0	15000

6.11.3 Wait Until

You can delay program execution until a precisely defined date and time with the *Wait until* function. Program execution is not resumed until the configured date and time has been reached.

IMPORTANT

Please note that the Wait until *function can only be correctly executed once. If the script is executed again, and the time point thus lies in the past, this function will simply be ignored. In other words, date and time of a* Wait until *function must be re-configured before restarting a script so that an effective delay can again be executed.*

6.11.4 Start Timer

The *Start Timer* function starts a specific timer. If you already have started a timer, you can use the *Wait for Timer* function to wait for the expiration of this timer. To configure a timer, enter a unique name for the timer (*Timer Name*) and the period till timer expiration (*Timer Running Time*).

If this function is called with the name of a timer that has already been started, that timer is restarted with that configured *Timer Running Time*.

6.11.5 Wait for Timer

The *Wait for Timer* function interrupts script execution until the specified timer has expired. A timer with the configured timer name must have been started before via the function *Start Timer*. If no timer has been started or if the timer with the given name has already expired, program execution will continue without interruption.

With the two above functions, *Start Timer* and *Wait for Timer*, it is very easy to execute functions at fixed intervals. For this, you can, for example, run a function or function sequence along a timer function in a *Parallel Sequence*.

6.12 Device Functions

6.12.1 Introduction

Figure 86: Generic device functions

The script system comes with a number of generic script functions for accessing devices (device process data, device properties). These functions are available independently of the loaded plugins. You find these generic device functions in the category *Device Functions*. Read Process Data

6.12.2 Read Device Property

The function *Read Device Property* allows you to read various process data from a module and save them in a variable.

Select the device you want to access from the drop-down list *Device* 2, which lists all devices that are part of the current configuration. To simplify the search for a device, you can filter the device list by selecting an appropriate filter (device type) from the *Filter* drop-down field 1. After the device has been selected, use the last drop-down list *Property* 3 to select the device property that you want to access.

IMPORTANT

Only devices and device properties are displayed that allow read access.

In the input field *Variable* 4, enter the name of the variable into which the value is to be saved during script runtime (figure below).

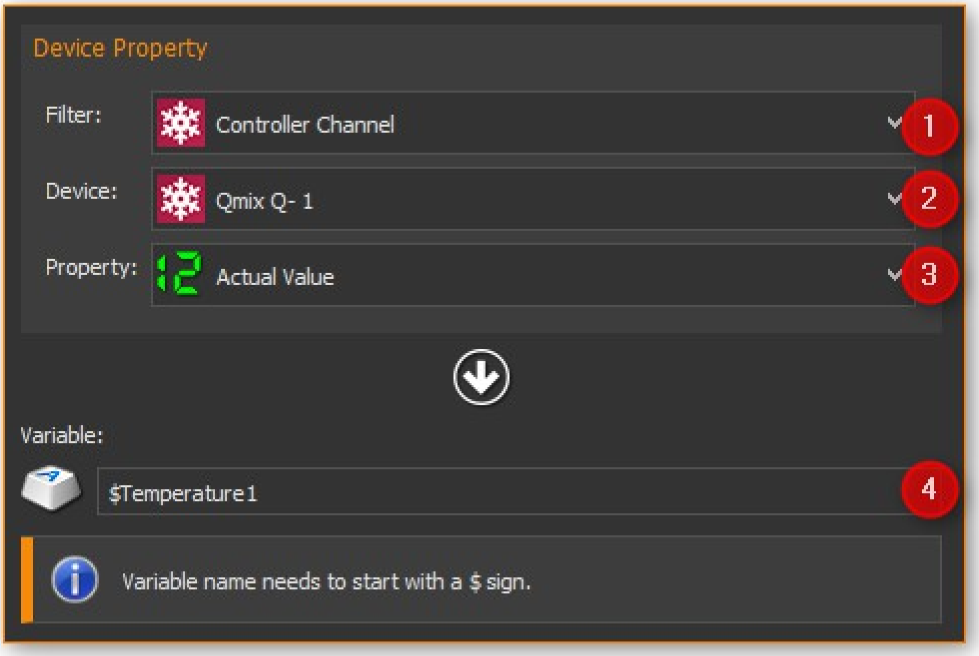
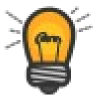


Figure 87: Reading device property set-up panel

In abbreviated form, the set-up follows the following logic:

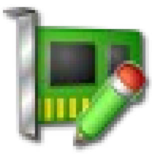
Read from device 2 *the value of parameter* 3 *and save it as variable* 4.



TIP

When entering a name, the auto-complete feature will provide you with a selection of all previously defined variable names.

6.12.3 Write Device Property



This function is for writing process data from devices. In the input field 1 enter the value to be written. You can also use variables or Device Property Identifiers in this field.

In the configuration panel (figure below), select the device and the appropriate device property that you want to write 2 as described for function Read Device

Property. The device selection controls 2 only show devices and device properties, which allow write access.

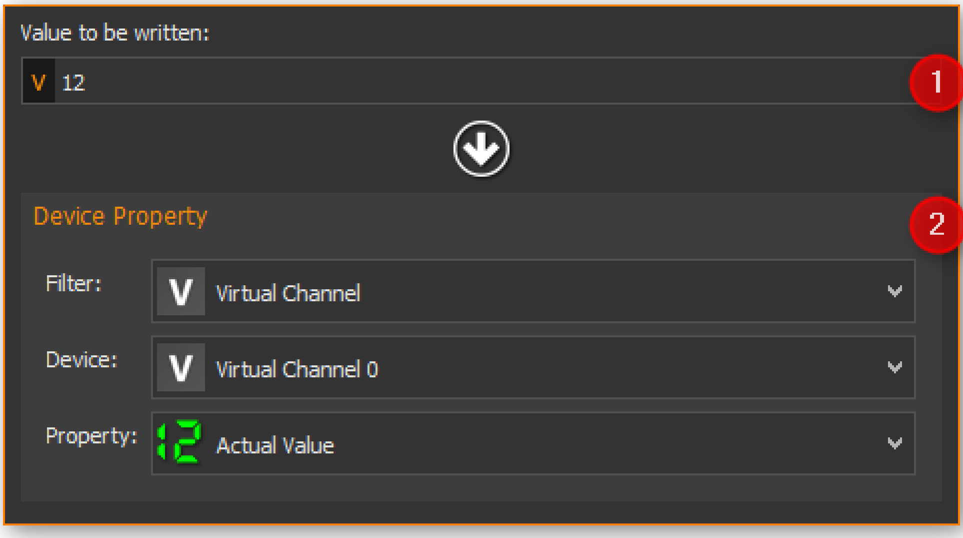


Figure 88: Scriptconfiguration Write Device Property

The value in the input field 1 is then written at run time in the selected device property.

6.12.4 Wait for Process Data

This function allows a script to wait for a specific device property to reach a predefined condition. Once this condition is fulfilled, the script will resume.

In the configuration panel (figure below), select the device and the appropriate device property that you want to monitor 1 as described for function *Read Device Property*.

In the *Condition* input box you can configure the condition that is to be checked against. First define an *operator* 2 and then the *target value* 3 the device parameter is to be compared with. In the value field 3, you can also use variables to set a test condition.

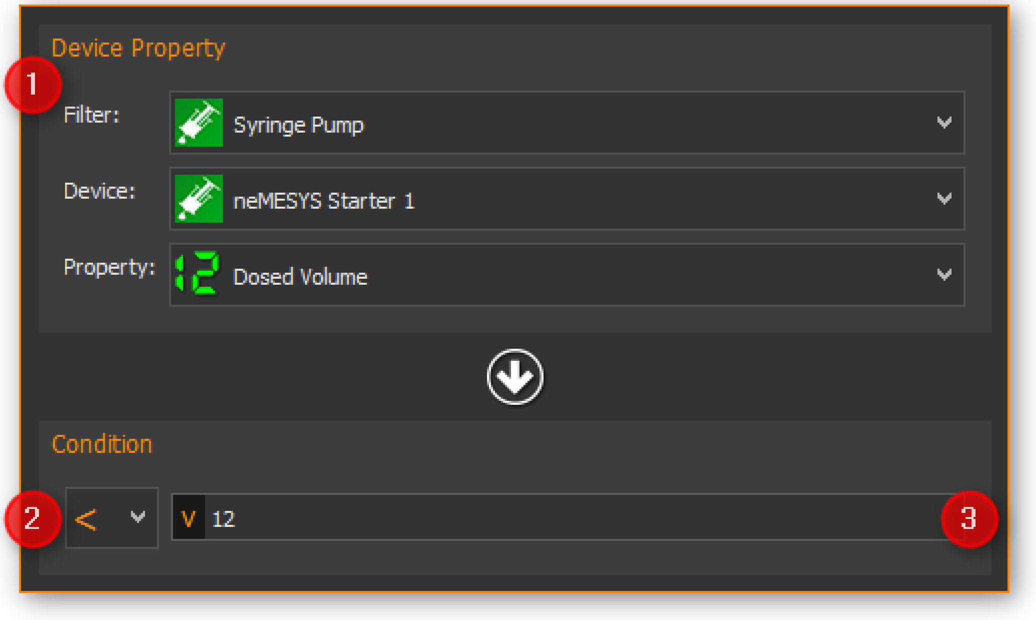


Figure 89: Script configuration Wait for Device Property

In short, the above set-up procedure reads as follows:

Continue script execution, if the selected property of the device 1 *meets the condition* 2 3.

TIP

As a test / comparison condition variables can be used.

6.13 Math Functions

6.13.1 Introduction

In the category *Math Functions* you can find mathematical functions that simplify complex calculations, thus reducing implementation complexity and increase the clarity of script programs.

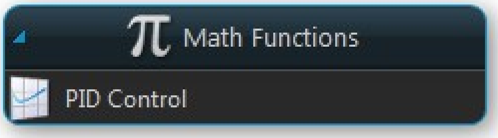


Figure 90: Math Functions

6.13.2 PID Control Function

6.13.2.1 Introduction

This function implements a PID controller using a PID algorithm for applications that require an efficient algorithm. The PID algorithm features control output range limiting with integrator anti-windup.

Currently, the Proportional-Integral-Derivative (PID) algorithm is the most common control algorithm used in industry. Often, people use PID to control processes that include heating and cooling systems, fluid level monitoring, flow control, and pressure control. In PID control, you must specify a process variable and a setpoint. The process variable is the system parameter you want to control, such as temperature, pressure, or flow rate, and the setpoint is the desired value for the parameter you are controlling. A PID controller determines a controller output value, such as the heater power or valve position. The controller applies the controller output value to the system, which in turn drives the process variable toward the setpoint value.

6.13.2.2 Configuration

In the configuration area of this function you can configure all parameters required for proper PID control.

Figure 91: PID Control function configuration

You need to configure the following parameters:

1 *Actual Value Input* – this parameter specifies the measured value of the process variable being controlled. This value is equal to the feedback value of the feedback control loop. Enter a variable name here, or enter a *Device Property Identifier* in order to directly read specific process data of a certain device.

2 *Setpoint* – specifies the setpoint value, or desired value, of the process variable being controlled. You can enter a fixed value (like 50°C) or you can pass the value in a variable.

3 *Control Loop Parameters* – specify the proportional gain (**K**), integral time (**T$_I$**), and derivative time (**T$_D$**) parameters of the controller and thus directly affect the control characteristic of the controller.

HINT

You can use the PID control function to implement a P, PI or PD controller by simply setting the unneeded PID gains to 0.

4 *Controller Output* – This section groups all settings for the the controller output.

Control Value Output returns the control output of the PID algorithm that is applied to the controlled process. I.e. if you implemented a temperature control loop, the *Control Value Output* would be the heating power that must be generated from the heater. Enter a variable name of a variable that can store the output value or use a [Device Property Identifier] to write the value directly into a device property of a certain device.

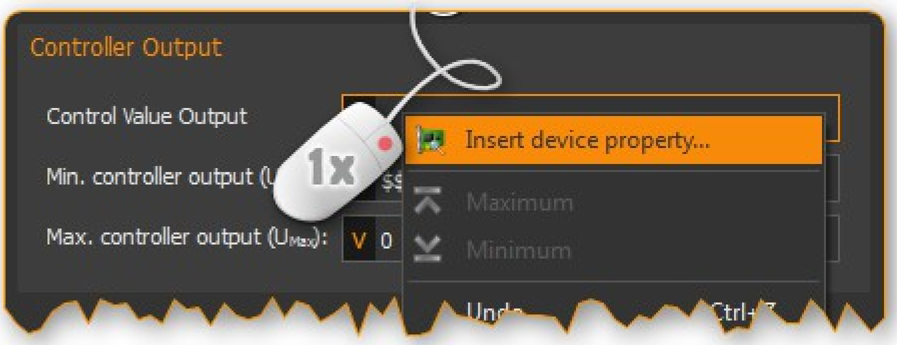


Figure 92: Enter process data identifier via context menu

With the parameters U_{max} and U_{min} you limit the range of the controller output. If you e.g. control the heating power via a 0 – 5 V analog output, then enter 0 for U_{min} and 5 for U_{max}. If the control algorithm generates values that are outside of this range, they will be limited to the range automatically.

6.13.2.3 PID control parameters

A set of PID control parameters contains a proportional, an integral and a derivative part.

Proportional part

The proportional part computes the control deviation from the setpoint (W) and the actual value (X)

$$e=(X-W)$$

and multiplied with the proportional gain, gives it as control value (Y) to the controlled system. The following equation shows the proportional part.

$$Y=K\cdot e$$

Integral part

Mathematics forms the integral part the area enclosed by control deviation and timeline. If there is a contant control error, the integral part grows ramp shape. For a constant control deviation the equation is:

$$Y=\frac{K}{T_i}\cdot e\cdot t+Y_{t0}$$

Y_{t0}:	Control value at the beginning of the observation
T_i:	Integration time

If the actual value equals the setpoint the control value does not change. The control value built up by the integral part remains and will not decrease until the actual value exceeds the setpoint value. With controlled systems containing a delay line the integral part eliminates the steady-state error which a proportional controller is not capable of. In general the following equation holds for the integral part.

$$Y = \frac{K}{T_i} \cdot \int_{t_0}^{t} e\,dt + Y_{t0}$$

With the integration time the speed of the controller can be changed. The smaller T_I the faster the integral part builts up a control value. The given equation states that the proportional gain K, too, impacts the integral part. In QmixElements the integral part can only be configured in conjunction with the proportional part. Thus the following equation applies:

$$Y = K \cdot e + \frac{K}{T_i} \cdot \int_{t_0}^{t} e\,dt + Y_{t0}$$

IMPORTANT

The integral part compensates the steady-state control error.

Derivative Part

The derivative part counteracts changes in the actual value. Two scenarios can be considered with respect to the effect of derivative part.

- After the actual value has reached a stable final value, it is decreasing because of a disturbance suddenly occurred. The derivative part gives an additional control value that helps increasing the actual value.
- If the setpoint value is increased, the actual value increases too. The derivative part recognizes the increasing actual value and slows down by an additional negative control value starting up to the target value.

In practical use the derivative part occurs only in combination with a proportional part. The controller equation is:

$$Y = K \cdot e - K \cdot T_d \frac{dx}{dt}$$

The bigger the proportional gain K and the derivative time T_D the bigger the effect of the derivative part and the stronger the change of the actual value is counteracted.

Summary

The following table summarizes the effect of the different control parameters.

PID parameter	Regulating on a disturbance of the controlled system	Start-up of the setpoint
K higher	stronger repsonse (reduced damping)	faster start-up
K smaller	weaker response (increased damping)	slower start-up
$\mathbf{T_I}$ higher	weaker repsonse, particularly on short-term disturbances, the controller has only a weak response	Slower start-up and compensation of the steady-state error
$\mathbf{T_I}$ smaller	stronger repsonse, particularly on short-term disturbances, the controller has only a weak response	faster start-up and compensation of the steady-state error (Overshooting if T_I is too small)
$\mathbf{T_D}$ higher	stronger repsonse	slower start-up (stronger reaction against changes in actual value)
$\mathbf{T_D}$ smaller	weaker response	faster start-up (weaker reaction against changes in actual value)

6.13.2.4 Programming the control loop

This section shows you how to realize a complete PID control loop in a script program by using the PID control function.

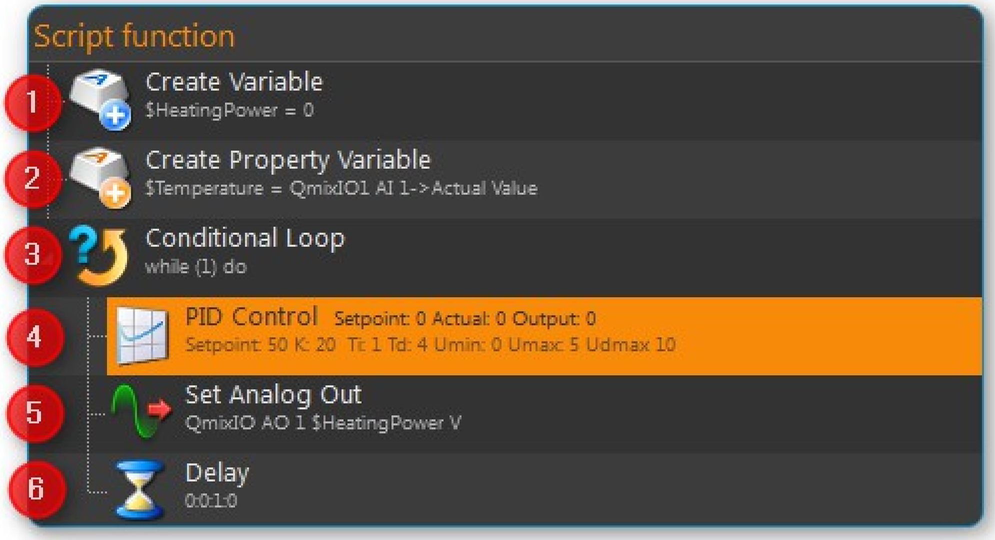


Figure 93: PID controller example scripts

To implement a control loop, proceed as follows:

1 Create a variable (*Create Variable*) in which you can save the output of the controller (in this case `$HeatingPower`).

2 Create a property variable (*CreateProperty Variable*) and connect it to a device property (in this case the variable `$Temperature` is connected to the analog input value of the device `QmixIO1 AI 1`) to access the input of the controller.

3 The controller must be called cyclically in a fixed time interval. For this, you should use a loop. In this case, use a conditional loop (*Conditional Loop*) with the loop condition 1. The condition 1 is always true and the loop runs forever and never stops, except the user stops program execution manually.

4 Now create a *PID Control* function within the loop. As control variable (*Setpoint*) enter a fixed value (e.g. 50°C). Use the property variable `$Temperature` as the process variable (*Actual Value Input*). Enter the variable name `$HeatingPower` in the field *Control Value Output* to save the output value into this variable.

5 Now you can pass the control loop output value that is stored in the `$HeatingPower` variable to the analog output function (*Set Analog Out*). Via this output the heating power is adjusted in this case.

6 Inside the loop you need to place a *Delay Function* to insert a defined delay time of one second. This time determines the frequency, with which the control algorithm is called and hence the *dt* that is used in the algorithm for the calculation of the control parameters.

Now you have built a control loop that reads the current temperature via an analog input, computes an output value in the PID control function, which is then used with an analog output for adjusting the heating power.

HINT

According to control theory, a control system must sample a physical process at a rate about 10 times faster than the fastest time constant in the physical process. For example, a time constant of 60 s is typical for a temperature control loop in a small system. In this case, a cycle time of about 6 s is sufficient. Faster cycling offers no improvement in performance

7 neMESYS Plugin

7.1 Introduction

The QmixElements neMESYS plugin controls the neMESYS syringe pumps.

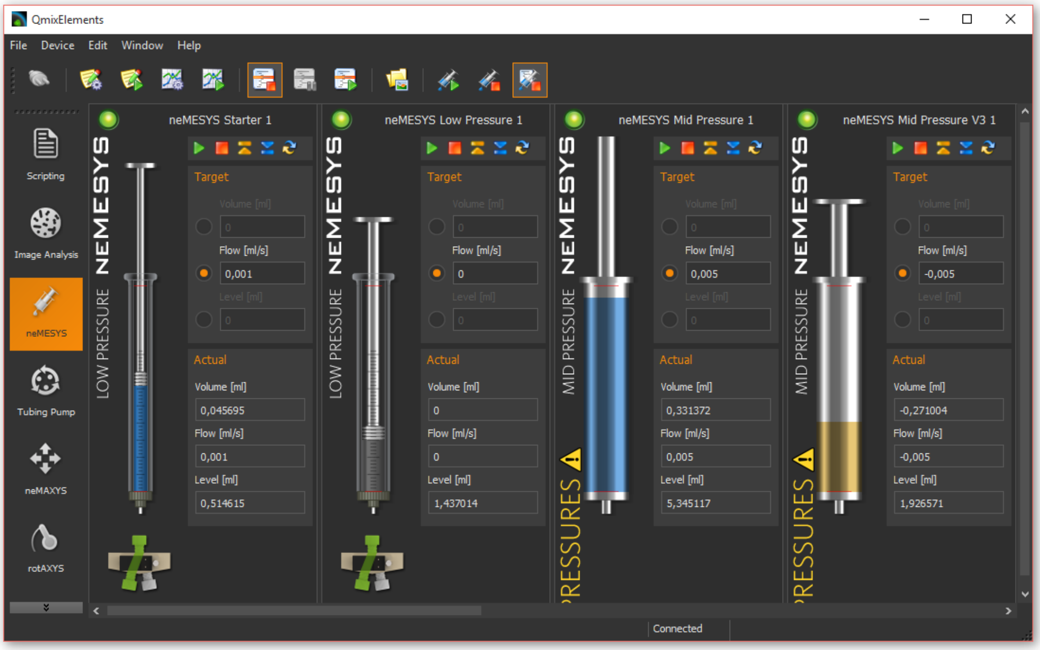


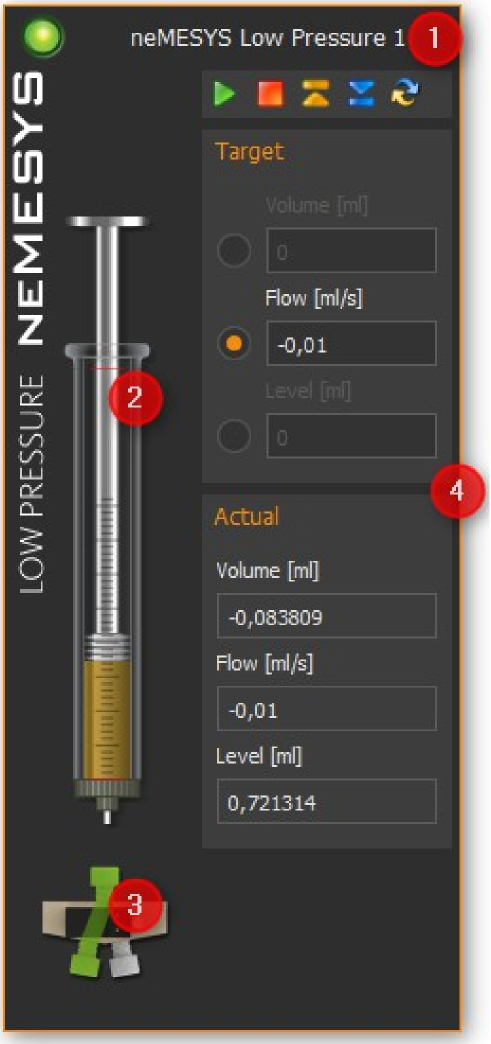
Figure 94: neMESYS workspace

You see operating panels for all the existing, configured syringe pumps displayed in the working area of the neMESYS plugins (see Figure above).

7.2 Operating Panel of Dosing Modules

7.2.1 Overview

Each operating panel is used to control, configure and visualize a neMESYS dosing module. It contains all the control elements for configuring and controlling the pump parameters and for visualizing the current status.



The panel consists of

1 Caption of the syringe pump

2 Level indicator

3 Valve switching and status

4 Control elements (depending on operating mode)

7.2.2 Changing module names

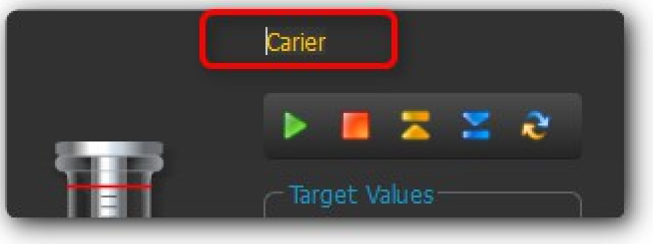
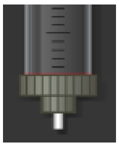


Figure 95: Changing module name

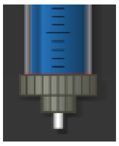
You can assign a unique name to each dosing unit. For example, you can indicate which liquid a dosing unit doses or what it is used for. To assign a new name, simply make a left mouse click on the name of the dosing unit. You can then enter a new name, and confirm the entry with *Return* (Figure below).

7.2.3 Syringe level indicator

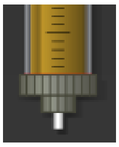
The fill level indicator of the syringe always shows the current position of the drive unit, and thus the current fill level of the syringe used. The colour of the syringe also indicates the current motion of the dosing unit, so you can immediately see the direction of motion of the syringe even with a very low flow rate. The following colours are possible:



Grey - The drive unit is currently inactive.



Blue - The syringe piston is moving in the direction of the syringe clamp (Dispensing).



Orange - The syringe piston is moving away from the syringe clamp (Aspirating).

This color coding of the direction of motion is also used in all other parts of the neMESYS plugin.

HINT

The colours of the syringes help you identify the direction of motion, especially with very low flow rates, because it is almost impossible to see the position of the syringe piston changing.

The thickness of the syringe in the display indicates the approximate size of the currently clamped syringe. You should be very cautious with the use of higher flow rates, especially with very large syringes, because very high pressures can develop very quickly, which could damage the device (valve) or your application.

ATTENTION

Risk of high pressures damaging the connectors in the liquid path. To avoid high pressures causing damage, never make a reference move with a high-pressure syringe clamped.

7.2.3.1 Software limits

The travel path of a syringe is limited in the software by a minimum and a maximum value in order to prevent irreparable damage to the syringe or the piston being pulled out of the body of the syringe. These values are indicated by two red marks in the fill level indicator (Figure below), and they have to be defined for each syringe in the syringe configuration (section 7.4).

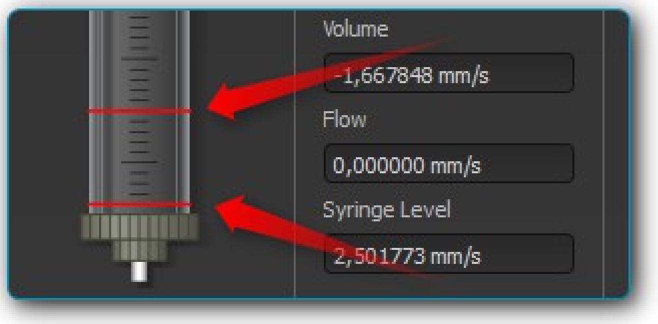


Figure 96: Software limits

7.3 Valve Switching

If your dosing unit has a valve, you can switch the valve or configure the automatic valve switching in this part of the operating panel. The current switching status of the valve is indicated by the valve control element.

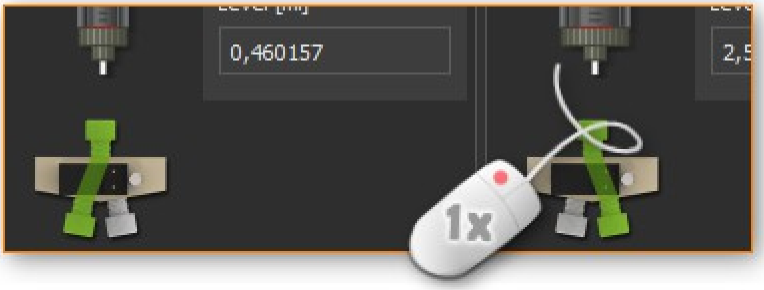


Figure 97: Valve switching

To switch the valve, simply make a left mouse click on the valve icon. You should hear a soft click, and the valve image should switch to the other switching status.

7.4 Syringe configuration

7.4.1 Introduction

The software provides user-friendly ways of configuring and managing your own syringes. The configuration and use of specific syringes in the software thus fulfil two requirements:

1. The software calculates all flow rates and volumes on the basis of the configured syringe.

2. The motion of the piston is limited by the configured syringe length and the set limits.

7.4.2 Opening the configuration dialog

The software constantly shows the current flow rate for each dosing unit. The program needs the parameters for each syringe in order to calculate the flow rates. You should configure these values properly each time a syringe is changed.

To do this, make a right mouse click on the axis of the image of the syringe you want to configure. Select the *Configure Syringe* menu item in the appearing context menu. Then select the syringe you want to configure, as shown in Figure below. The *Configure syringe* dialog then opens.

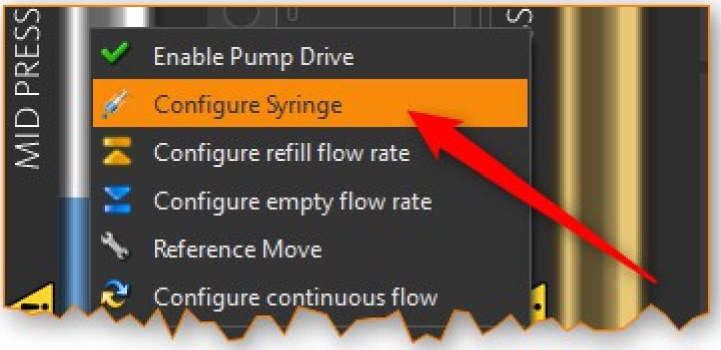


Figure 98: Starting syringe configuration

IMPORTANT

Please note that when a syringe is changed, the fill flow rate and empty flow rate of this dosing unit are reset to 0. This means that you have to reconfigure these values after each syringe change.

7.4.3 Syringe selection dialog

In this dialog, you can also select the desired syringe from a list of available syringes. The following options are available here:

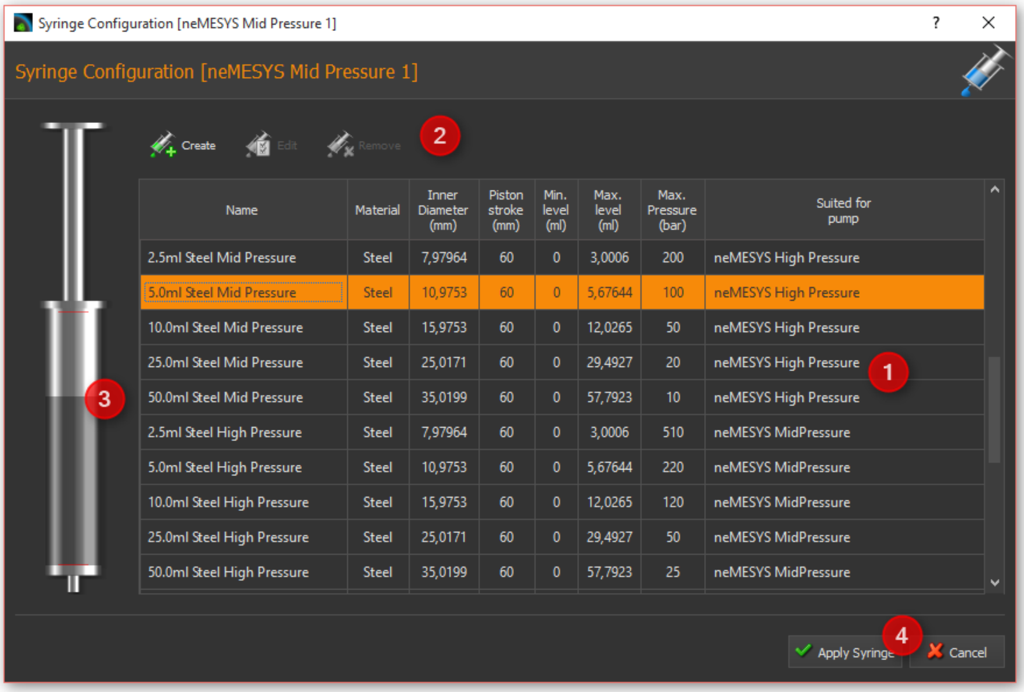


Figure 99: Syringe selection dialog

The list view 1 displays all the syringes that you have previously created. If the list contains a suitable syringe, you can select it, and close the dialog by clicking *Apply Syringe*. Alternatively, you can simply select a syringe by double clicking the syringe.

If the list does not contain a suitable syringe, a new syringe can be created by clicking the *Create* 2 button. Clicking the *Edit* button opens a window to edit the syringe parameters of the currently marked syringe. You can remove a syringe from the list by clicking the *Delete* button. A syringe can also be removed by pressing the *Delete key*.

On the left you will see a preview image 3 of the currently selected syringe.

The selection can be completed at any time by clicking *Apply Syringe* 4 or cancelled by clicking *Cancel*.

IMPORTANT

Predefined syringes can not be edited or deleted.

7.4.4 List of available syringes

The list of available syringes shows all syringe parameters in tabular form:

The following columns are available:

Column	Description
Name	Unique syringe name. Syringes with the same name are not allowed.
Material	Steel or glass – the syringe display is adjusted accordingly
Inner Diameter (mm)	The configured internal diameter - important for the conversion of speed and position into flow rate and volume
Piston Stroke (mm)	The maximum physical piston stroke, in which the piston does not slip out of the syringe cylinder. This value limits the travel of the pump axis.
Min. level (mm)	The minimum syringe level which must not be exceeded - software limit to restrict the traversing range.
Max. level (mm)	The maximum syringe level which must not be exceeded - software limit to restrict the traversing range.
Max. Pressure (bar)	Maximum syringe pressure – important for pressure monitoring
Suited for pump	Shows for which pump a syringe is suited. If this field is empty, it is a syringe that was created by the user.

7.4.5 Configuring the syringe parameters

If you click the *Create* or *Edit* button, the (*Syringe Configuration Wizard*) for configuring the syringe parameters opens. Please configure a syringe with the following steps:

(1) First assign a unique name to the syringe in the field *Unique syringe name* 1 (Figure below). If you have selected a syringe for editing and only change its name, you can generate a copy of the selected syringe. Then select the syringe material 2. Click *Next* to go to the next step.

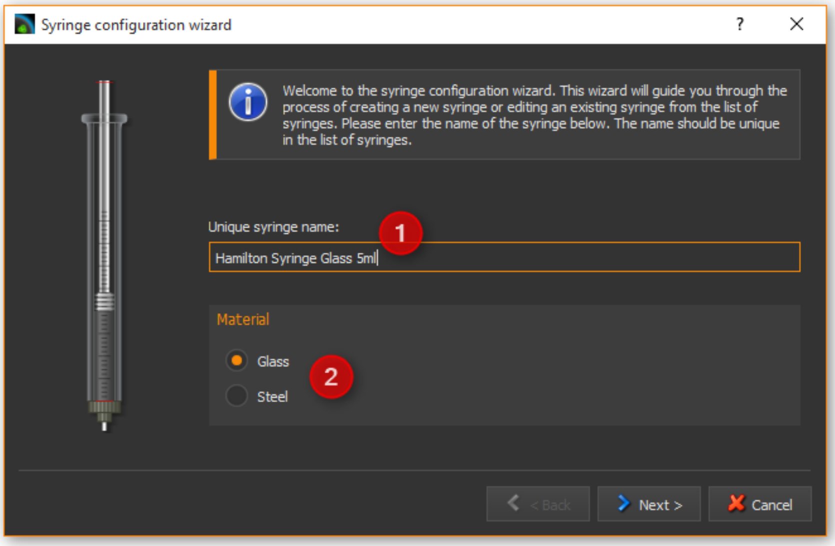


Figure 100: Syringe configuration step 1 - Enter name

(2) You should now configure the inner diameter of the syringe. The inner diameter of the syringe is required in order to calculate flow rates and dose quantities. If you do not know the inner diameter, you can alternatively define the inner diameter via the *Scale length* in mm, and the volume it contains via *Scale volume* in µl. The software then uses these values to calculate the inner diameter of the syringe. In this step, select how you want to configure the inner diameter of the syringe. (Figure below). Depending on the selection, you continue the configuration with point **(3)** or **(4)**. Click *Next* to go to the next step.

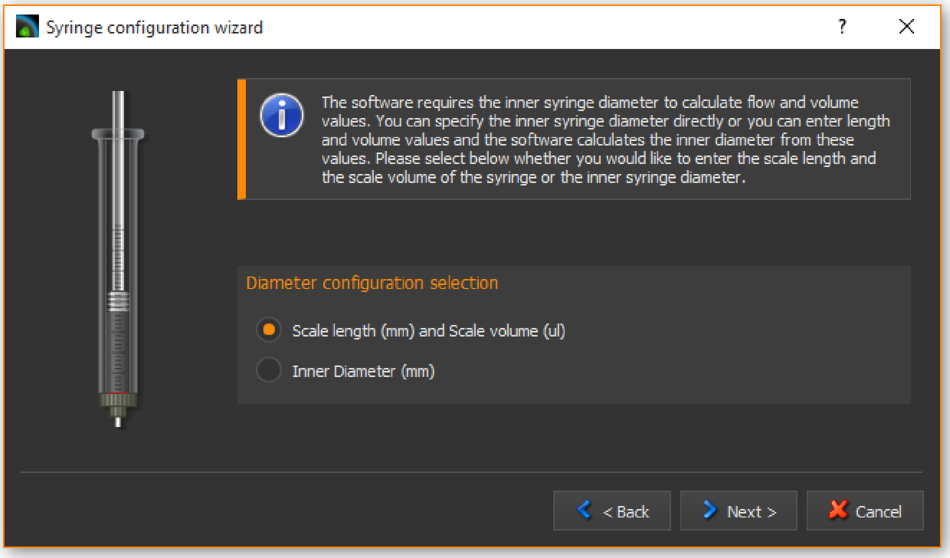


Figure 101: Syringe configuration step 2 - Select method

(3) You configure the inner diameter of the syringe by entering the scale length and volume within this scale length. (Figure below). To do this, measure, the scale with a suitable measuring instrument (calliper gauge), and read the volume shown on the inscription on the scale. Then click *Next* to go to the next step, and read further at point **(5)**.

(4) You configure the inner diameter of the syringe by measuring it with a calliper gauge. Then enter the value in the *Inner Syringe Diameter* field, and click *Next* to go to the next step.

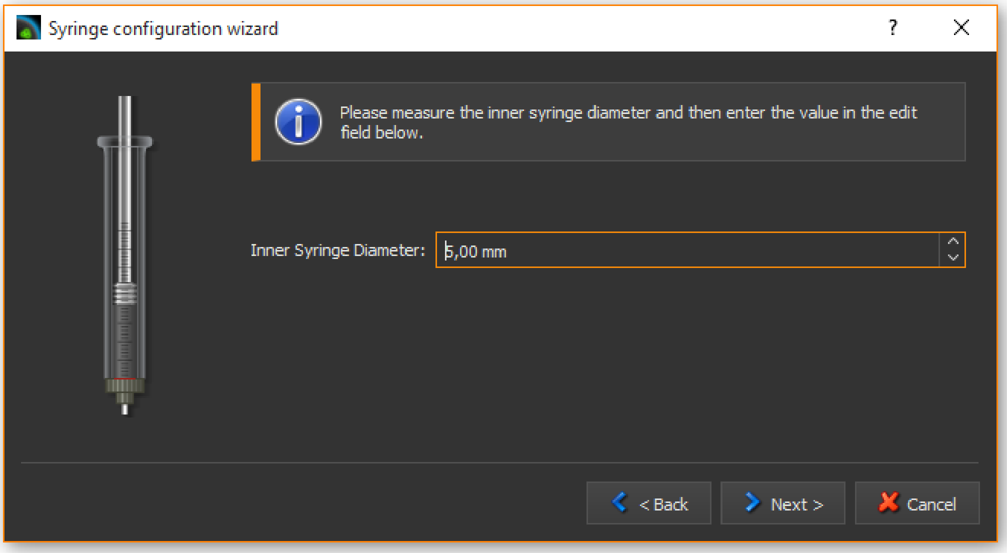


Figure 102: Syringe configuration step 4 - Enter inner syringe diameter

(5) Now configure the piston stroke (*Piston Stroke*) of the syringe (Figure below). The piston stroke is the maximum length through which the piston can be moved in the syringe without it leaking and liquid emerging. It is advisable to use the length of the scale on the syringe as the maximum piston stroke.

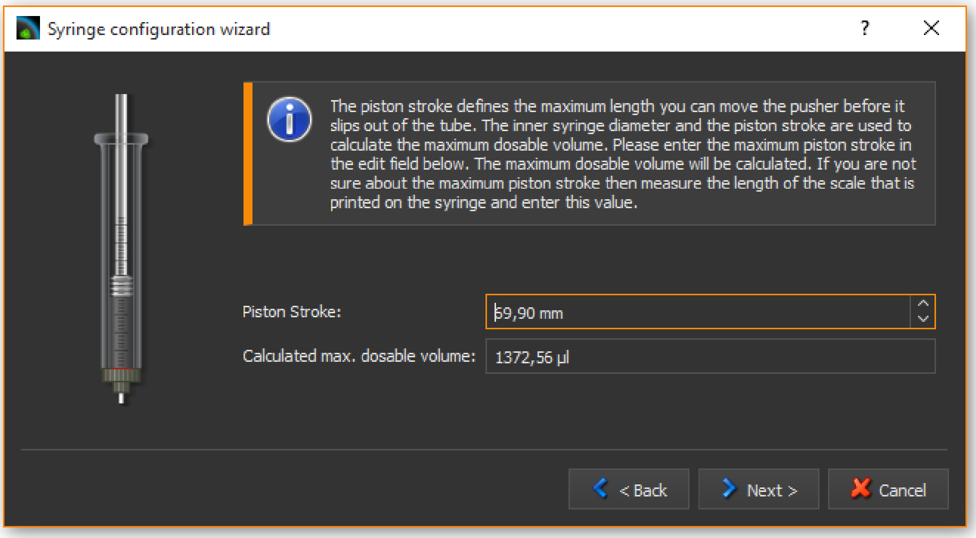


Figure 103: Syringe configuration step 2 – Piston stroke

If syringes of different lengths are used in a multi-syringe holder, the syringe with the shortest piston stroke limits the range of travel of the drive. When you have entered the piston stroke, the software uses this and the inner diameter to calculate the maximum dosable volume.

(6) In this step you can define the limits for the maximum and minimum syringe fill levels. (Figure below). In this way, you can limit the range of travel of the piston by two additional software limits. You can enter these parameters optionally in microlitres or millimetres. You should use these values to limit the range of travel, for example when using a syringe stirrer. If you enter the value in one unit (e.g. millimetres), the value in the other unit (e.g. microlitres) is calculated by the software.

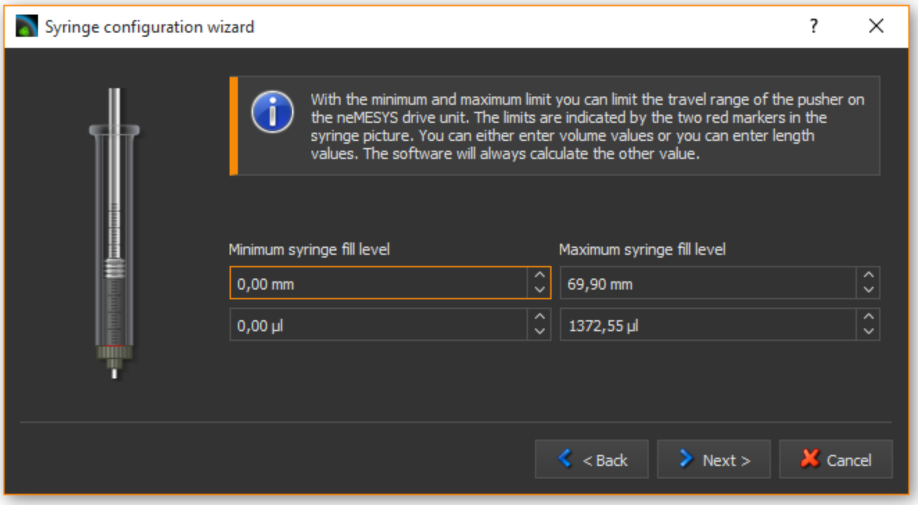


Figure 104: Syringe configuration step 6 - Limits

The syringe preview on the left-hand side always shows you the current configuration of the syringe. The width of the syringe changes when the inner diameter changes. The length of the syringe changes when the maximum piston stroke is changed, and the two red marks on the syringe indicate the two additional limits. The syringe is subsequently also shown corresponding to this preview in the software.

(7) Finally enter the maximum pressure the syringe can be used with. This value is the maximum value you can use for pressure monitoring. (see section 7.13.Pressure- / Analog-In-Monitoring)

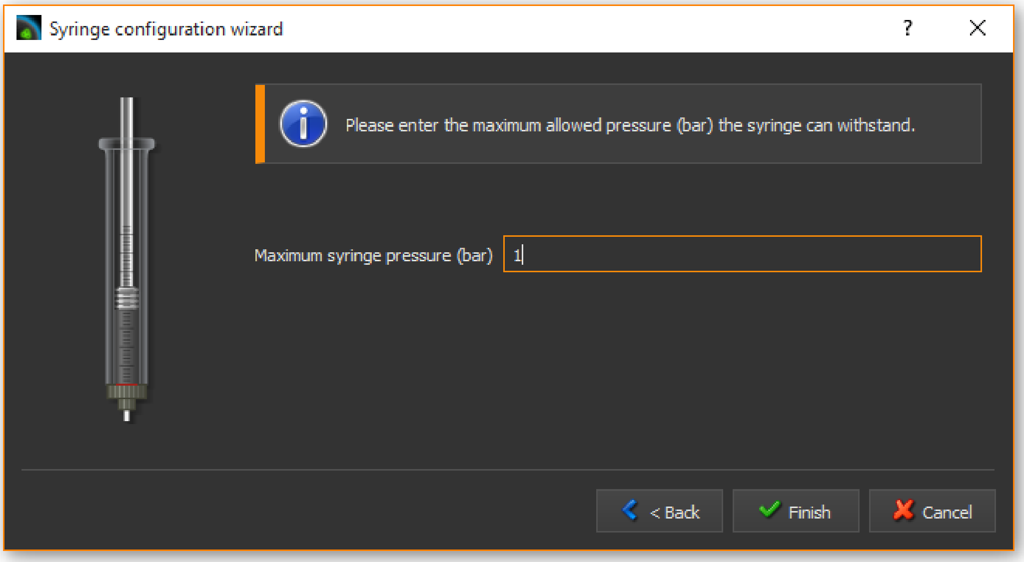


Figure 105: Syringe configuration step 7 - maximum pressure

(8) Close the configuration of the syringe parameters by clicking the *Finish* button. You can cancel the configuration at any time by pressing the *Cancel* button. You can always return to the previous configuration step with the *Back* button.

7.5 Configuration of SI units

When dosing liquid flows, the main parameters are the flow rates used and the quantities of liquid to be dosed or already dosed. For optimal adaptation to the particular application, the user can configure the SI units individually for displaying the flow rates and volumes for each individual dosing unit. Proceed as follows to configure the units:

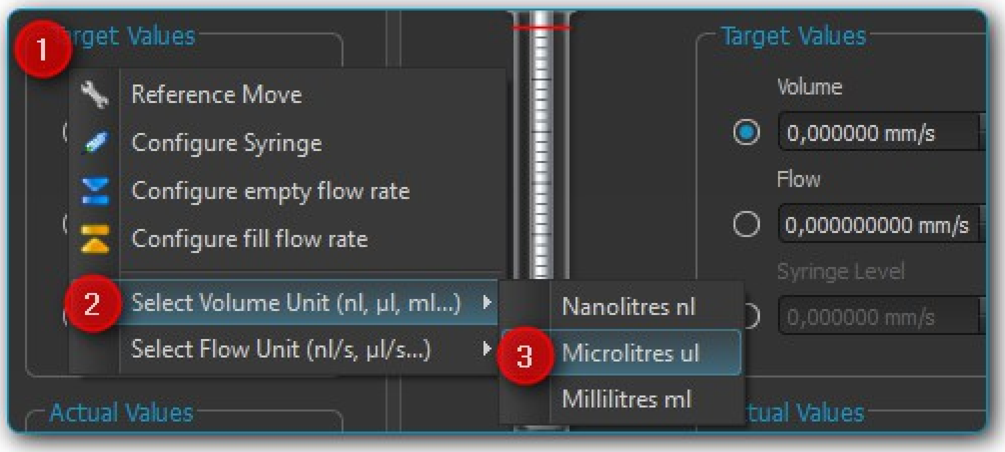


Figure 106: Selection of SI units

1 Please make a right mouse click on an empty area in the operating panel to open its context menu.

2 Select Select *Volume Unit* or Select *Flow Unit* menu item to open the units submenu.

3 Then select the desired unit with a click.

7.6 Carrying out Reference move

The drives of the dosing units are monitored and controlled by a digital positioning unit. You can make a reference move of the dosing unit to calibrate this positioning unit. A suitable time for this is, for example, when a syringe is changed, because the syringe has to be removed from the dosing unit in order to make a reference move. During the reference move, the dosing unit travels to its lower limit position, and calibrates your zero position when it reaches the limit position.

IMPORTANT

Always make a reference move if you operate the pumps from another PC, because the calibration data are stored on the PC and *not in the dosing units.*

To start a reference move, make a right mouse click in the operating panel of the dosing unit that is to be calibrated. Select the *Reference Move* menu item in the context menu that will open now.

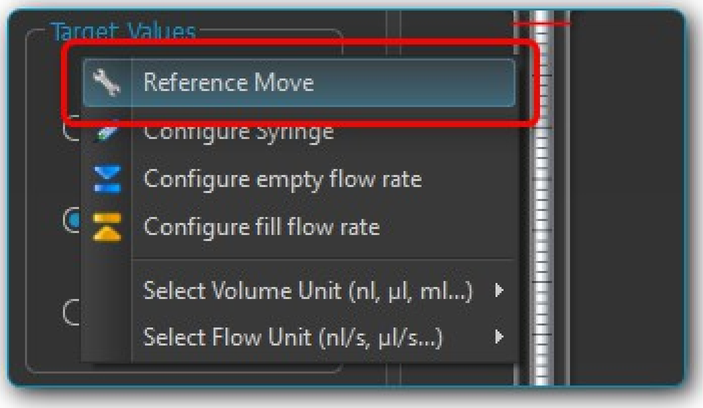


Figure 107: Start reference move

ATTENTION

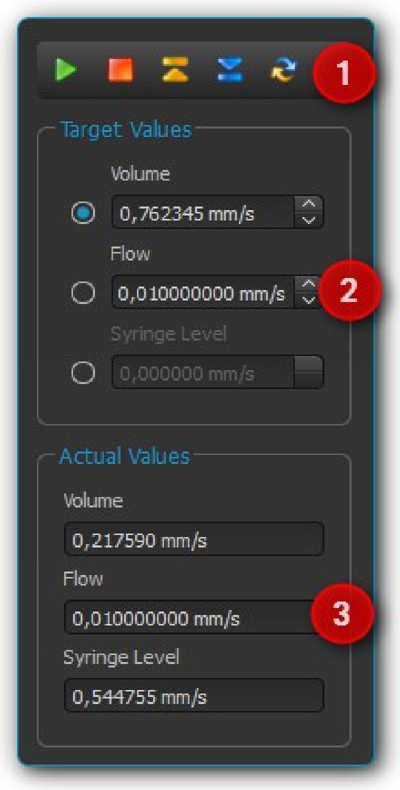
Danger of damaging syringes! The system may only be calibrated when a syringe is not installed in the dosing unit.

ATTENTION

Risk of high pressures damaging the connectors in the liquid path. To avoid high pressures causing damage, never make a reference move with a high-pressure syringe clamped.

7.7 Direct Control

7.7.1 Overview



Use the direct control for interactive control of the dosing units. In this way, you dose precisely defined quantities of liquid with defined flow rates or generate constant liquid flows.

The direct control also displays the current flow rate, the dosed volume and the fill level of the syringe. If you use a high-pressure module, the current pressure measured by the pressure sensor is also displayed.

Following controls and indicators are available:

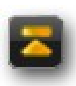
1 Toolbar
2 Target values of the dosing module
3 Actual values of the dosing module

7.7.2 Toolbar

The toolbar includes buttons for starting dosing processes. The following buttons are available:

 Start the dosing with set target values

 Stop dosing

 Aspirate reagent - Move dosing unit to maximum position

 Empty syringe - Move dosing unit to minimum position

Configure the continuous flow

ATTENTION

Danger of damaging the syringe during emptying! When the syringe is emptied, according to the configuration, the dosing unit is moved toward the lower limit position at high speed.

7.7.3 Target value

7.7.3.1 Selecting dosing mode

You define the type of dosing with the selection buttons on the on the left-hand side.

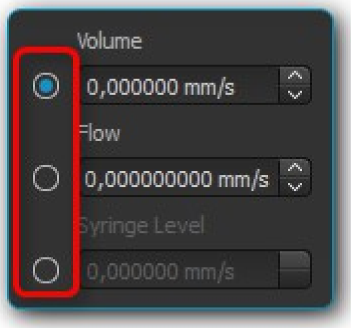


Figure 108: Select dosing mode

The following options are available:

- *Volume* - Select this mode to dose a specific volume at a defined flow rate
- *Flow* - In this mode, you generate a constant flow of liquid. The dosing continues until a limit position is reached or until you stop the dosing process.
- *Syringe Level* - In this mode, you can define a specific fill level for the syringe which is to be reached at a defined flow rate.

7.7.3.2 Setting flow rate and volume

Use the *Volume*, *Flow* and *Syringe Level* input fields in order to enter the flow rate, the flow volume or the syringe level. Setting the setpoints does not start the drives or change the current flow rate. The new values are not transferred to the dosing unit until a dosing process is started by pressing the *Start* button in the toolbar.

7.7.3.3 Volume dosing

If you want to dose a specific volume, you have to enter the volume to be dosed and the flow rate. The volume is entered relative to the current position of the syringe piston. This means that you enter a negative volume to aspirate reagent and a positive volume to dispense reagent. In this operating mode, the flow rate is always a positive value, and defines the flow rate of the aspiration/dispensing of reagent.

7.7.3.4 Constant flow rate

In order to generate a constant flow of liquid, only the flow rate has to be defined, and the volume input field is disabled for entries. In this operating mode, the flow rate can be either positive or negative. A negative value indicates reagent aspiration, and a positive value reagent dispensing.

7.7.3.5 Setting syringe level

You can achieve a precisely defined syringe fill level by entering the values for the syringe fill level and the flow rate. The syringe fill level must be a positive value lying between the minimum and maximum values of the syringe fill level. (see section Fehler: Referenz nicht gefunden Fehler: Referenz nicht gefunden). In this operating mode, the flow rate is always a positive value, and defines the flow rate of the aspiration/dispensing of reagent.

ATTENTION

Danger of damaging the valve or connections in the liquid path High flow rates can cause high pressures to develop very quickly, which can damage the valve, connections in the liquid path or your application.

IMPORTANT

For dosing very low flow rates, select syringes with small inner diameters in order to ensure pulsation-free dosing.

7.8 Actual Values

The current values reported by the device are shown in the *Actual Values* area.

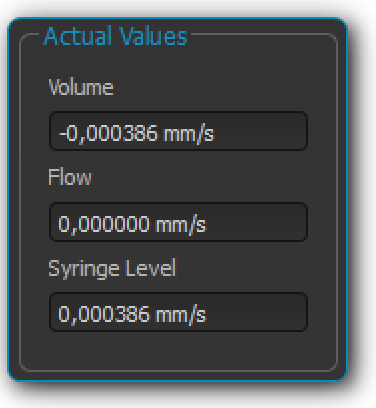


Figure 109: Actual values

The *Flow* field shows the current flow rate in the dosing unit. The *Volume* field shows the volume that has been pumped since the last time the dosing unit was started. If the drive is stopped and restarted, the actual value of the pumped volume is reset to 0. The *Syringe Level* field shows the current fill level of the dosing unit in the configured SI volume unit.

7.9 Continuous Flow

7.9.1 Introduction

In Continuous flow operating mode, you can link two dosing units to each other in the software in order to generate a continuous flow over a long period of time by means of alternating aspiration and dispensing of reagent. In this case, one of the two pumps doses into the application at a specific flow rate, while at the same time the other pump is aspirating reagent from the reservoir. When the dosing pump has emptied the syringe, the software automatically switches to the second dosing unit with the full syringe, and continues the dosing process with this pump. This creates a continuous, uninterrupted flow, so you can dose at a constant flow rate for an almost unlimited period of time.

7.9.2 Controls and indicators

If continuous flow has been configured for a dosing unit, you will see that the button bearing the continuous flow icon is pressed in the direct control area. (Figure below).

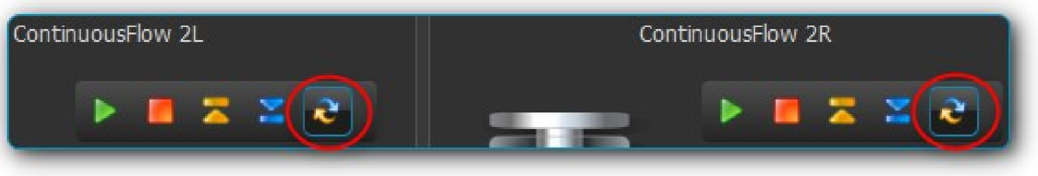


Figure 110: Continuous flow mode indicator

Continuous flow operating mode is ended automatically as soon as you change the syringe configuration of one of the two linked dosing units. In this case, you have to reconfigure all the parameters. When continuous flow is active for a dosing unit, the *Continuous flow* panel is shown in the operating panel to indicate continuous flow status. (Figure below).

Figure 111: Continuous flow state indicator

The following display elements are then visible:

- *Remaining time* - The remaining time indicates the time remaining for continuous dosing. The continuous flow is stopped automatically when this indicator reaches zero.
- *Accumulated volume [ml]* - The accumulated, dispensed volume indicates the total of the volumes which have so far been pumped into the application by both pumps.
- *Linked pump* - The linked pump is the second pump that is linked to this pump to provide the continuous flow.

7.9.3 Configuring continuous flow

In order to generate a continuous flow, you need at least two neMESYS dosing units with identical configurations. Both units must have a valve. In order to configure the continuous flow, make a right mouse click in the operating panel of the direct control, and select Configure continuous flow in the context menu (Figure below).

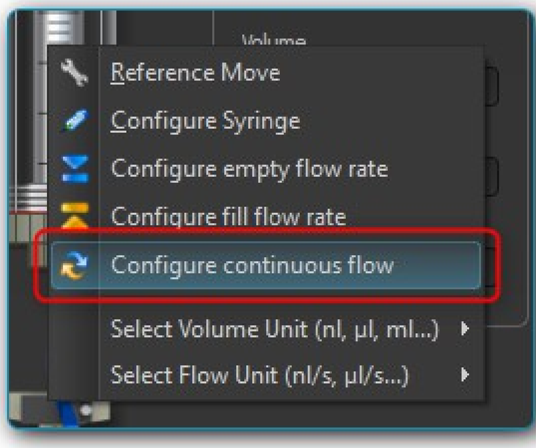


Figure 112: Configure continuous flow

A configuration dialog appears: The operating panel of the dosing unit in which you call the configuration of the continuous flow is that of the first of the two dosing units that are linked to one another. If continuous flow has not yet been configured for the dosing unit, you can also open the configuration dialog by pressing the *Continuous flow* button in the direct control (Figure below).

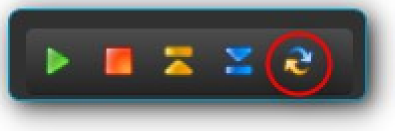


Figure 113: Direct opening of the configuration dialog

7.9.4 Selecting second dosing module

On the first page of the configuration dialog, select the second dosing unit for the continuous flow from the list of available dosing units (Figure below). Select the dosing unit by clicking the name in the list. Then click the *Next* button to continue with the configuration.

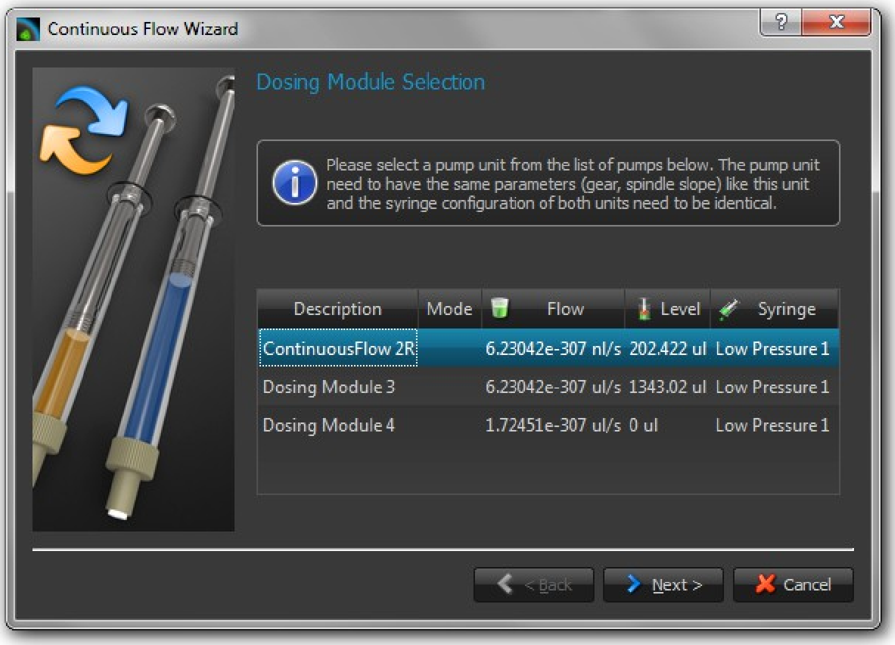


Figure 114: Select second doing module

IMPORTANT

You should always select two dosing units for the continuous flow which are linked directly to each other and are shown in the software alongside each other.

7.9.5 Choosing the continuous flow mode

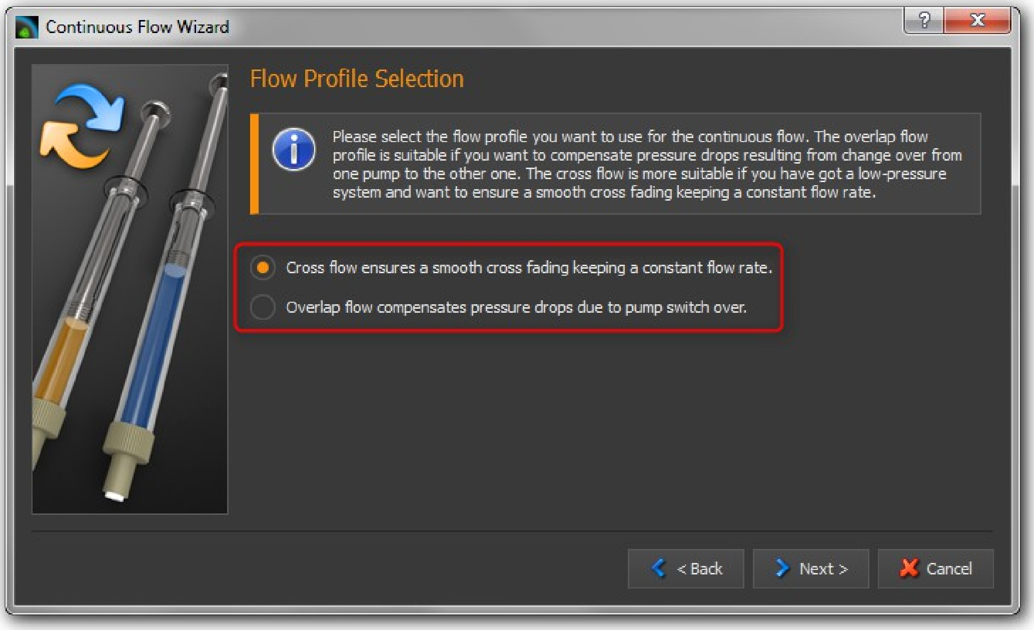


Figure 115: Choosing the continuous flow mode

The continuous flow can be used with the following modes:

- Cross flow
- Overlap flow

Operating in cross flow mode you get a constant flow rate and a smooth crossfading of pump flows on switch-over from one pump to the other one. This mode is applicable if you work against a low fluidic system pressure. Operating in overlap mode you can configure the pumps in a way that one pump is still dosing using a certain flow rate while the other one has already overtaken the continuous flow. This allows compensating a fluidic system pressure drop resulting from pump switch over. In the following sections the different modes are explained in detail. Left-click the button of the mode you want to use and click *Next* in order to continue.

7.9.6 Configuring flow parameters

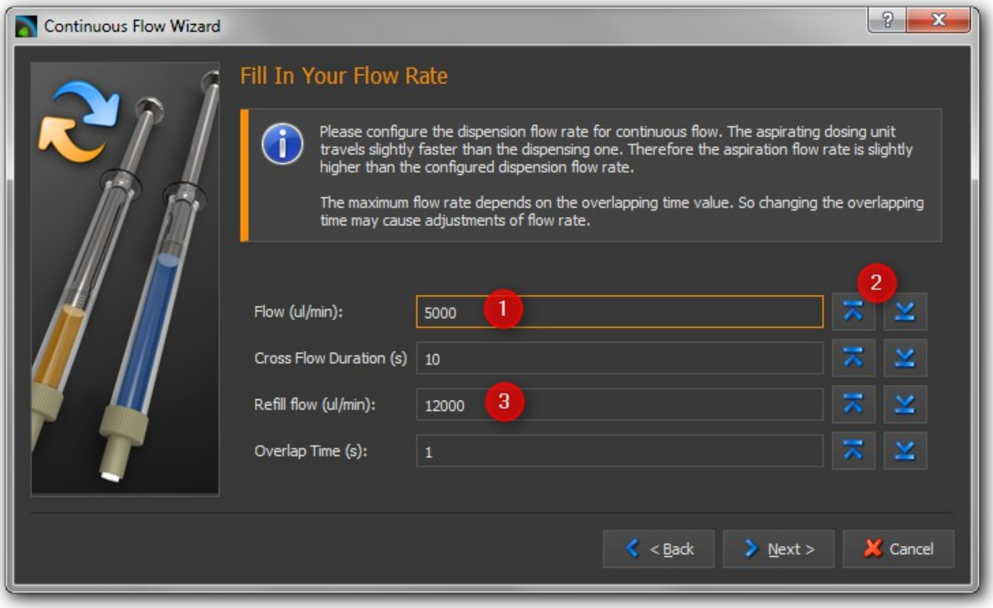


Figure 116: Configure continuous flow parameters

In the next step, you configure the flow rate for the continuous flow. The SI unit of the flow rate is the unit configured in the operating panel of the dosing unit with which you started to configure the continuous flow. Enter the flow rate in the input field *Flow* 1 (Figure above). You can set the flow rate to the minimum or maximum value by pressing the *Min* and *Max* buttons 2 respectively. The *Refill flow* 3 field displays the flow rate which is used to fill the syringes. This flow rate is always slightly larger than the flow rate in the *Flow* field 1 since the syringes have to be filled faster to be ready in time for the next dosage. The minimum value of the input field *Refill flow* 3 is the minimum flow rate required by the system to operate the continuous flow properly. You can select a higher flow rate, if you want a faster refill process. This may be necessary if you work with back pressure valves that require a certain minimum switching pressure and thus a certain minimum flow rate to generate this pressure.

7.9.7 Configuring the cross flow parameters

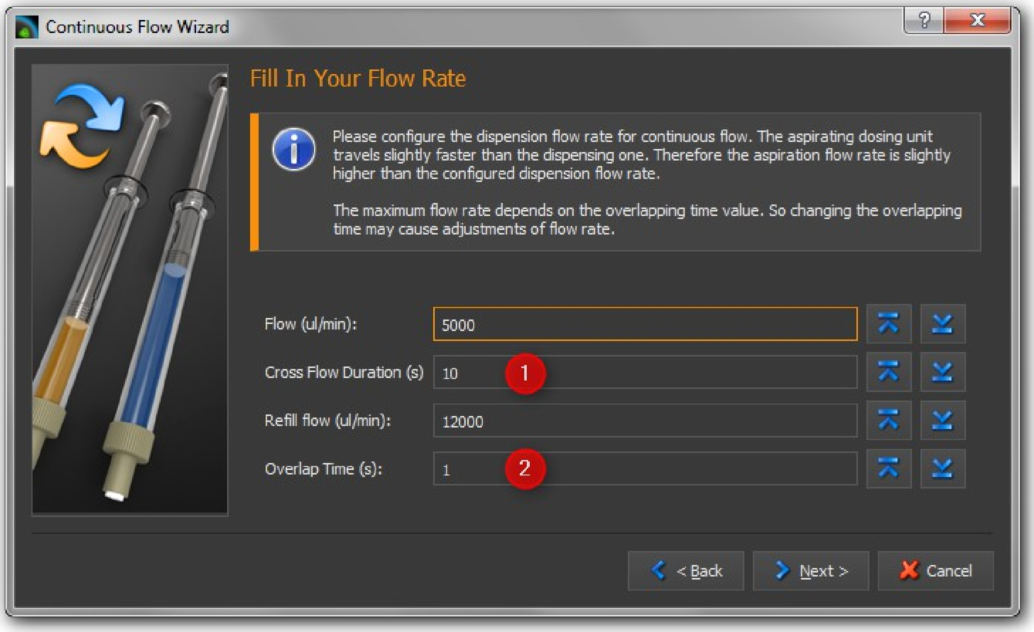


Figure 117: Configuring the cross flow parameters

The cross flow allows a smooth crossfading of the both particiapting pump flows by configuring the cross flow duration 1 (see figure above).

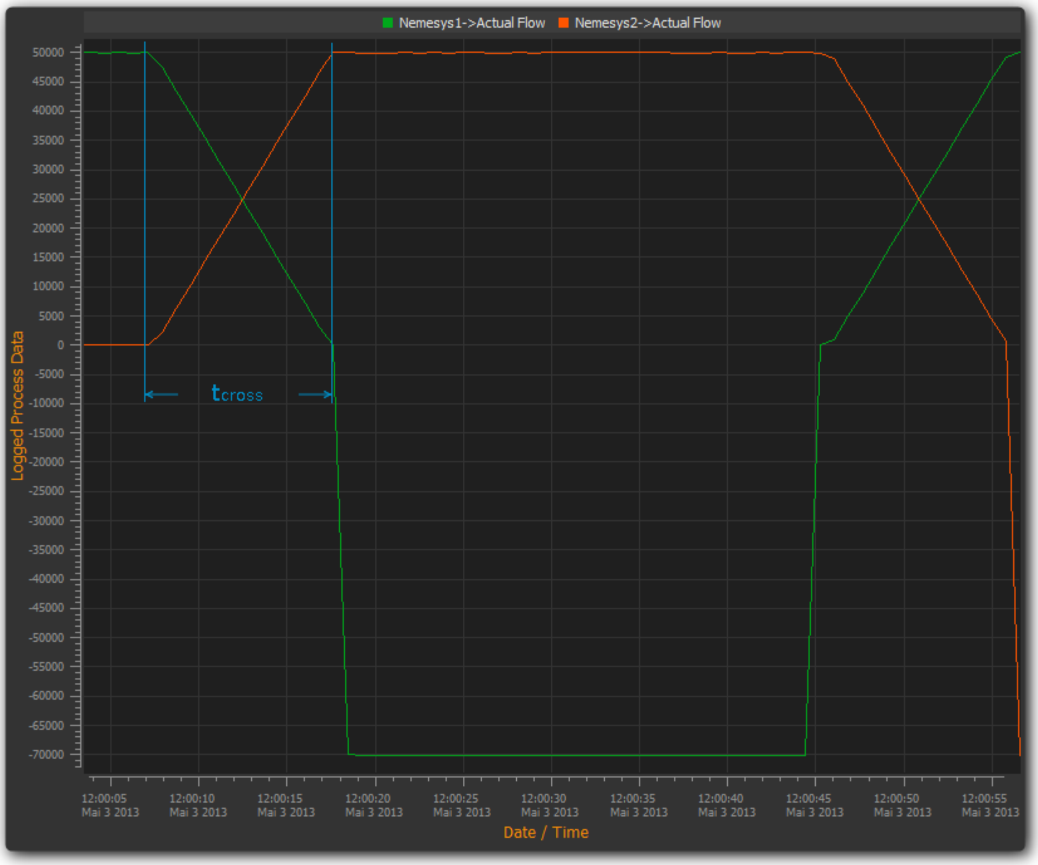


Figure 118: Cross flow profile

The *cross flow duration* t_{cross} determines the duration of crossfading the flow from one dosing module to the other one. The smaller t_{cross} the steeper the flow ramp runs and the faster flow crossfading is finished (see figure above).

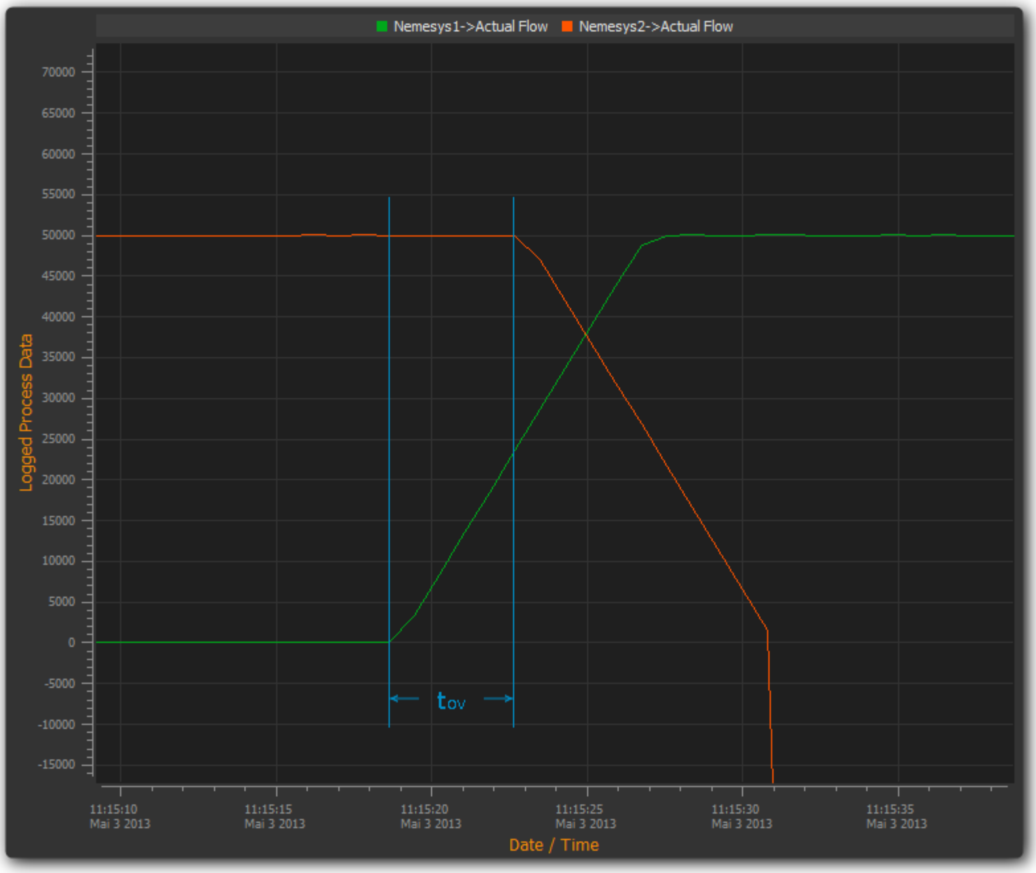


Figure 119: Overlap Duration

The cross flow offers a simple variant of pressure drop compensation. The overlap time 2 t_{ov} determines the duration the flow curves of both dosing modules are shifted into each other. The bigger t_{ov} the longer both dosing modules dose at the same time (see figure above).

7.9.8 Configuring the overlap flow parameters

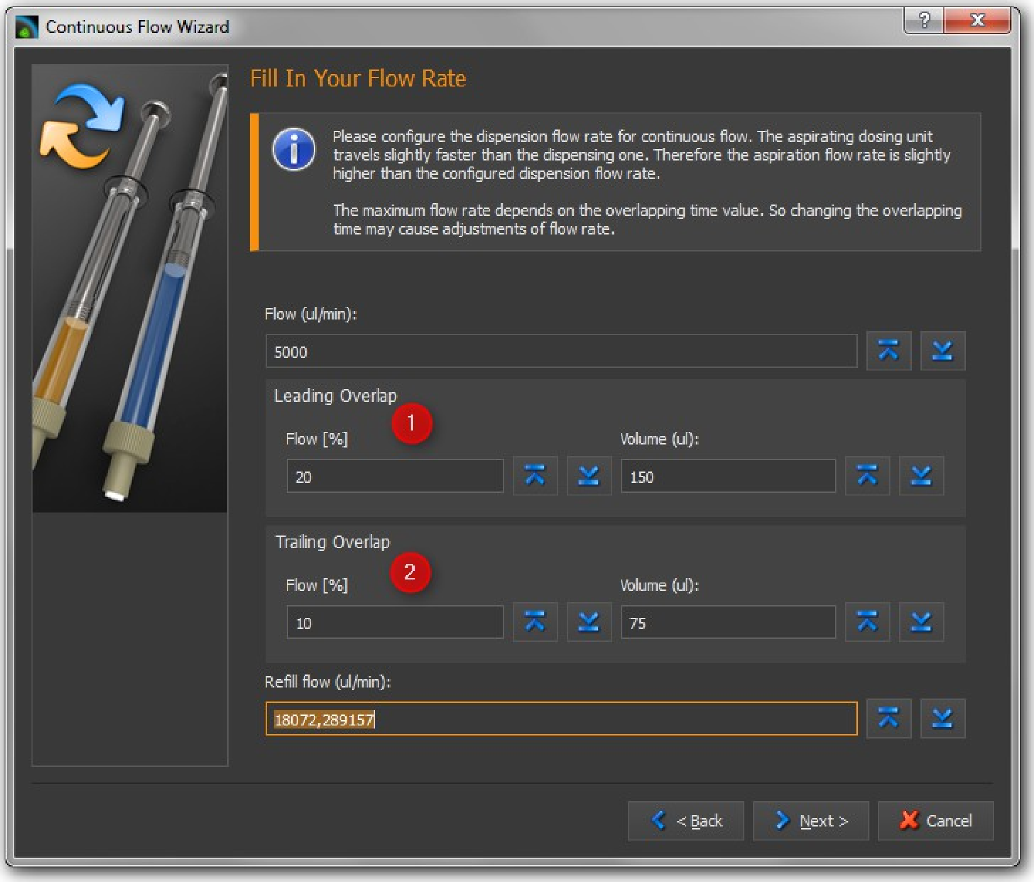


Figure 120: Configuring overlap flow parameters

In mode overlap flow you can compensate a pressure drop resulting from pump switchover by configuring a leading 1 and a trailing 2 overlap. The overlap volume is specified by absolute value, the overlap flow as percentage from dosing flow.

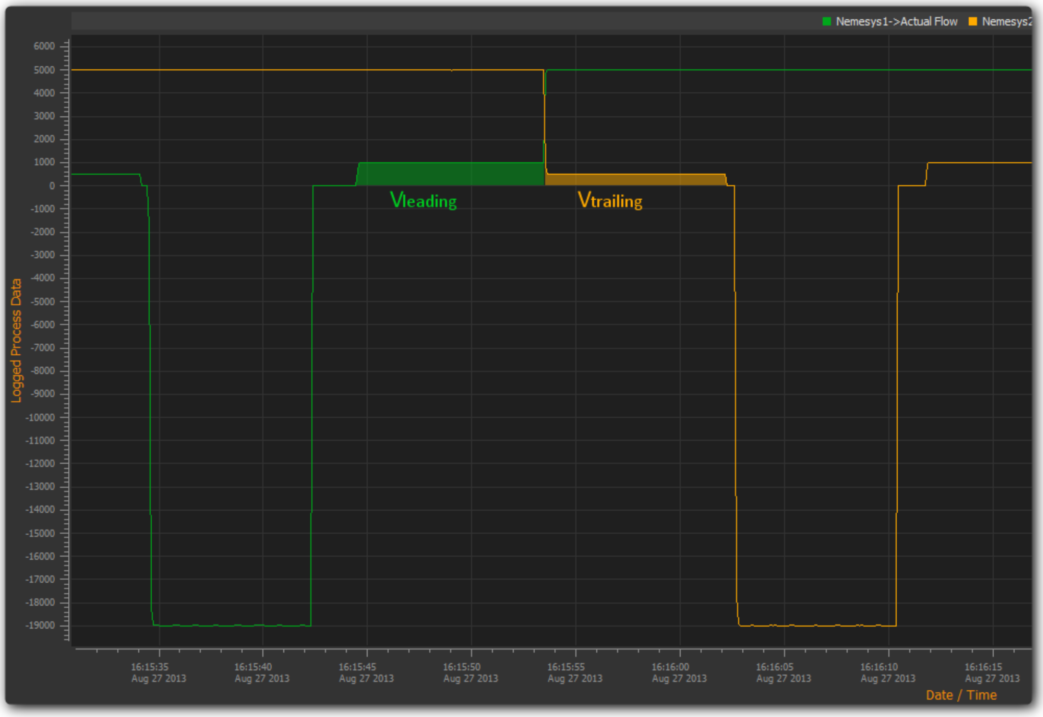


Figure 121: Overlap flow profile

The leading overlap is the volume that is dosed by the second pump before the first pump has finished dosing. The trailing one is the volume the first pump is dosing after the second pump has taken over. (see figure above)

7.9.9 Continuous flow duration

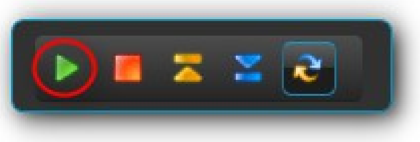
You can limit the duration of the continuous flow in this window (Figure below). If you do not want to limit the duration, leave the selection at the default value *Dose unlimited*. With this setting, the continuous flow continues until the dosing is stopped manually.

The software offers 3 options of limiting the duration of the continuous flow.

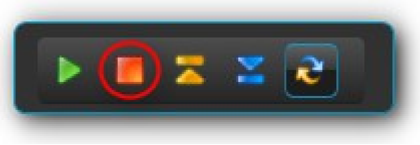
- *Dose total volume* - The continuous flow is stopped as soon as a defined total volume has been dosed.
- *Dose for* - The continuous flow ends after a set period of time.

- *Dose until* - The continuous flow ends at a specific time in the future.

7.9.10 Starting / Stopping continuous flow



After the configuration, start the continuous flow by a left mouse click on the *Start dosing* button in the direct control



You can interrupt the continuous flow by pressing the *Stop* dosing button

When one of the dosing units in the continuous flow is stopped, the linked dosing unit is also always stopped. If you have configured the continuous flow for a specific period of time, stopping the dosing unit interrupts this period. This means that when you restart the dosing unit, the continuous flow resumes with the period of time still remaining at the time of the interruption. If you have configured a total target volume for the continuous flow, the volume remaining at the time of the interruption is dosed as from the restart.

7.9.11 Interrupt/restart continuous flow

The continuous flow ends when the time set for the continuous flow has expired or the total target volume has been reached, and both linked dosing units are stopped. By left-clicking the *Start* button 1 a finished continuous flow can be restarted. In this case the duration or total target volume are reset to their initial values. You can leave the continuous flow mode and abort the running continuous flow by left-clicking the *Continuous flow* button 2. In this case a restart is not possible. (Figure below)

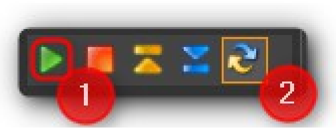
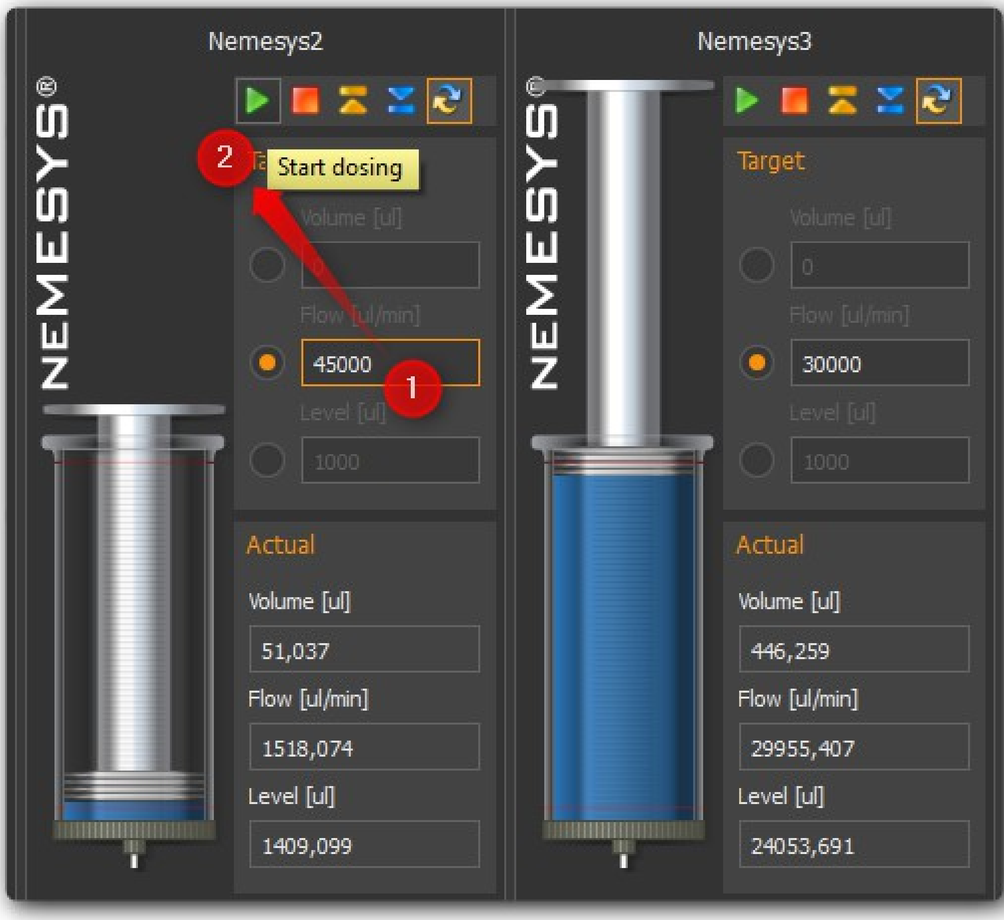


Figure 122:Restart or abort continuous flow

7.10 Changing the flow rate

You can change the flow rate of a running dosing task in all modes of operation. Enter the desired flow rate into the flow field and confirm by left-clicking the *Start* button. (see figure below)



The flow rate is accepted by the dosing module, all other parameters like dosing volume or continuous flow parameters remain unchanged. During configuration of a continuous flow the maximum flow rate that can be dosed with using the current refill flow rate and the other continuous flow parameters is displayed to you. (see figure below)

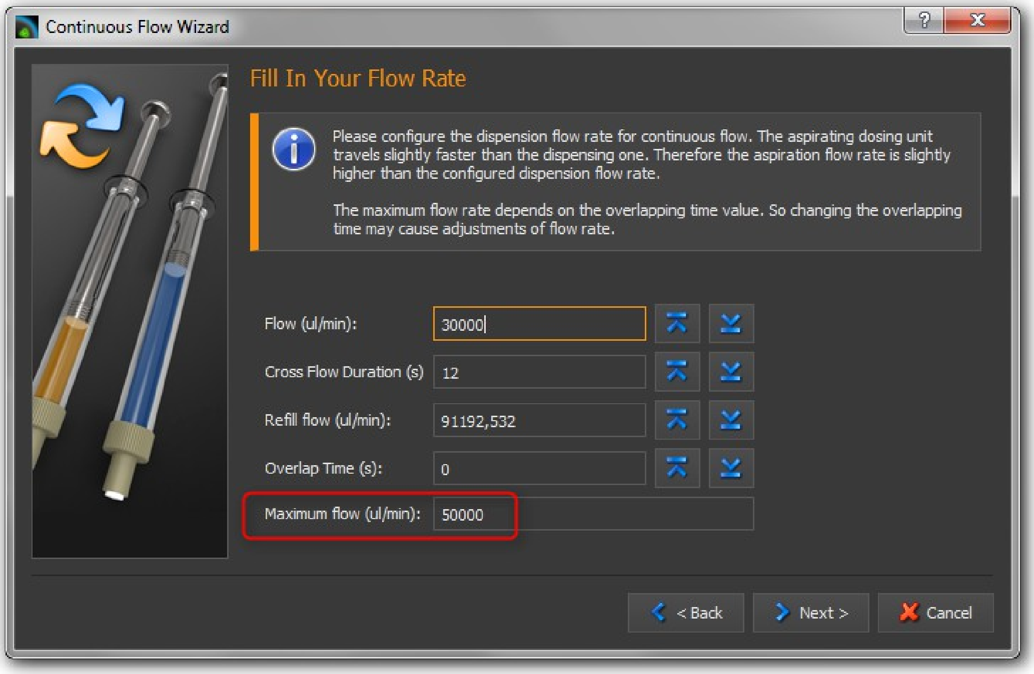


Figure 123: Maximum continuous flow rate

7.11 Synchronous pump start / stop

In the main toolbar, you will find two buttons for simultaneously start / stop multiple pump modules.

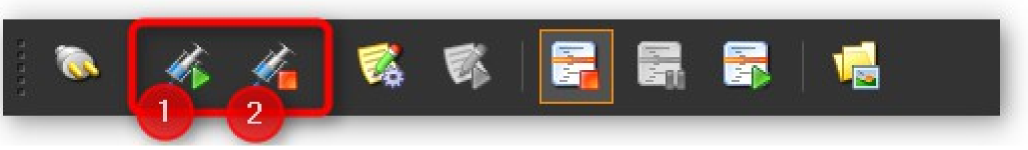


Figure 124: Synchronously pump start / stop

When you click the *Start* button 1, a selection dialog appears in which you select the pumps that you want to start at the same time (see figure below).

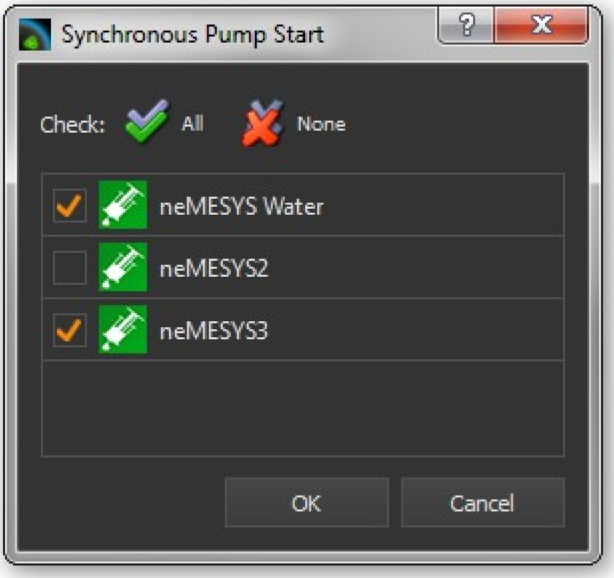


Figure 125: Pump selection for synchronous start

Set a checkmark for each pump you want to start. Dosage starts as soon as you klick the *OK* button.

IMPORTANT

Before starting, you need to configure the dosing parameters (volume, flow) of all selected pumps in the control panel of each pump.

Before the software starts the dosage, it checks the configured parameters of all selected pumps. If the software found invalid parameters (eg flow or volume values of 0), the synchronous start is not performed. In the application *event log* you will get information about the problems encountered.

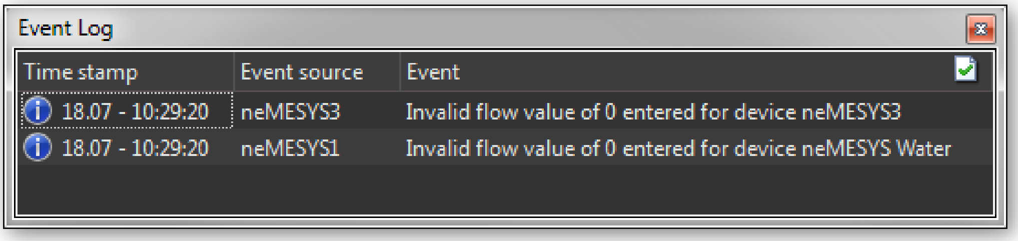


Figure 126: Sync Start errors in the event log

When you click the *Stop* button 2, all pumps are stopped immediately.

7.12 Linking pumps to the scripting system

You can link the neMESYS pumps to the scripting system. This way manually stopping the scripting system forces all pumps to stop. This can be achieved by clicking the appropriate button of the main toolbar.

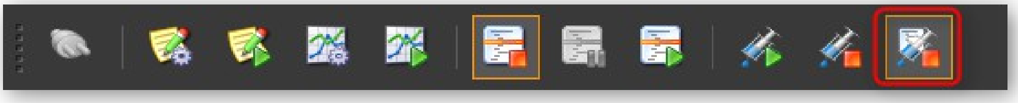


Figure 127: Linking pumps to the scripting system

7.13 I/O Interface

7.13.1 Overview

Various neMESYS pumps offer an I/O interface for connecting external sensors and signals, or for outputting trigger signals (example shown below).

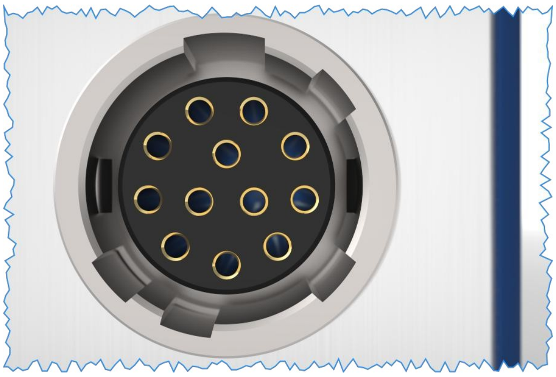


Figure 128: I/O interface neMESYS mid pressure pump V3

Depending on the device configuration the I/O interface features digital inputs and outputs and/or analog inputs. All I/O channels can be found in the software window *I/O Channels*. If this window is not visible, you can show it by clicking the menu item *Window → I/O Channels* in the main menu (see figure below).

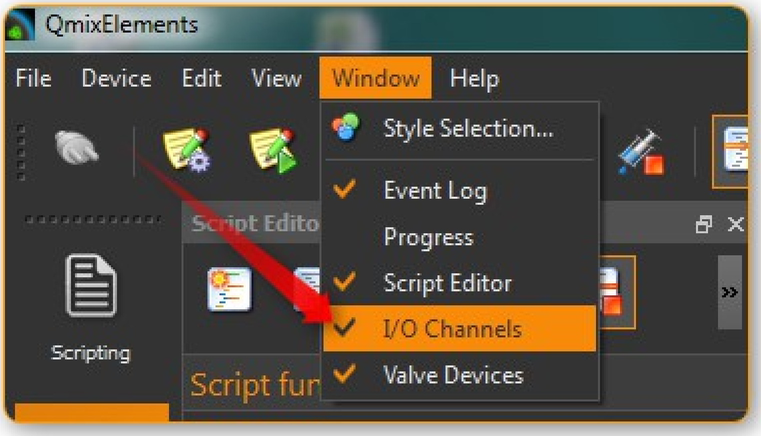


Figure 129: Showing I/O channel window

Each I/O channel has a unique name (e.g. *neMESYS 1 Analog In 1*), which is composed of the pump name (e.g. *neMESYS 1*) and the name of the channel (e.g. *Analog In 1*) like in the figure below.

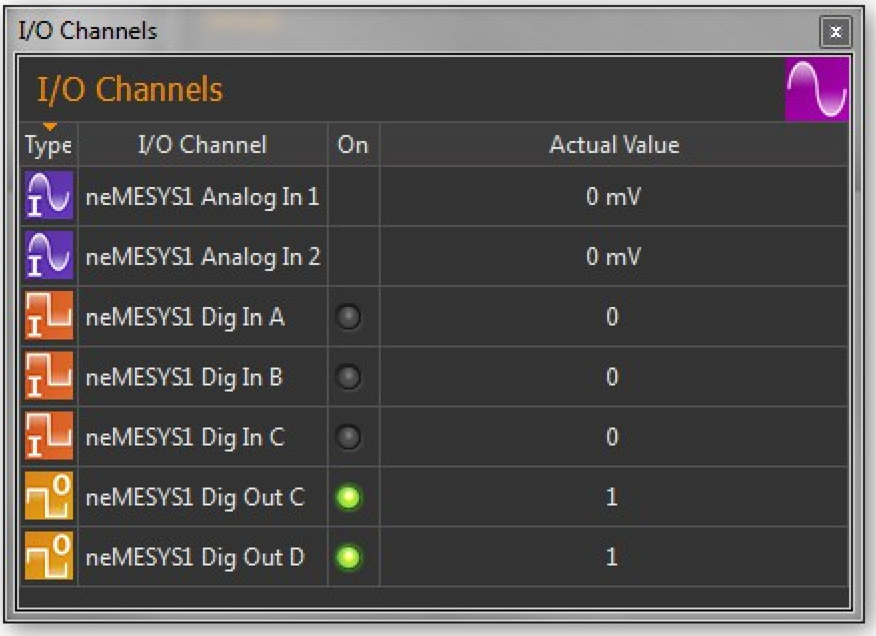


Figure 130: neMESYS I/O channels

7.13.2 Scaling of the analog inputs

The analog inputs of the modules measure the input voltage in the range of 0 - 5000 mV. For each channel, you can specify a separate scaling. This allows you for example to scale the voltage value of 0 - 5000 mV into a pressure value of 0 – 20 bar, if you have connected a pressure sensor. For details on configuring the channels or on scaling, read the documentation of the Qmix I/O plugin.

In the following example, we will show you, how to connect a pressure sensor with the measuring range: 0.5 V - 4.5 V corresponds to 0 - 20 bar. The sensor is connected to analog input 1. We will now define a scaling for this pressure sensor. To do this, click with the right mouse button in the channel row and select the menu item *Configure channel*. The following configuration dialog is then displayed.

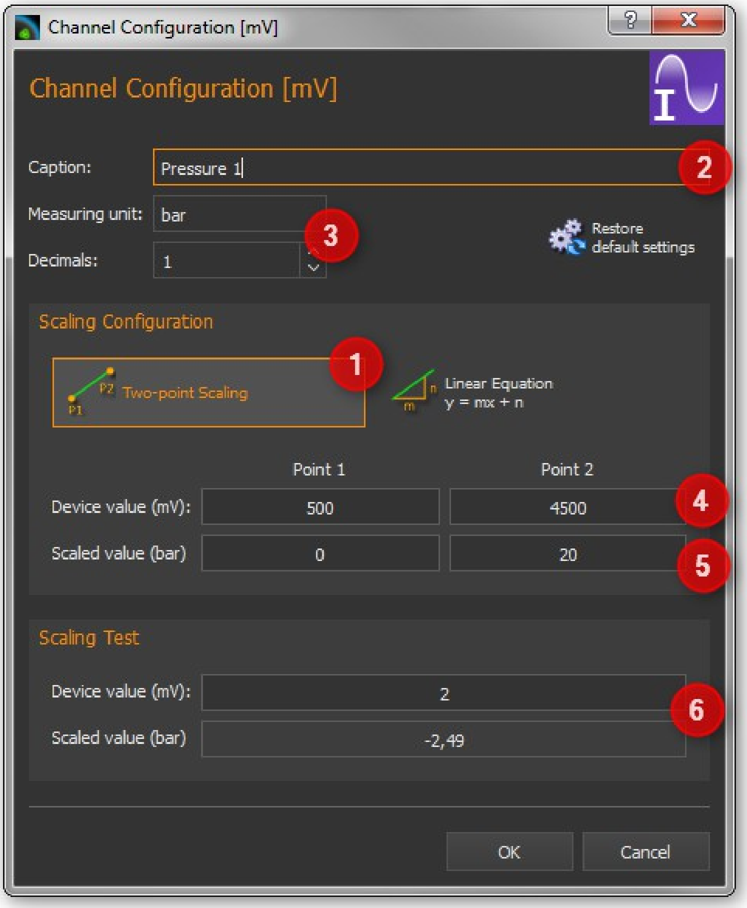


Figure 131: Example: input scaling for pressure sensor

In this dialog, we configure the following values:

- 1 we use a two-point scaling

- 2 we select a meaningful name for the channel: *Presssure 1*
- 3 as unit of measurement we use *bar*
- 4 in the line *Device value* we enter the range of the analog input: *0,5 V – 4,5 V*
- 5 in the line *Scaled value* we enter the measurement range of the sensor: *0 – 20 bar*
- 6 in the area *Scaling Test* we can now test the scaling by manually entering values for the analog input
- by clicking OK we complete the configuration

The measured pressure of the sensor is now displayed in the I/O Channel window in bar (see figure below).

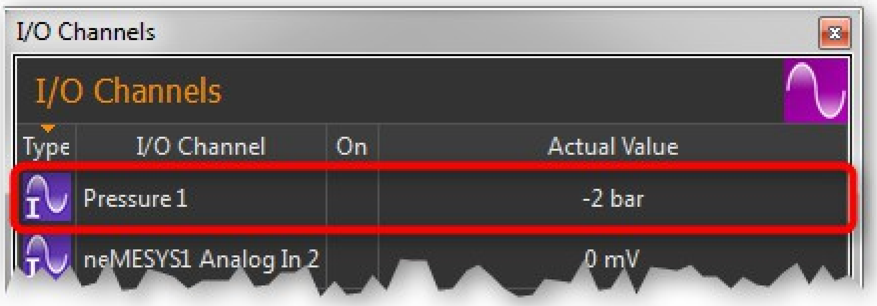


Figure 132: scaled measurement for pressure sensor

HINT

For a detailed description of I/O channels, configuration and scaling, see the Qmix I/O plugin documentation.

7.14 Pressure- / Analog-In-Monitoring

Dosing tasks of neMESYS devices can be monitored by pressure measurement and other analog inputs. To accomplish this you define a safe value range of the analog input and force a certain behaviour of the syringe pump, like stopping the current dosing task, if that range is left. To configure pressure monitoring you need to have a pressure measurement input available. For this purpose you can either configure an analog input of the neMESYS device (see section <<8.5 Predefined configurations>> of the qmixio plugin chapter) as pressure measurement channel or you can use a Qmix p device. Proceed as follows to configure pressure monitoring:

1 Right-click the operating panel of the appropriate dosing module 1 and select *Monitoring* from the context menu 2.

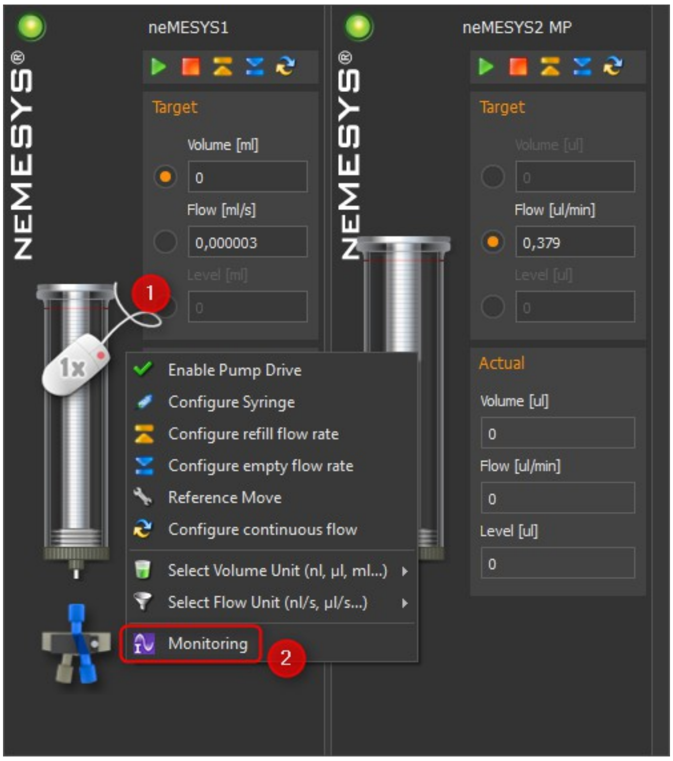


Figure 133: Opening the Analog Channel Monitoring dialog

2 A configuration dialog appears. Select the *pressure* tab to configure pressure monitoring.

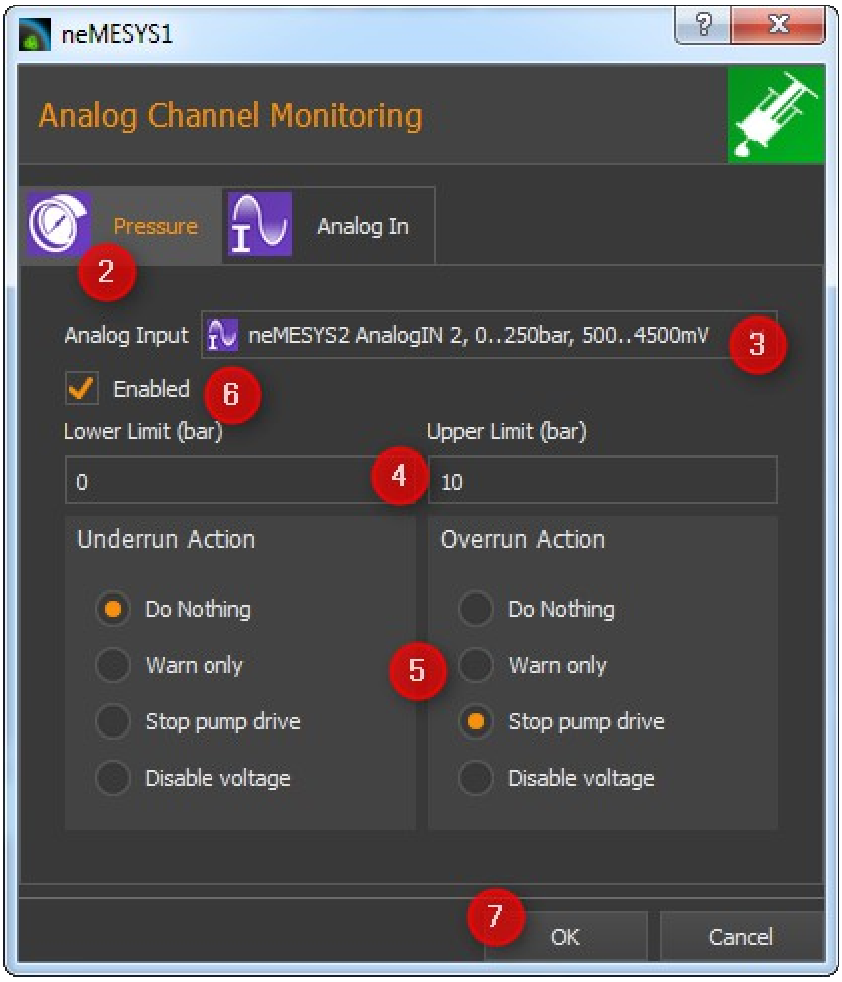


Figure 134: Dialog Analog Channel Monitoring

3 The *Analog-Input*-combobox provides all analog inputs configured as pressure measurement channels available. Select the input you want to use for pressure monitoring

4 Define a safe range by providing a lower and an upper limit. The upper limit value is limited to the maximum pressure of the syringe currently configured for the dosing module.

5 Define what happens if the analog channel value underruns the lower limit or overruns the upper limit by selecting the appropriate button.

6 Check that the *Enabled* button is checked. Otherwise pressure monitoring will be inactive.

7 Accept the configured pressure monitoring by pressing the *Ok* button.

The sections *Overrun* and *Underrun Action* provide the following actions for choice:

- *Do Nothing* – Leaving the safe range of the analog input will be ignored. The neMESYS dosing module continues the running dosing task. The software does neither show a warning nor an information.
- *Warn Only* – A warn message will be shown if the analog channel leaves the safe range. The neMESYS dosing module continues the running dosing task.
- *Stop pump drive* – The running dosing task will be stopped if the analog channel value leaves the safe range. In addition a warn message will be shown.
- *Disable voltage* – The dosing module will be stopped by turning off motor voltage. If the motor is not under voltage the syringe piston can be moved by pressure built up. In addition a warn message will be shown. In order to use the dosing module again it needs to be enabled. (see section 7.14.Enable a disabled pump)

In addition the Analog Channel Monitoring dialog contains the tab *Analog In*. With the help of that one you can configure monitoring functionality using other analog input channels like temperature measurement inputs. The configuration of them follows the same pattern like that one of the pressure monitoring functionality. (see figure below)

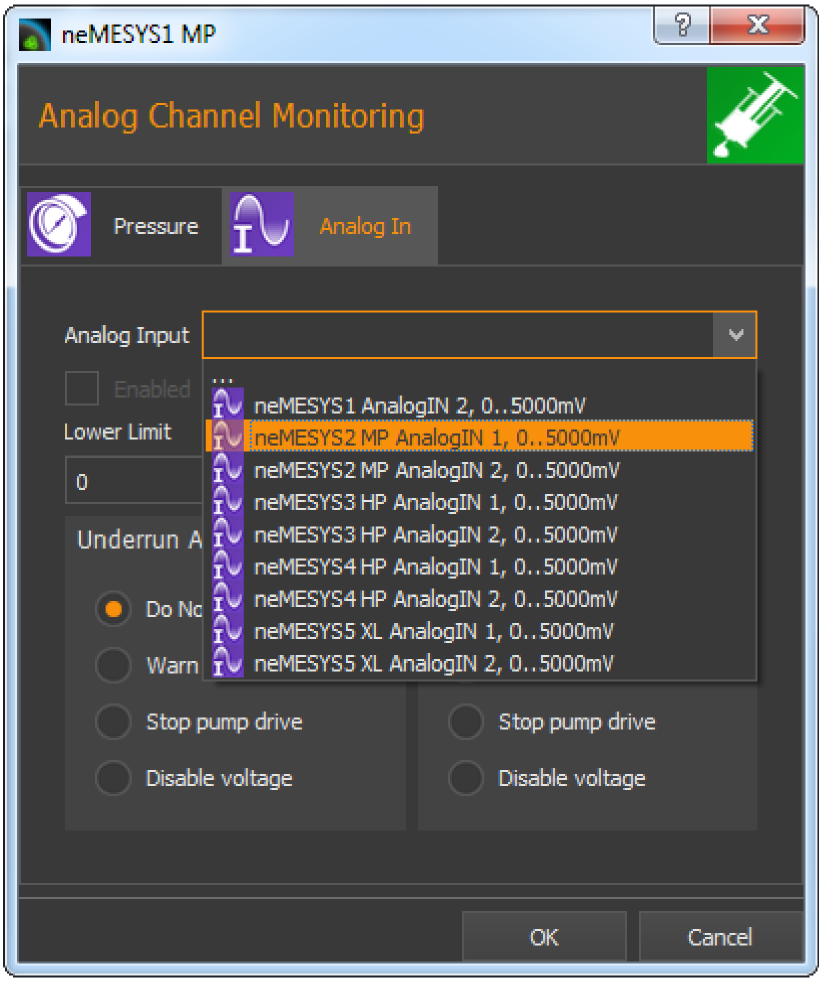


Figure 135: Analog-In-tab

7.15 Enable a disabled pump

If a neMESYS dosing module is disabled by the monitoring functionality, the led on the dosing module operating panel changes from green 1 to red 2.

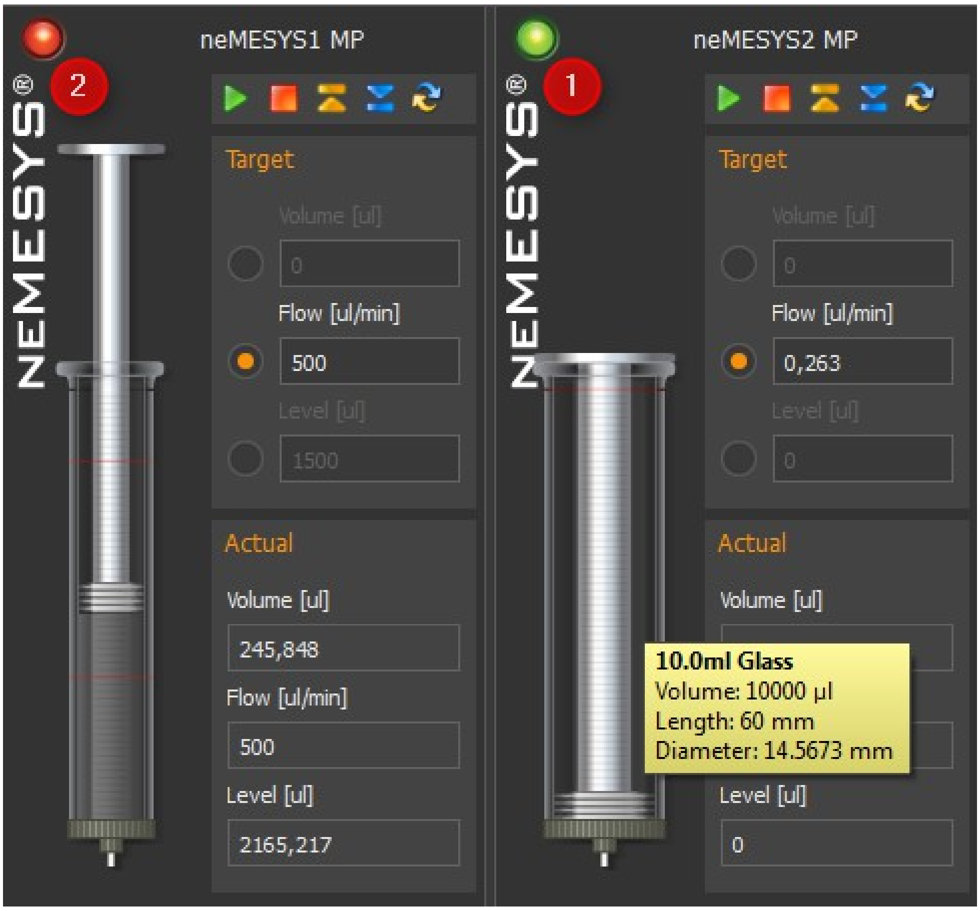


Figure 136: Pump state display

In order to use the pump again you have to enable it. For this, right-click the pump operating panel 1 and select *Enable pump drive* 2 from the context menu. (see figure below)

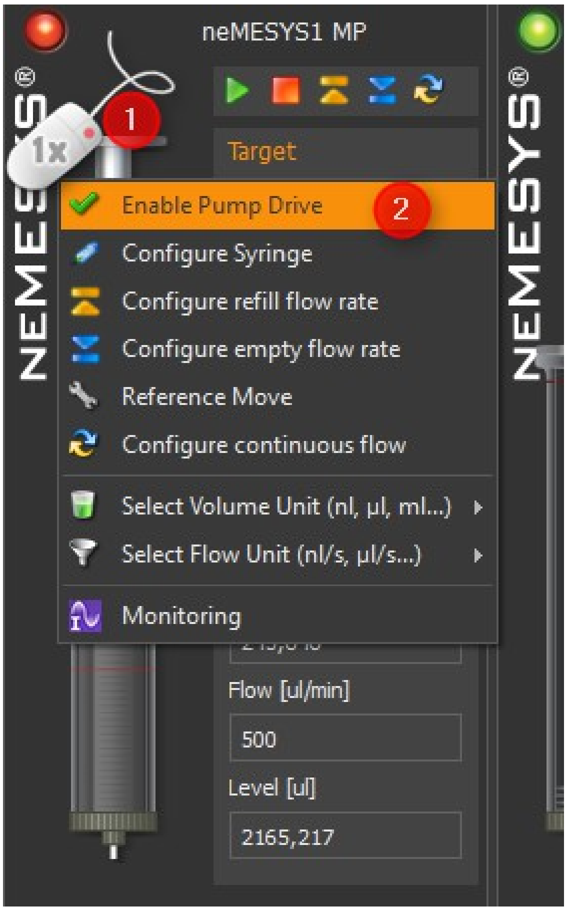


Figure 137: Enable a pump

7.16 neMESYS Script Functions

7.16.1 Introduction

The neMESYS plugin offers a number of script functions which can be used to program automatic sequences. The following script functions are available:

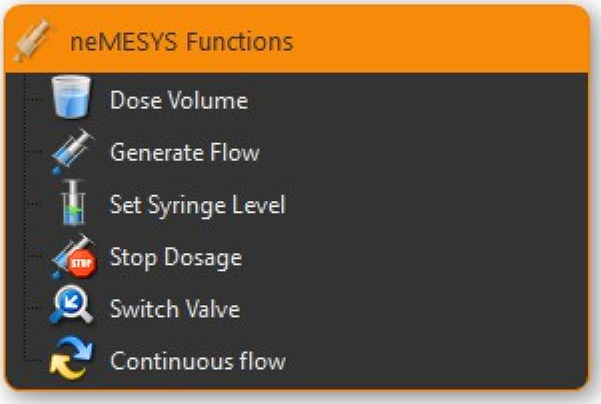


Figure 138: neMESYS script functions

7.16.2 Dose Volume

With this function, you can dose a specific volume at a precisely defined flow rate. You can set all the parameters in the configuration area, for example the dosing module ❶, the volume to be dosed, and the flow rate ❷.

You can also activate or deactivate the *Run to completion* ❸ parameter in the configuration area. When *Run to completion* is activated, the script execution is not continued until the complete volume has been dosed and the dosing process has ended. If this parameter is not active, the dosing is started, and then the next script function is executed immediately. This enables you, for example, to start a number of dosing modules almost simultaneously.

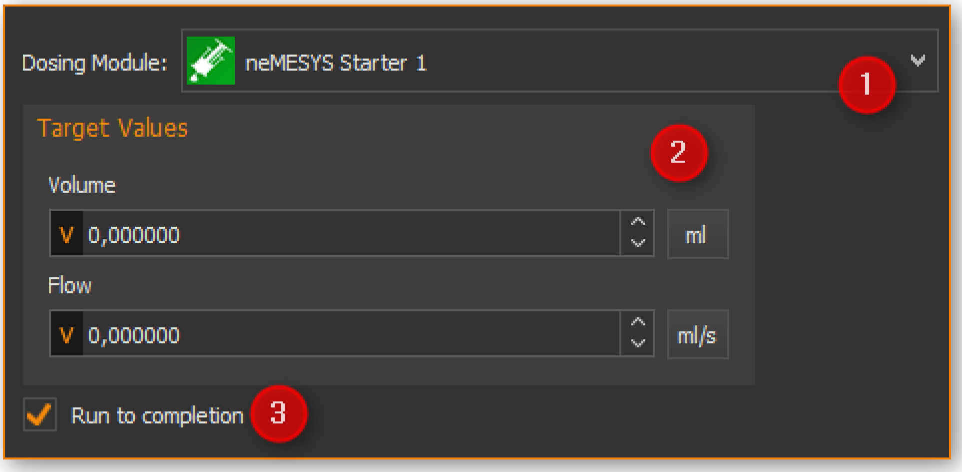


Figure 139: Dose Volume Script Parameter

HINT

*All the dosing functions support the use of variables. That means, in all input fields marked with a coloured **V** in the script configuration panel (e.g. flow rate and volume) you can enter variables.*

7.16.3 Generate Flow

This function is used to generate a constant flow rate. In the configuration area, you can select the dosing module and set the flow rate. If the *Run to completion* parameter is active, the next script function is not executed until the module has stopped or reached one of the limit positions.

7.16.4 Set Syringe Level

You can use this function in a script if you want to reach a specific syringe fill level. The dosing module then doses until the target fill level is reached. You can set the dosing module, the fill level and the flow rate in the configuration area of this function.

7.16.5 Stop Dosage

You can stop an active dosing process of a module with this function.

7.16.6 Switch Valve

Use this function in a script to switch the 3-way valve of a dosing unit.

7.16.7 Continuous Flow

This function allows you to start a continuous flow of two pumps from a script. To do this, select the first dosing unit in the script configuration area of this function and then start the *Continuous Flow Wizard* to configure all parameters.

8 Qmix I/O Plugin

8.1 Introduction

The Qmix I/O plugin is used to integrate various Qmix I/O modules into the QmixElements software and for displaying I/O channels of other devices like neMAXYS positioning systems or neMESYS syringe pumps. The following modules are currently supported:

- *QmixIO* – generic I/O module
- *QmixIO-B* – I/O module with extensible bus terminals
- *QmixP* – two-channel pressure measurement module

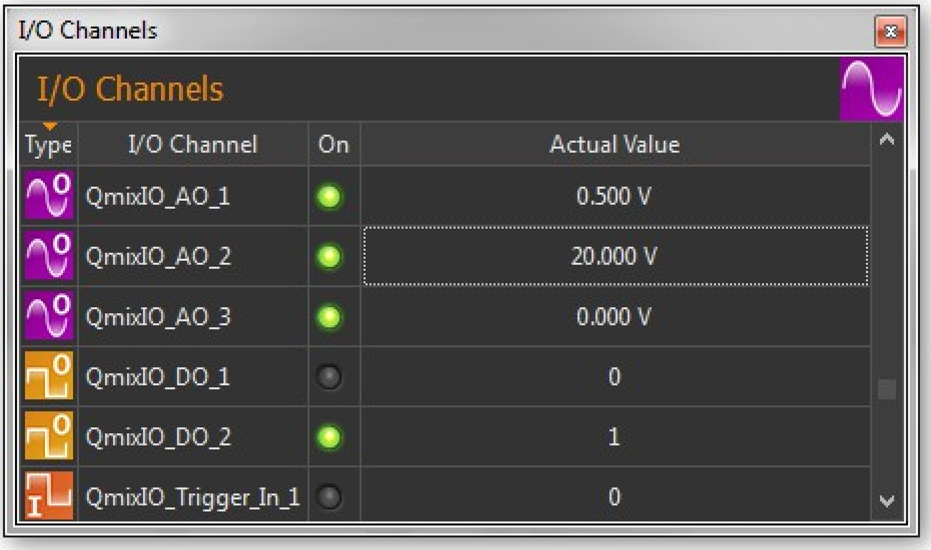


Figure 140: List of I/O channels

8.2 List of I/O channels

All available Qmix I/O channels are shown in the list of I/O channels. You can distinguish the different types of I/Os (AI: analogue input, AO: analogue output, DI: digital input, DO: digital output, Pressure: pressure sensor) by their symbols and names (see figure above). The window with list of I/O channels can be moved freely to another position in the graphic interface by Drag-&Drop via the title bar; you can also move it out of the interface into a separated window.

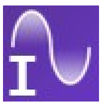
The channel list shows you all the available I/O channels in tabular form. The following columns are present:

- *Type* – shows the type of channel in symbol form: output channels are shown in red and input channels in blue.
- *I/O Channel* – contains the name of the I/O channel.
- *On* – a lit green LED indicates that a channel is switched on and that a digital channel is 1 (instead of 0), respectively.
- *Actual Value* - Shows the current value of the channel - in the case of output channels, this is the value that is output, and in the case of input channels the value read from the device.

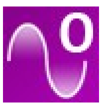
You can sort the table by clicking a column header. For example, you can sort by channel type or by channel name. Clicking the same column header again reverses the sorting order.

8.2.1 Channel types

The following types of channel are currently supported:

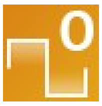


analog inputs

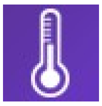


analog outputs

 digital inputs

 digital outputs

 analog pressure sensor inputs

 analog temperature sensor inputs

virtual channels

8.3 I/O Channel Configuration

8.3.1 Changing channel names

You can change the name of each channel at any time, for example, to assign a memorable name suitable for your particular application. You may change a name by the following steps:

1. Double-click the name you want to change.
2. The name is now highlighted in yellow: Enter the new name (figure below).
3. Complete your entry by pressing the *Return* key.

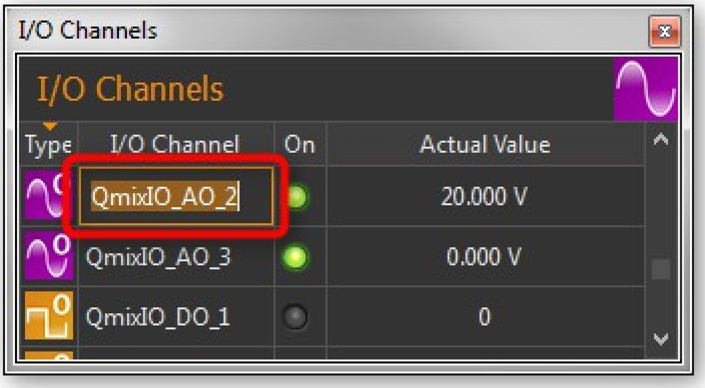


Figure 141: Changing a channel name

8.3.2 Activating the Configuration Dialog

In addition to customizing the name, for most I/O channels there are additional parameters that may be configured, such as the scaling of analog in- and outputs. You will find these configuration parameters in the respective configuration dialog of each channel.

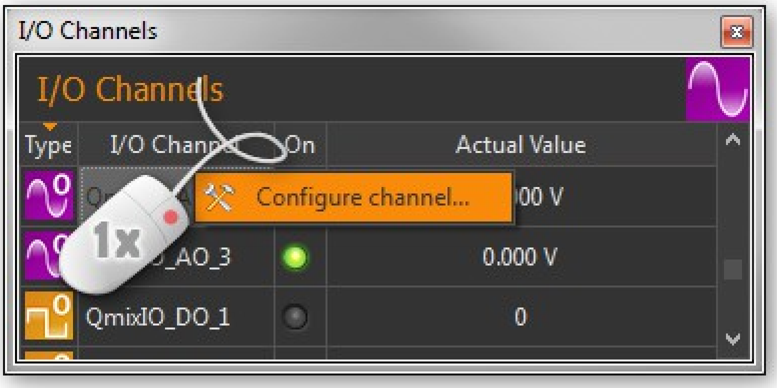


Figure 142: Opening the channel configuration panel

To open the configuration panel, right-click on the respective channel name from the I/O channel list and select the context menu item *Configure I/O channel* (figure above).

8.3.3 Configuration Dialog

In the configuration dialog you can configure all accessible parameters of an analog in- and output channel:

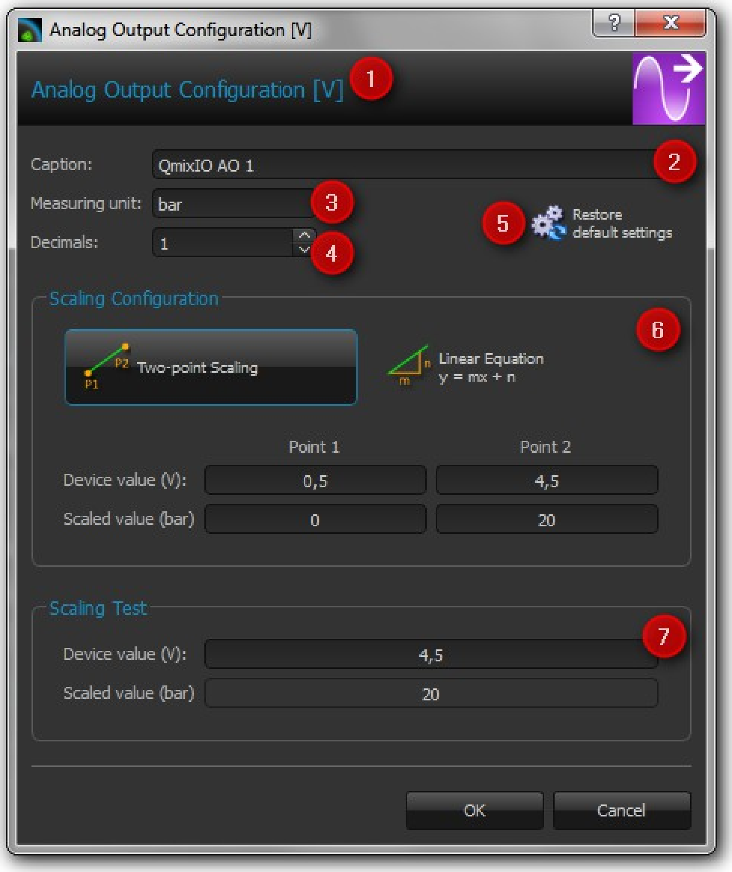


Figure 143: Configuration dialog for I/O channels

1 The header of the dialog identifies the analog channel as an input or output channel. This is followed, in brackets, by an input's native unit of measurement (for example, „V“ for a voltage or „bar“ for a pressure).

2 *Caption* – Here you can set a new name for that channel.

3 *Measuring Unit* – In this field you can set the unit of measurement that is to be used for all scaled values. For instance, if you want to connect the pressure sensor to a voltage input channel, you can set the unit of the

pressure sensor to, for example, “bar”.

4 *Decimals* – This field is to set the number of decimal points of the scaled unit of measurement.

5 *Restore default settings* – By clicking this button, all parameters (channel name, decimal points, scaling, *etc*.) to their original values.

6 *Scaling Configuration* – This is to set the scaling of the channel.

7 *Scaling Test* – Here you can test your scaling parameters: Insert a device value; the software will calculate and return the calculated scaled value.

HINT

All changes will be activated only after being accepted by pressing ***OK****. To cancel all changes use* ***Cancel****.*

IMPORTANT

Clicking ***Restore default settings*** *will instantly reset all parameters to their original setting – clicking* ***OK*** *is not required.*

8.4 Scaling Analog Inputs & Outputs

8.4.1 Introduction

You can change the scaling of all analog inputs and outputs as required. This allows you, for instance, to connect sensors to your analog input ports and to scale the sensor values to your liking. The scaling parameters are set in the configuration dialog of the respective channel.

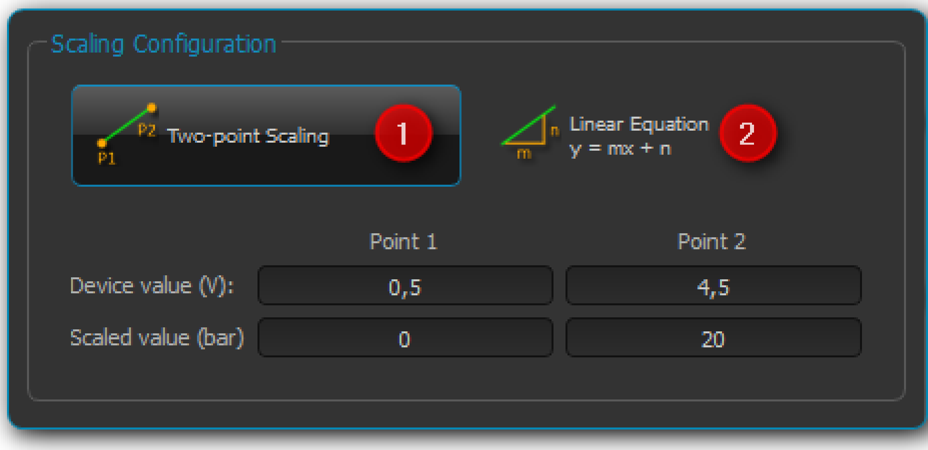


Figure 144: Configuration of the parameter scaling

The scaling parameters set the linear scaling of analog measurement values. There are two ways to enter the scaling parameters (see figure above): via a two-point scaling 1 or via a linear equation 2.

IMPORTANT

Currently, the software only supports linear scaling of measurement values.

8.4.2 Two-Point Scaling

The two-point scaling method is best suited in cases when you can assign a measurement range of a sensor (*e.g.*, bar) to the analog input range of that sensor (*e.g.*, volt).

For example, if you have a pressure sensor with an analog input range from 0.5 V to 4.5 V and a measurement range from 0 bar to 20 bar, you can readily use the two-point scaling method.

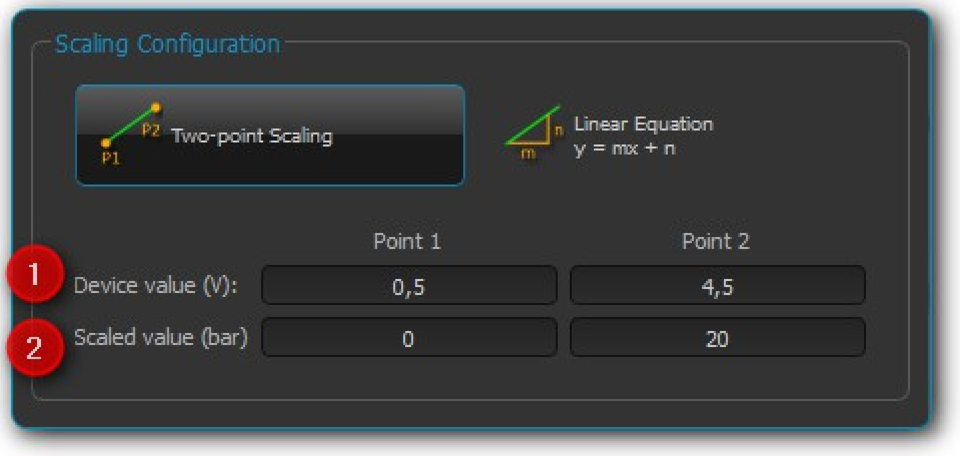


Figure 145: Two-point scaling

In the configuration panel, first insert the two *Device value* points (low, high) of the analog input range (*e.g.*, 0.5 V and 4.5 V for a pressure sensor) (see figure above 1).

HINT

*The description of **Device value** field shows in parentheses the native unit of measurement of the analog channel.*

Insert the *Scaled value, i.e.*, the respective values (low/high) that define the measurement range of the sensor input (*e.g.*, 0 bar and 20 bar) (figure above 2).

Via *Scaling Test* you can now check the correct scaling by entering test values.

IMPORTANT

*The inserted scaling parameters will only be save and become active after the configuration dialog has been completed via **OK**.*

8.4.3 Linear-Equation Scaling

The software internally uses a linear equation ($y = mx + n$) for scaling. The two-point scaling method above is merely a simplified method to provide the software with the required scaling parameters. If you select the *Linear Equation* method in the configuration dialog, you can directly set the parameters for the linear equation underlying the scaling.

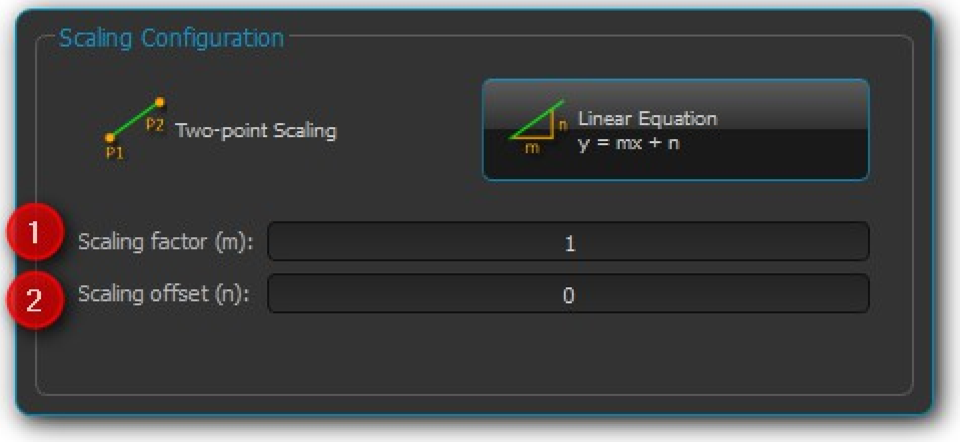


Figure 146: Setting the linear scaling equation

First insert „m“ (*Scaling factor* 1) followed by the offset „n“ (*Scaling offset* 2) in the respective input boxes (figure above).

8.5 Predefined configurations

Some analog input and output channels offer a choice of predefined configurations. These include pressure sensor configurations for the analog inputs of neMESYS devices.

IMPORTANT

Predefined configurations are not offered by all analog channels. For those channels the menu item Select predefined configuration is not displayed in the context menu.

To select a predefined configuration proceed as follows:

1 Right-click the respective analog channel in order to show the context menu.

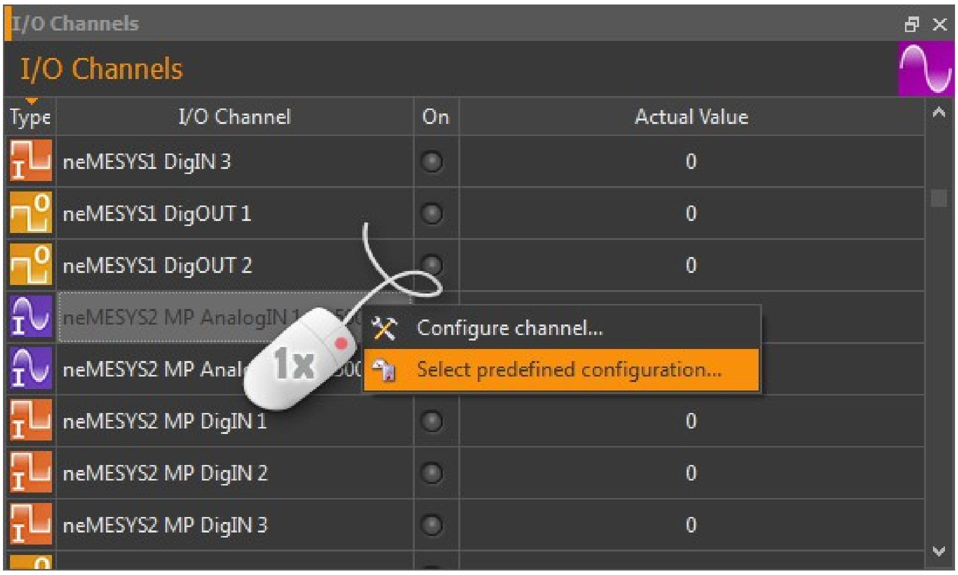


Figure 147: Opening predefined configurations dialog

2 Choose *Select predefined configuration*

3 A dialog containing a selection of configurations appears. Select the configuration you are going to use and confirm your choice by left-clicking *Ok*.

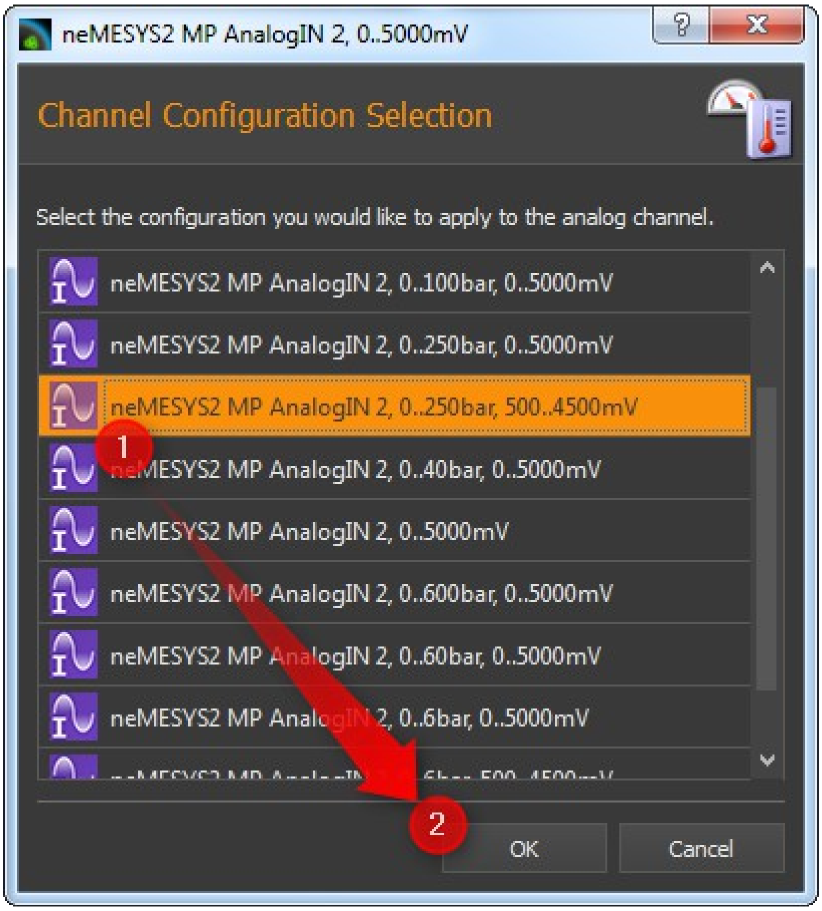


Figure 148: Selecting predefined configurations

After you have chosen a predefined configuration you can overlay a user-specific configuration. In order to do so just proceed as described in section 8.3.3.Configuration Dialog before.

HINT

Predefined configurations can be overlayed by user-specific configurations/scalings. This way you can, as an example, show the value of a pressure measurement channel in an alternate pressure measurement unit as psi instead of bar.

When changing to another predefined configuration it's default settings are applied. That means user-specific configurations and scalings get lost.

IMPORTANT

When changing predefined configurations, user-specific settings and scalings get lost.

8.6 Virtual Channels

8.6.1 Creating virtual channels

The software allows the creation of virtual I/O channels. These channels are not assigned to a physical I/O device, but they are a kind of memory. You can write values into virtual channels and read them out later - just like a memory. Using these channels, you can, for example, show calculated values from a QmixElements script in the graphical logger. You only have to create the channel, add the channel to the logger, and then you can write values from the script into the channel and you will see them in the graphical logger.

To create a virtual channel, click with the right mouse button in the I/O channel list and select the context menu item *Create virtual channel*.

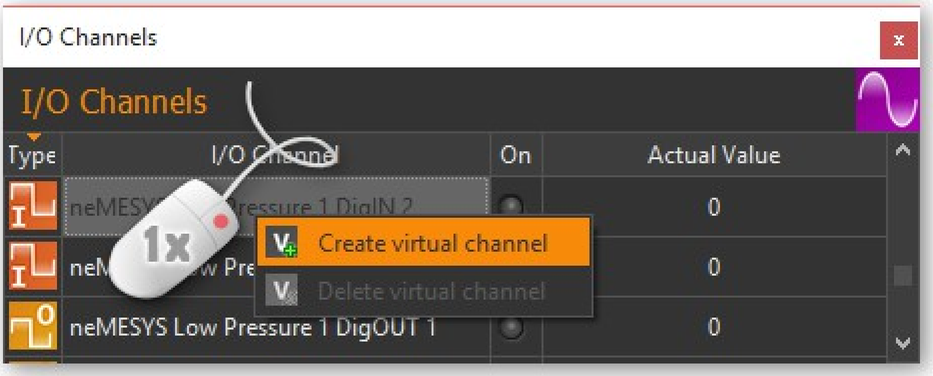


Figure 149: Creating virtual channels

A virtual channel is then inserted at the end of the I/O channel list (see figure below) and you can further configure the channel – e.g. change the channel name.

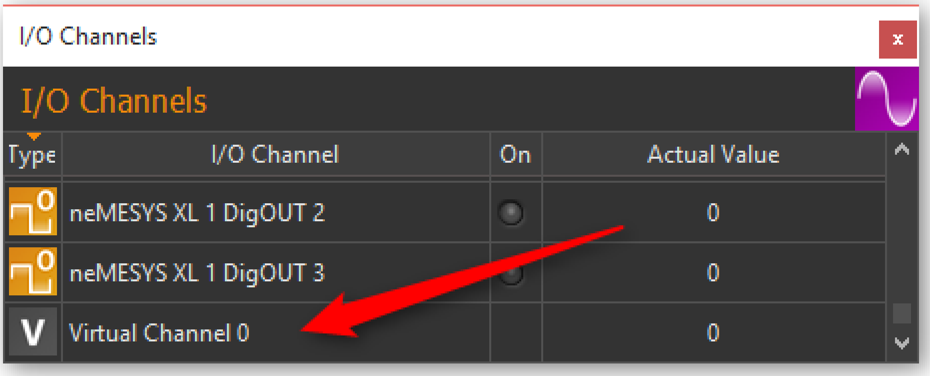


Figure 150: Virtual channels in the list of I/O channels

8.6.2 Access to virtual channels out of script programs

To access virtual channels out of QmixElements script programs (read and write access), you can use the script functions from the category Device Functions (figure below).

Figure 151: Read and write access to virtual channels

8.6.3 Deleting virtual channels

To delete a virtual channel, click with the right mouse button on the channel in the I/O channel list, and then select from the context menu the item *Delete virtual channel* (figure below).

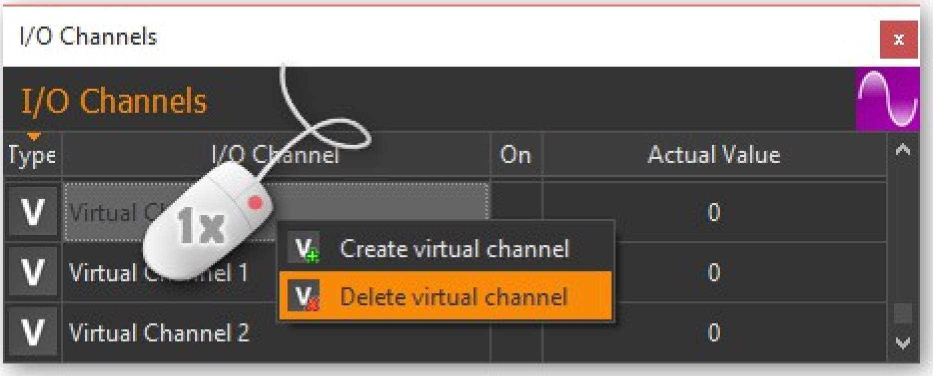


Figure 152: Deleting virtual channels

8.7 I/O Script Functions

8.7.1 Introduction

The Qmix I/O plugin contains script functions for switching digital outputs and setting the initial values of the analogue outputs.

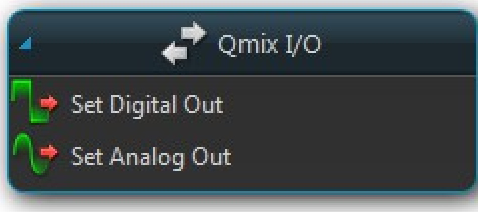
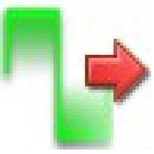


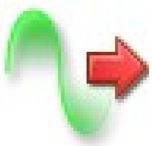
Figure 153: I/O script functions

8.7.2 Set Digital Out



You use this function to set/delete a digital output from a script. Select the digital channel in the configuration area of the function and then set the desired initial value.

8.7.3 Set Analog Out



With this function, you can write a value from a script to an analogue output channel. Select the analogue channel in the configuration area, and then configure the analogue initial value that is to be set during the subsequent execution of the function.

This function supports the use of variables. This means that, instead of a numeric value, you can insert a name of a variable into the field *Value*. This variable will then be set to the analog output value when the script is run (figure below). This variable may then subsequently be used for calculations or to carry out value-specific functions.

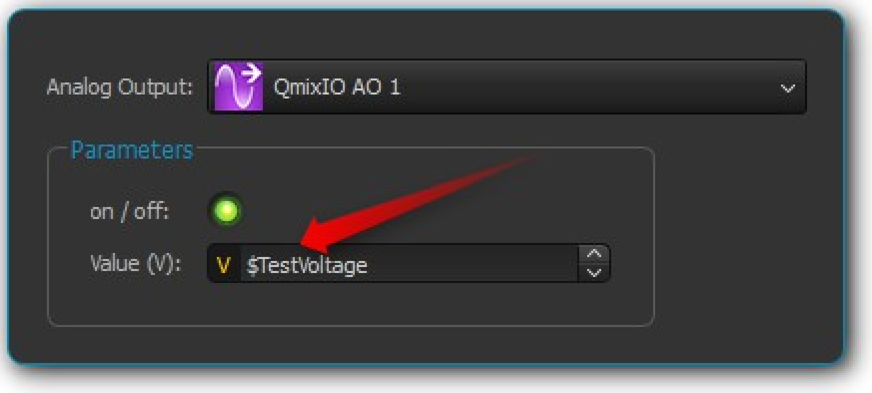


Figure 154: Analog output variable configuration

9 Qmix Controller Plugin

9.1 Introduction

The Qmix Controller plugin is used to integrate the Qmix controller modules into the QmixElements software. The following modules are supported:

- *Qmix Q-* - thermoelectric cooling module
- *Qmix Q+* - two-channel heating module
- *Qmix TC* – tow channel controller module for external heating- / cooling-systems (i.e. for connecting syringe heating)

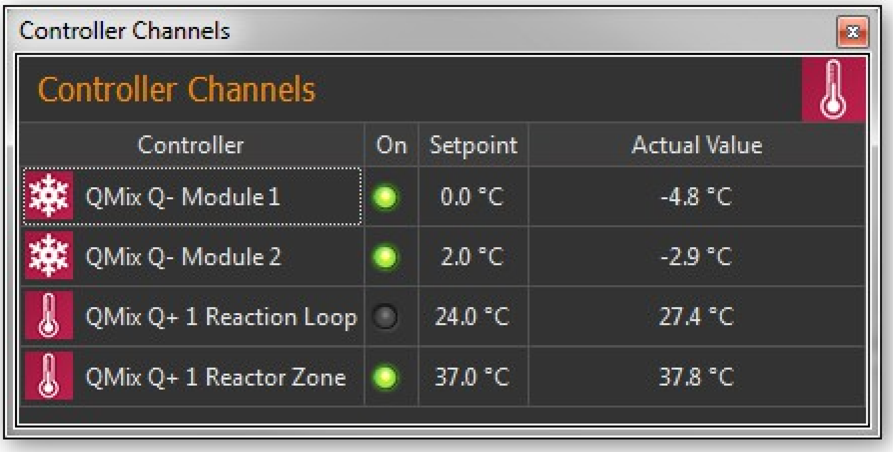


Figure 155: Controller Channel list

9.2 Controller Channel List

All Qmix controller channels are shown in the Controller Channels list. You can identify the different types of channels (cooling module Q-, heating module Q+...) by the different signs in front of the module names (see Figure above). The list of controller channels is in a tool window, that you can move freely at any time to another position in the graphic interface by dragging and dropping the title bar, or you can move it out of the interface to become a separate window.

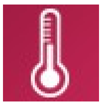
The channel list shows you all the available Qmix controller channels in tabular form. The following columns are present:

- *Controller* - shows the name of the controller module and indicates its type by a sign.
- *On* - indicates by a green LED whether the controller is switched on or off. Click the LED to switch the control loop on / off.
- *Setpoint* -contains the set setpoint of the controller channel
- *Actual Value* - shows the actual value

9.2.1 Channel types

The following types of channels are currently supported:


Snowflake icon: Qmix Q- cooling modules


Thermometer icon: Qmix Q+ heating modules

9.2.2 Changing channel names

You can change the name of a channel at any time and, for example, assign a name suitable for your particular application.

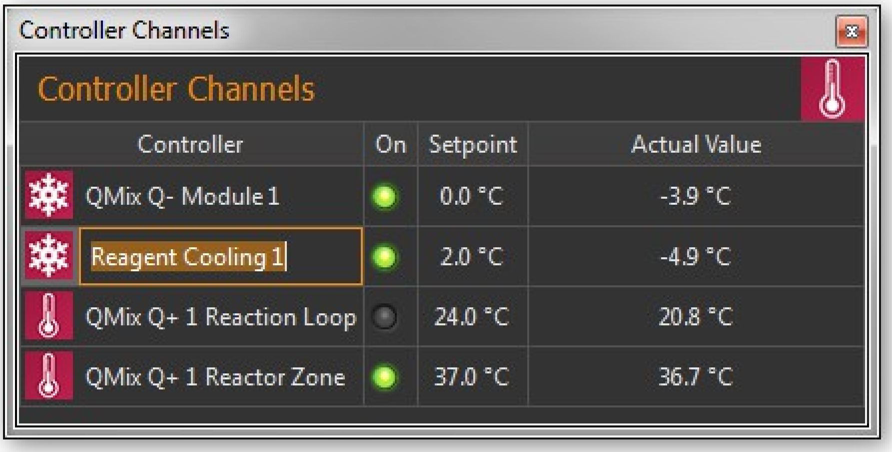


Figure 156: Changing channel names

You change a name by the following steps:

1. Double-click the table cell containing the name you want to change.
2. Enter the new name in the Editing window which now appears (Figure above).
3. Complete your entry by pressing the *Return* key.

9.2.3 Switching control devices on / off

To switch the controller on or off, simply click the LED of the channel you want to switch.

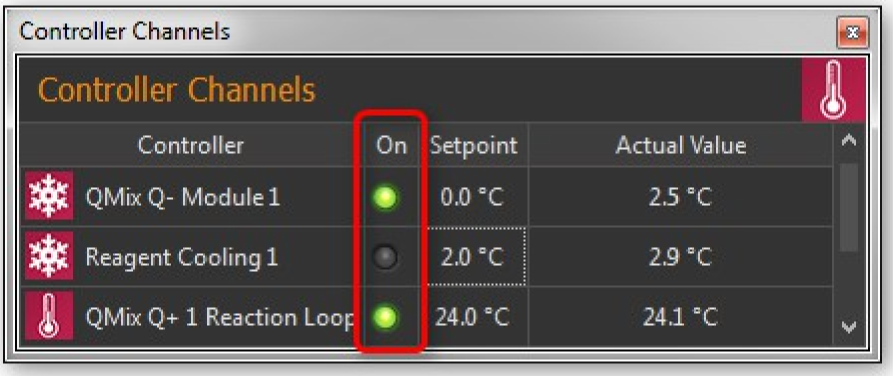


Figure 157: Switching control devices on/off

9.2.4 Entering setpoint

To input a setpoint, double-click in the Setpoint column of the channel with the setpoint you want to change. Now enter the setpoint in the Editing window that opens (Figure below) or use the arrow buttons to raise or lower the setpoint incrementally.

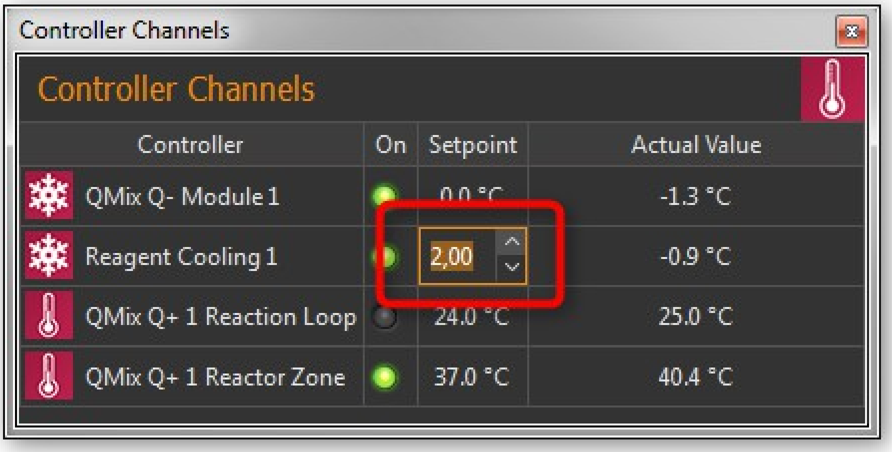


Figure 158: Changing setpoint

9.3 Context menu for control channels

By right-klick into the control channel list you can display a context menu with additional functions.

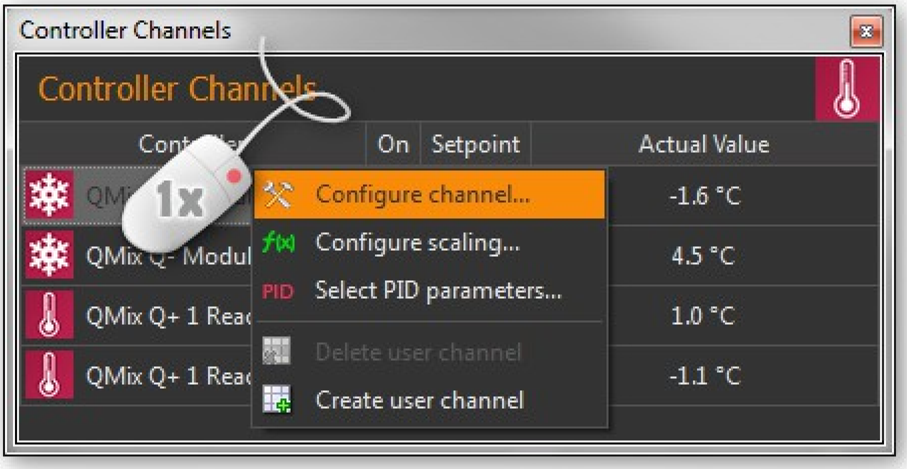


Figure 159: Opening the context menu

The context menu contains the following menu items:

Configure Channel...

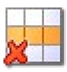
Opens the configuration dialog of the channel for configuring all channel parameters.

Configure scaling...

Opens the configuration dialog displaying the page for configuration of the controller scaling.

Select PID parameters...

Opens the configuration dialog displaying the page for configuration of the PID control parameters.



Delete user channel

If the selected channel is a user-specified channel, it is deleted by selecting this menu item. For other channels this menu item is disabled.

Create user channel

Opens up the wizard for creating user-specified control channels.

9.4 Configure scaling

For opening the scaling configuration dialog select *Configure scaling* in the context menu of the control channel. (see section 9.3)

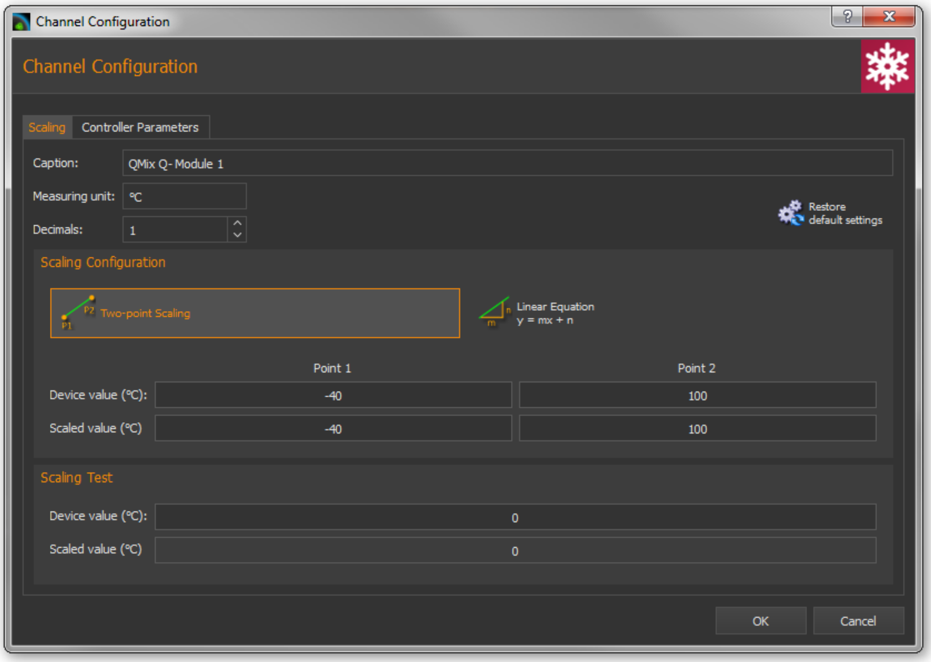


Figure 160: Configure scaling

The configuration of the control channel scaling equals the I/O channel scaling configuration of the Qmix I/O plugin. For a detailed description read the section I/O Channel Configuration.

9.5 Selection and configuration of controller parameters

9.5.1 Overview

To set the optimal control behavior, you can adjust the controller parameters of each single channel. For this you can either choose from a list of predefined PID parameter sets or create new parameter sets. To access the controller parameters selection, choose *Select PID parameters* from the controller channel [context menu](#).

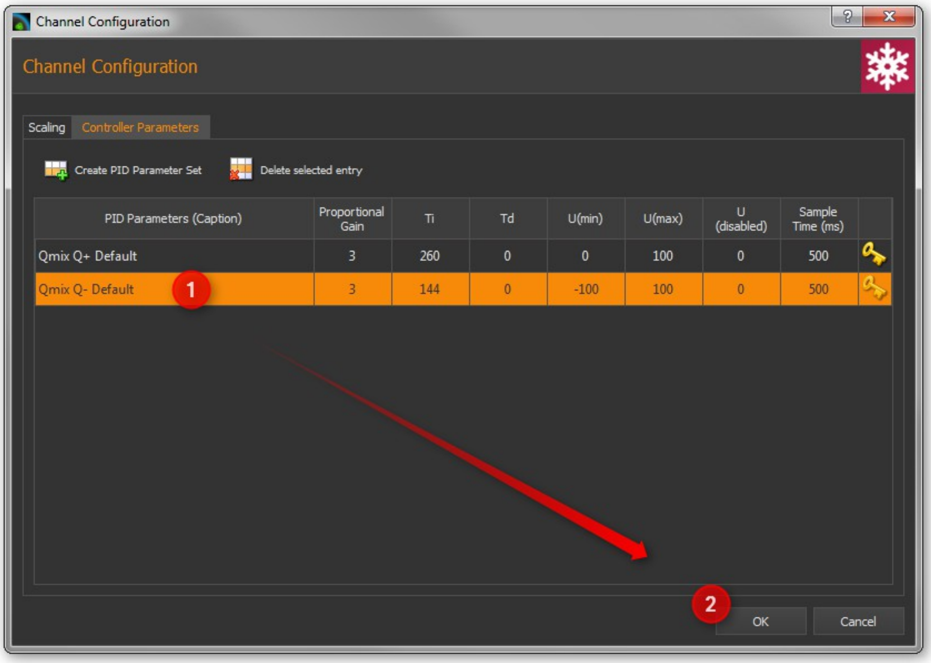


Figure 161: Selecting a control parameter set

From the table of predefined control parameters select one parameter set by left-click and finish configuration by clicking the OK button. The table already contains default controller parameter sets for different Qmix devices like Qmix Q+, Qmix Q- or Qmix TC or certain accessories, i.e. syringe heating or tube heating.

HINT

For optimum adaptation to the controlled systems in your application, you can create your own parameter sets with controller parameters.

9.5.2 Creating a set of controller parameters

By selecting the *Create PID Parameter Set* button 1 you create a new set of controller parameters 2. (see figure below) You can then edit the individual values of the parameter set.

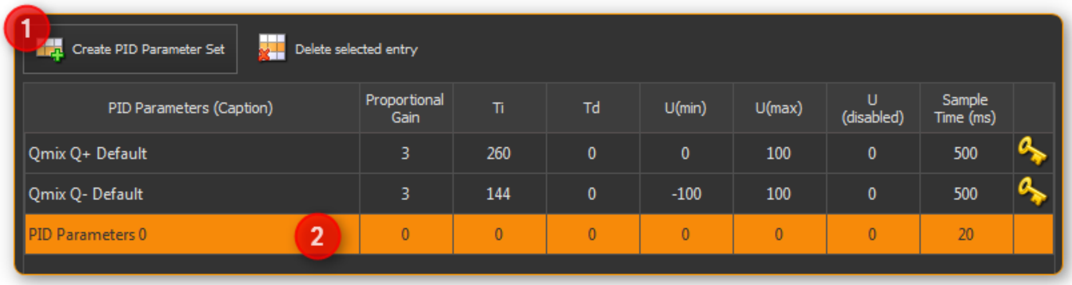


Figure 162: Creating a set of PID parameters

9.5.3 Editing controller parameters

You can change a certain value of a set of controller parameters by double-click in the appropriate table field. (see figure below)

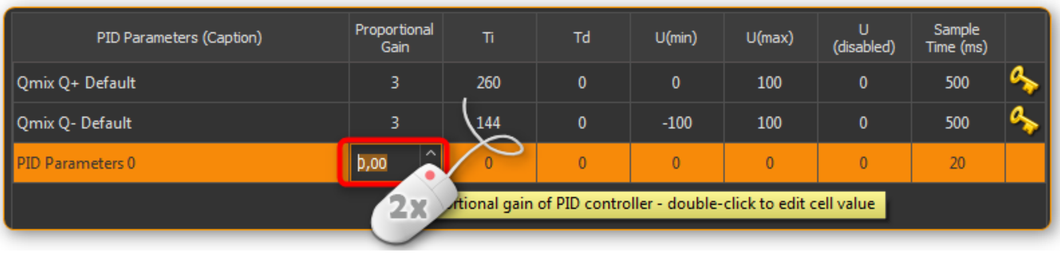


Figure 163: Edit a PID parameter set

For finding adequate controller parameters proceed as described in section Procedure for setting controller parameters.

9.5.4 Deleting controller parameters

Select a set of controller parameters from the table 1 and left-click the *Delete selected entry* button 2 for deleting it. (see figure above)

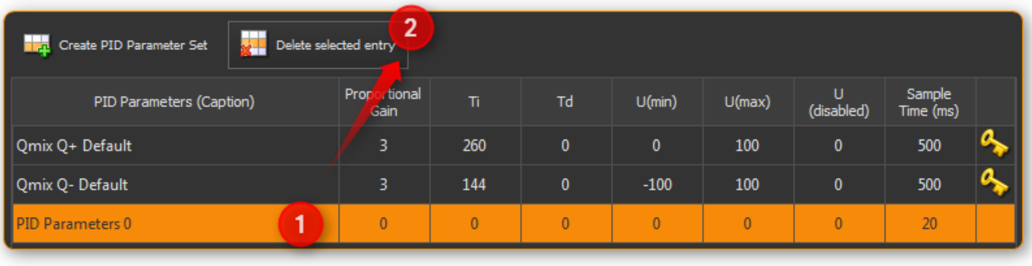


Figure 164: Deleting a PID parameter set

IMPORTANT

Predefined controller parameters are locked and can not be deleted. Locked parameters can be recognized in the table by the yellow key symbol.

9.6 Customized control channels

9.6.1 Introduction

You can create customized control channels using arbitrary device properties for control loop input and output. Thus using i.e. a pressure measurement channel of a Qmix p device and a neMESYS syringe pump you can build up a pressure control.

9.6.2 Creating control channels

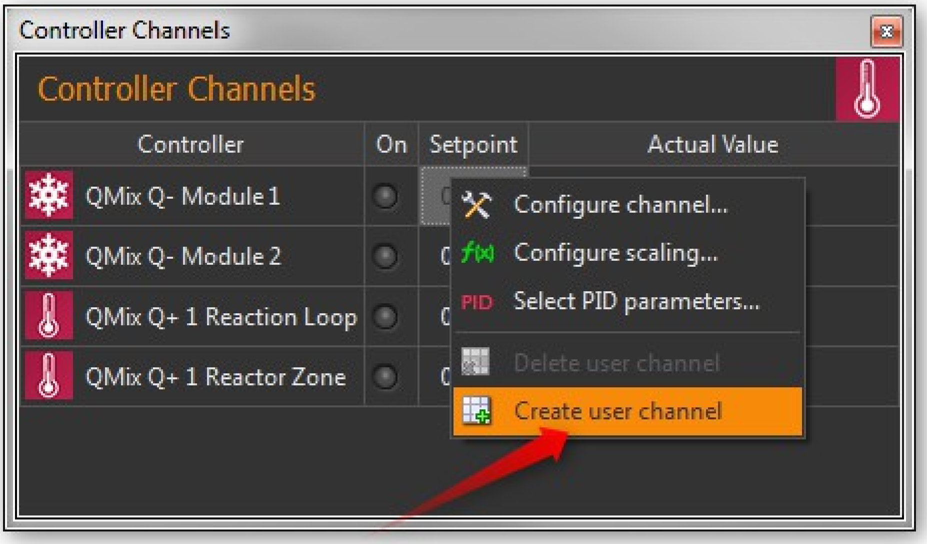


Figure 165: Open controller channel creation dialog

By right-clicking the control channel list and selecting the button *Create user channel* you open the dialog for creating control channels. In the configuration wizard that is displayed now, proceed as follows:

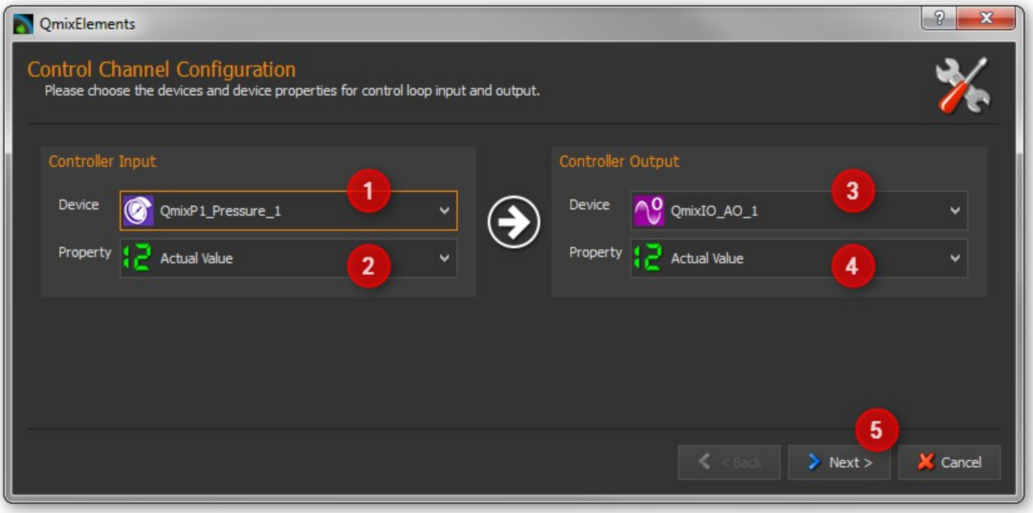


Figure 166: Selecting input and output values of the controller channel

1 Select the device that provides the measurement value (controller input) of the controller.

2 Select the device property that is used as measurement value.

3 Select the device that provides the control value (controller output) of the controller.

4 Select the device property that is used as control value.

5 Click *Next* in order to proceed.

The final page of the wizard allows configuration of controller channel parameters as described in section Selection and configuration of controller parameters. You complete the control channel creation by clicking the *Finish* button.

HINT

You can change the controller parameters at any later time and adapt them perfectly to your controlled system.

9.6.3 Changing the output value scaling or unit

The output value is determined by the PID control algorithm and written to the output device without any scaling information. This means that you need to adjust the controller parameters, in particular the control value limits, each time you change the scaling or the unit of the appropriate device. With a neMESYS dosing module this is also true if you change the syringe size.

IMPORTANT

If you change the scaling or unit of a device beeing part of a control loop you must check the control parameters and adjust them if necessary. With a neMESYS dosing module this also applies if you change the syringe.

9.7 Controller Script Functions

9.7.1 Introduction

The Qmix controller plugin contains a script function for changing the controller parameters from a script. This makes it possible to achieve, for example, time controlled temperature curves.

Figure 167: Qmix controller script functions

9.7.2 Set Controller Param

With this function, you can transfer a new setpoint to the controller channel or switch the control loop on or off. To switch it on and off, simply click the *Control loop on / off* LED in the configuration area (see figure below).

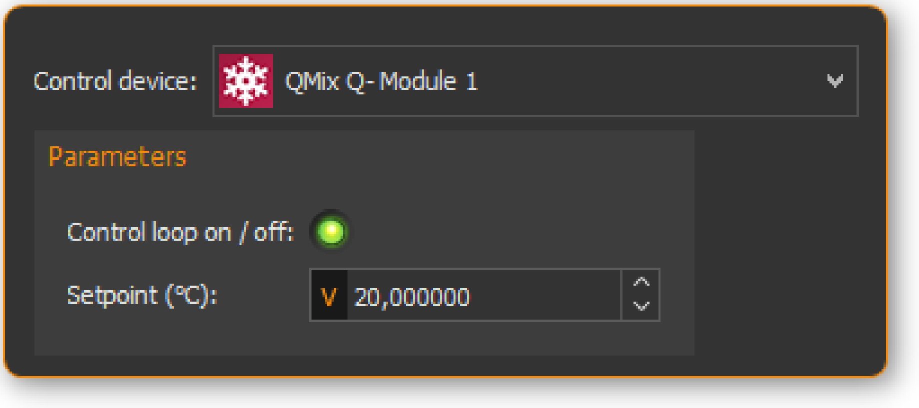


Figure 168: Configuration of the controller script function

9.8 Procedure for setting controller parameters

9.8.1 Closed control loop and PID controller equation

The controller (e.g. PID controller) and the controlled system (e.g. temperature-controlled system) together make up a feedback system, the closed control loop. (see figure below)

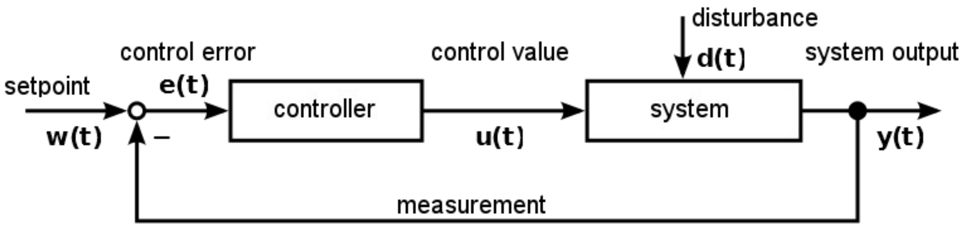


Figure 169: Closed control loop

A PID-controller determines the control value *u* at the time t_1 using the following equation:

$$u(t_1)=K_p \cdot e+\frac{K_p \cdot \Delta t}{T_i} \cdot \sum_{t=t_0}^{t_1-\Delta t} e+K_p T_d \frac{\Delta y}{\Delta t}$$

The control value contains 3 shares.

$K_p \cdot e$	The proportional share (P) forms by means of the factor K_p the direct effect of the control error on the control value.
$\frac{K_p}{T_i} \cdot \sum_{t=t_0}^{t_1-\Delta t} e$	The integral share (I) computes the sum of the error over the time and by means of K_p and the time constant T_i maps it to the control value. The bigger K_p and the smaller T_i the bigger the integral share of the control value.
$K_p T_d \frac{\Delta y}{\Delta t}$	The differential share depends on the temporal change of the actual value that is mapped to the control value by K_p and T_d.

9.8.2 Preparations for setting controller parameters in QmixElements

Initially make the plot of the datalogger plugin show the measurement value and the control value to you. (refer to section Process Data Graph) 1 It is recommended to do the same with the setpoint value. It is not necessary but facilitates orientation within the plot. (see figure below)

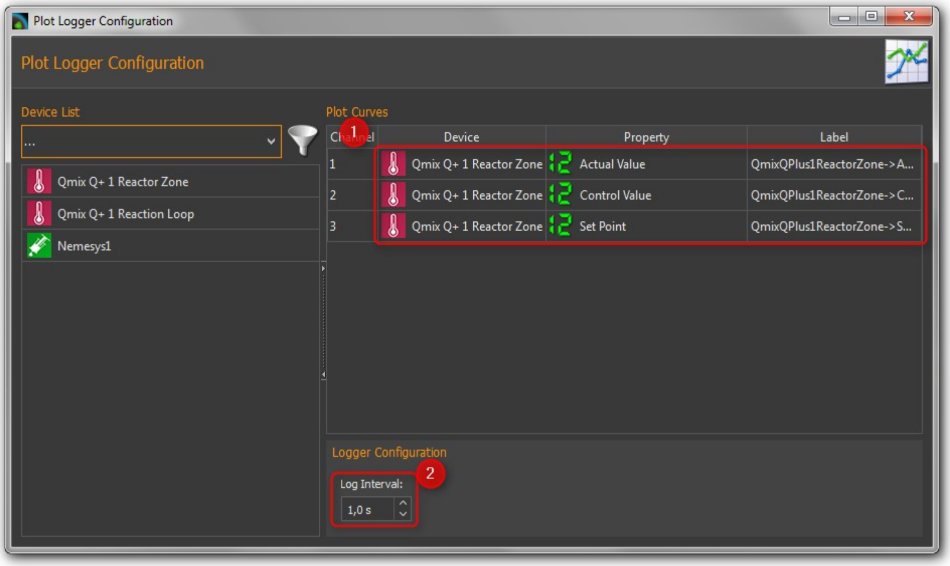


Figure 170: Configuration of the graphical logger for setting controller parameters

The setting for *Log Interval* 2 depends on the change frequency of the actual value. You should get a useful graph if you set *Log Interval* to the sample time used by your control channel. (see section 9.8.3)

9.8.3 Choosing the sample time

The time between 2 computations of the control value is defined as sample time. The smaller the sample time the more often the control value is calculated. As a rule of thumb you can keep in mind that the sample time should not be higher than one tenth of the smallest time constant in the closed control loop. Experiences have shown that the following values achieved useful results (stable control) with the appropriate devices.

Application	Sample Time (ms)
Qmix Q+	500
Pressure control using neMESYS and Qmix p	50

> **HINT**
>
> *For the sample time choose values that are less than or equal to 1/10 of the smallest time constant occurring in the control loop.*

9.8.4 Setting the control value limits

You can limit the minimum (U_{min}) and the maximum (U_{max}) control value of the QmixElements controllers. The control value should have a sufficient stroke in order to reach the desired setpoint values. But you should also take care not to damage the controlled system by choosing to large control value limits. (e.g. too high flow rate of a neMESYS dosing module in a pressure controlled system leads to damage of the fluidic system) You should test the control value limits by temporarily acting with them on your controlled system. (e.g. dose with a neMESYS dosing module using a flowrate equal to the control value limit) Futhermore you have to choose a value assumed by the controller if the control channel is disabled ($U_{disabled}$) which is usually zero.

> **ATTENTION**
>
> *Insufficient limitation of the control value can lead to damage of the controlled system.*

9.8.5 Determination of PI control parameters using the example of a temperature control system

A temperature control system usually makes up a delay line containing one or more delay time constants. It can often be approximated by a first-order delay line. The appropriate step response function is: $F(s) = \frac{K_s}{1 + sT_1}$

Setting the controller parameters targets the compensation of the delay time constant T_1 and adapting the controller gain K_p in order to reach an efficient control behaviour. In practice T_1 is often unknown, but you can gradually approach to a useful result using the following procedure.

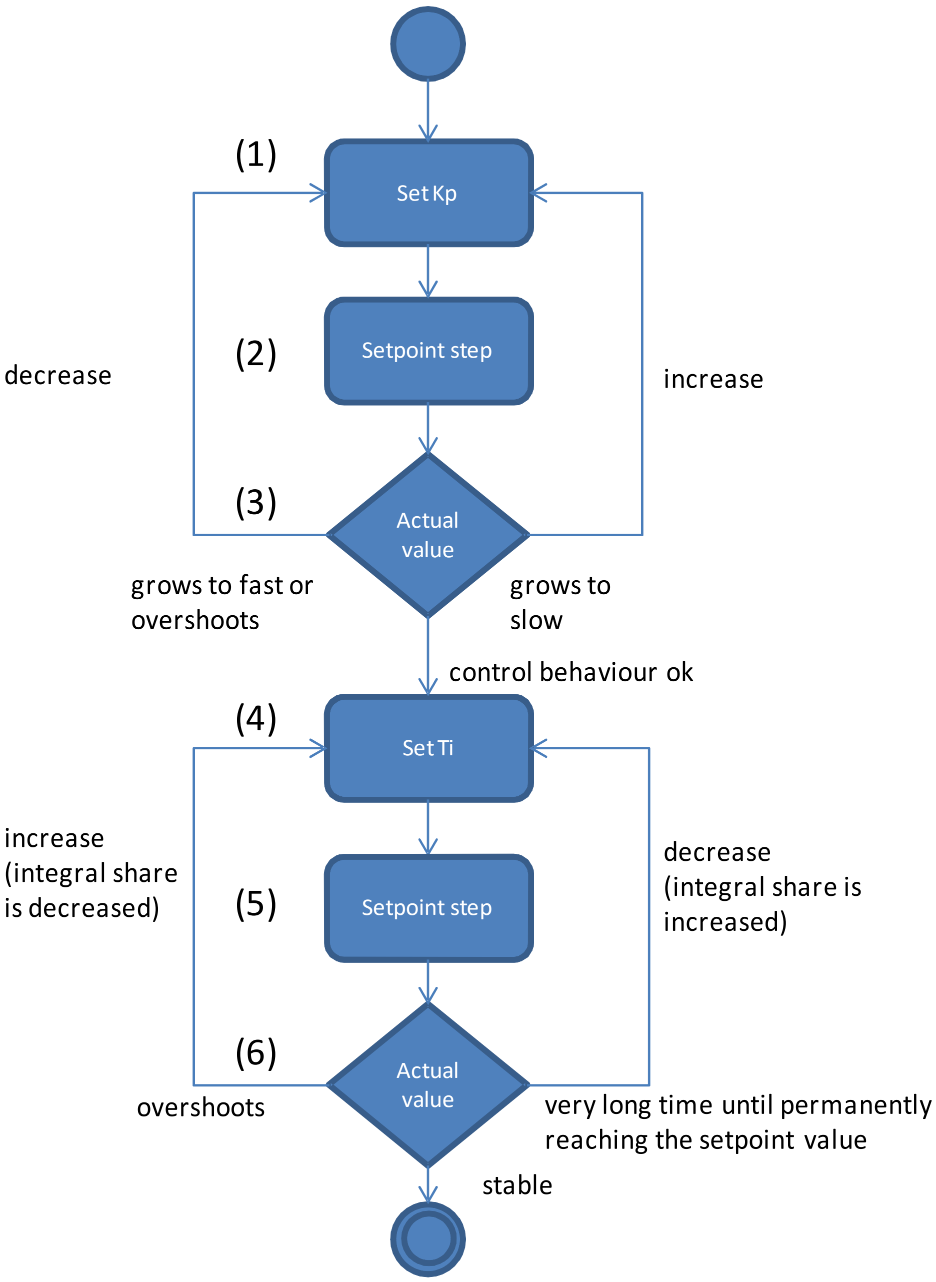


Figure 171: PI-controller setting for a first-order delay line

(1) Initially create a new set of controller parameters as described in the section 9.5.2. Please choose the values for Sample Time, U_{min}, U_{max} und $U_{disabled}$ according to the recommendations of the sections 9.8.3 and 9.8.4.

Disable the differential and integral share by setting the controller time constants to 0. This leads to a simplified controller equation. $U = K_p \cdot e$

Set a moderate value for the proportional gain. Keep in mind that by means of K_p the control error immediately impacts the control value. Choosing a too big value for K_p can drive the controller into saturation.

(2) Give a setpoint step to your control loop, i.e. by changing the setpoint temperature from room temperature to 50°C and activating the control channel. (see section 9.2.4 and 9.2.3)

(3) Providing a first-order delay line the actual value will behave according to the following figure.

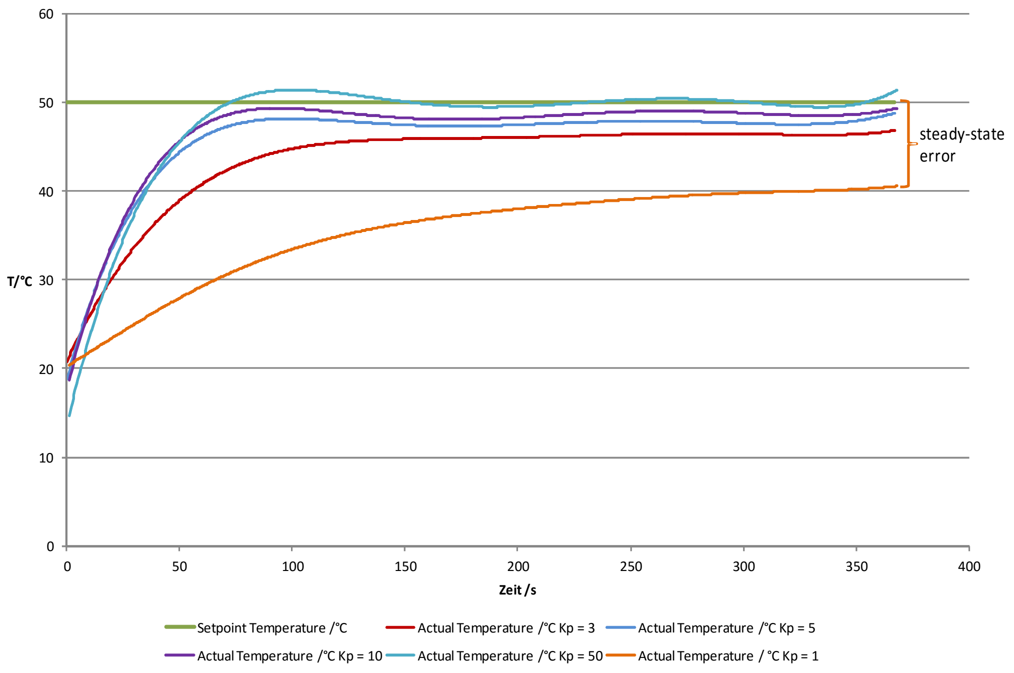


Figure 172: Temperature control using P-controller with different K_p values

Depending on chosen controller proportional gain K_p the actual value will vary quickly settle near the setpoint value. Due to the fact that a proportional controller is not able to fully compensate a delay line, a steady-state error establishes. If K_p is set too low, the actual value approaches only very slowly the target value. (see curve for K_p = 1 in figure above) If K_p is set to high, the actual value overshoots, optionally oscillates about the setpoint. (see curve for Kp = 50 in figure above) In the provided example the actual value reaches a steady-state quickly without overshooting using K_p=3. This is why we use that value for further optimizations.

(4) In the next step set T_i in a way that the steady-state error is compensated. You should begin using a large time constant T_i which means a small integral share.

(5) Give a setpoint step to your control loop, i.e. by changing the setpoint temperature from room temperature to 50°C and activating the control channel. (see section 9.2.4 and 9.2.3)

(6) Lower the time constant T_i if you want to reduce time for permanently reaching the setpoint value. Please notify that a time constant T_i set to small (large integral share) can lead to the control loop oscillating. In the provided figure you can see that T_i=260s leads to a good result. The actual value matches the setpoint value and the system does not oscillate. Using T_i=1000s the setpoint is not reached within the illustrated time range and T_i=20s leads to the system overshooting heavily. (see figure below)

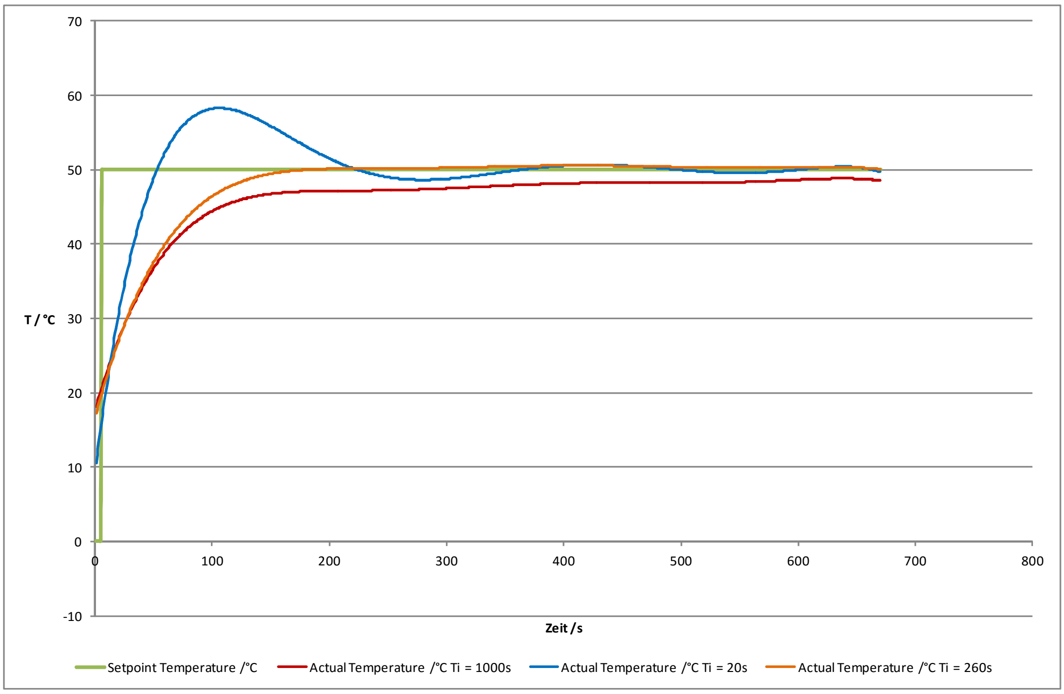


Figure 173: PI temperature-controller using Kp=3 and variable Ti values

(7) In many cases (i.e. temperature control) a PI controller is sufficient. There is no steady-state error and the dynamic behaviour is satisfactory. If the controller shall be robust with respect to sudden disturbances, it might be useful to include a differential component. A detailed consideration of control stability, control behaviour with respect to setpoint changes and disturbances is beyond the scope of this practical introduction. Reference is therefore made at this point on the control engineering literature.

10 QmixV Plugin

10.1 Introduction

The QmixV plugin is used to control the QmixV valve modules or to control valve devices that are part of other devices (i.e. valves mounted on neMESYS syringe pumps).

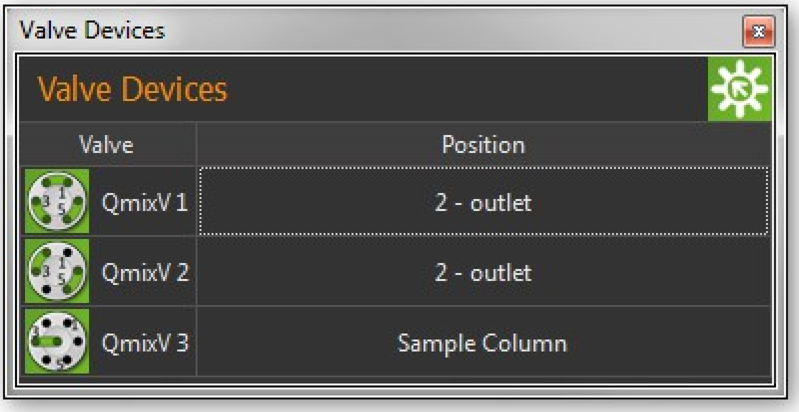


Figure 175: QmixV valve device list

10.2 Valve Device List

The plugin mainly consists of the valve device list (see Figure above), which is displayed as a tool window in the graphical user interface. You can move the window to another position in the graphic interface at any time by dragging and dropping the title bar, or you can move it out of the interface to become a separate window.

The valve device list shows you all the available Qmix valve devices in tabular form. The name of the module and the actual valve position status icon is shown in the left-hand table column, and the right-hand column shows the name of the current position of the valve.

10.2.1 Editing valve names

You can change the name of a valve at any time and, for example, assign a name

suitable for your particular application. You change a name by the following steps:

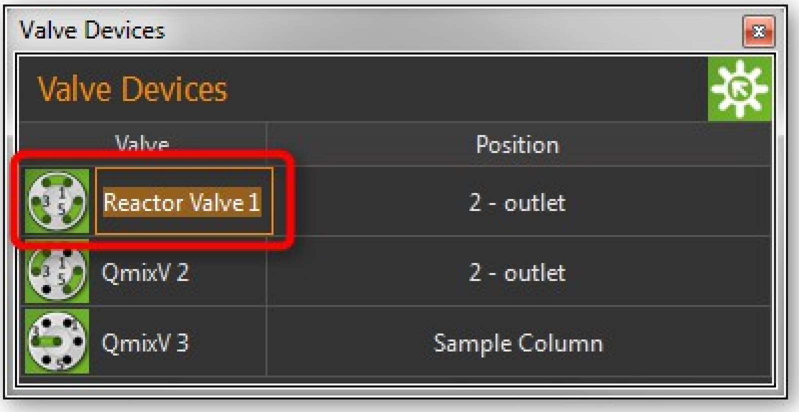


Figure 176: change valve name

1. Double-click the table cell containing the name you want to change.
2. Enter the new name in the Editing window which now appears (see Figure above)
3. Complete your changes by pressing the *Return* key.

10.2.2 Switching valve position

You can also switch the valve position in the module list. For this purpose proceed as follows:

1. Double-click in the table cell showing the valve position.
2. Select the desired valve position in the selection box that now appears (see Figure below).
3. Confirm the selection by pressing *Return* key or click on another table cell to close the selection box.

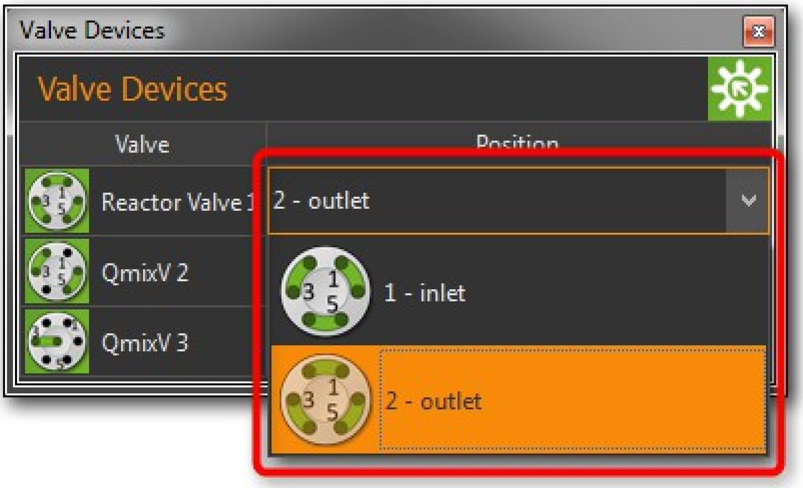


Figure 177: Switch valve position

The valve is now switched into the new valve position.

10.2.3 Naming valve positions

You can give each valve position an individual name in the software, and so configure the names of the valve positions to suit your application. Perform the following steps to change the names of the valve positions.

(1) Make a right mouse click in the line of the table of the valve for which you want to rename the positions.

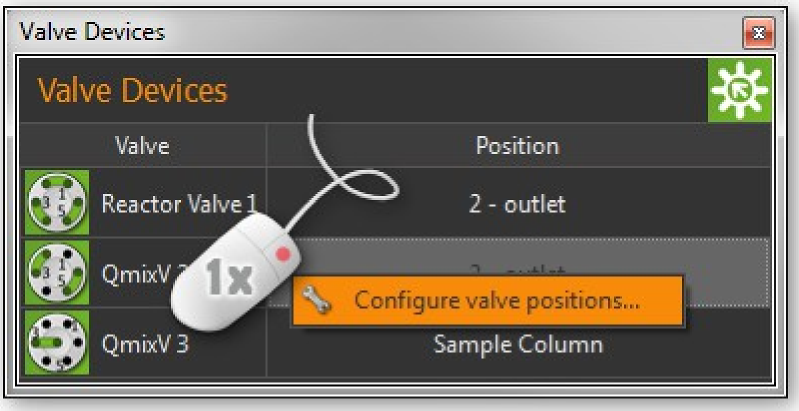


Figure 178: Open valve configuration

(2) Select the *Configure valve positions...* menu item in the context menu, that is being displayed.

(3) A configuration dialog opens (Figure below), which consists of a valve selection box 1 and the list of valve positions 2.

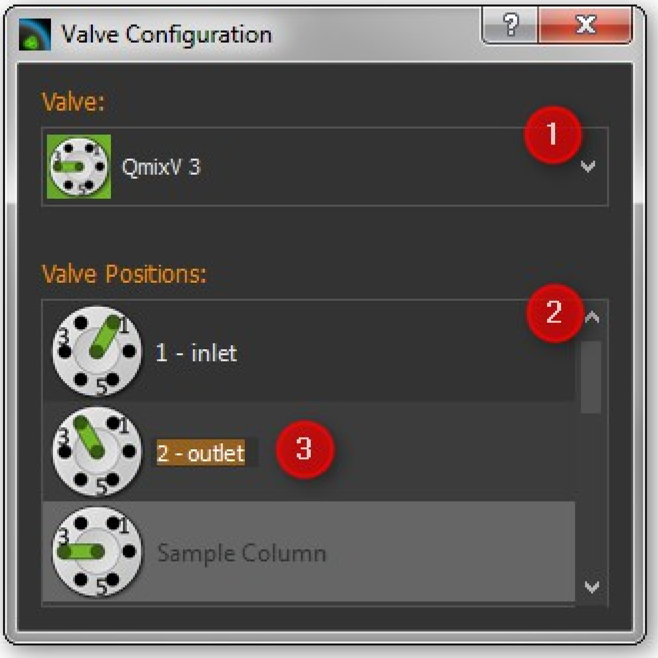


Figure 179: Open valve configuration

(4) Double click the line in the table containing the valve position you want to name.

(5) An Editing window 3 opens in the table cell, in which you can enter the new name of the valve position.

(6) Confirm your entry by pressing the *Return* key.

(7) You can close the configuration dialog when you have named all the valve positions.

From now on, the valve positions will always be shown with the new names in all areas of the software.

10.3 Valve Script Functions

10.3.1 Introduction

The Qmix valve plugin contains a script function for switching the Qmix valves from a script.

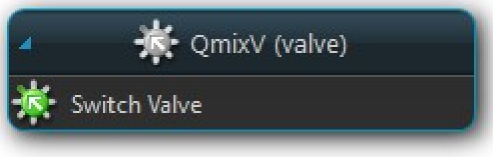


Figure 180: Qmix valve script functions

10.3.2 Switch Valve

You use this function to switch the valve position. In the configuration area of this function, you can select the valve module, and the target position at which the valve is to be switched.

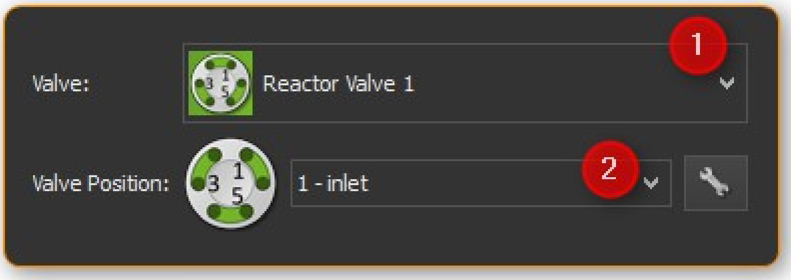


Figure 181: Switch Valve function configuration

11 rotAXYS / neMAXYS Plugins

11.1 Introduction

The QmixElements software supports the control and visualization of various cetoni positioning systems. The description in the following sections is valid for all rotAXYS and neMAXYS devices.

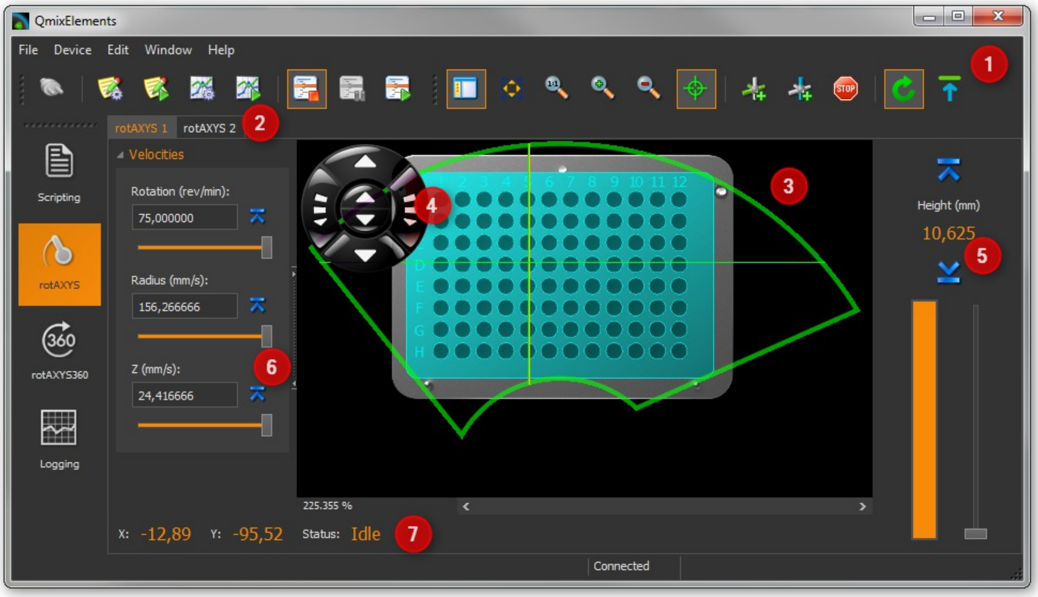


Figure 182: Work area for positioning systems

The working area of the this plugin mainly consists of the following components:

1 Toolbar

2 Device selection tabs

3 Positioning map for XY positioning

4 Control pad for manual control

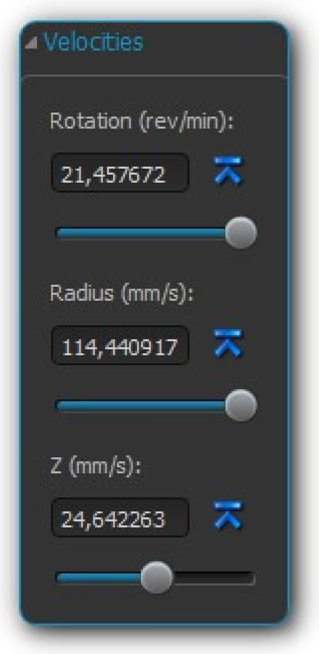
5 Z panel for positioning of Z axis (vertical axis)

6 Sidebar

7 Status Bar

If you work with multiple devices simultaneously, you can use the device selection tabs 2 to show the control panel for the positioning device you want to work with.

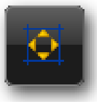
11.2 Sidebar



In the sidebar, which can be turned on and turned off, you can find additional controls to configure and control your positioning systems. In the sidebar there is the *Velocities* panel where you can configure the velocities of all axes. All positionings in the *Positioning Map* or in the *Z-Panel* are then carried out with the set speeds.

11.3 Toolbar

Hides or shows the sidebar with additional configuration and control elements



Adjust the magnification of the positioning map so that the complete display fits into the working area.

Changes the magnification of the positioning map to the actual pixel size

Increases the magnification factor

Decreases the magnification factor

Switches between free positioning and well positioning (see section 11.5 - Positioning Map)

Adds the script function *Move XY* with the current XY position as target position parameter to the script editor

Adds the script function *Move Z* with the current position of the Z axis as target position parameter to the script editor

Immediately stops the movement of all axes of the positioning system

Activates / deactivates the lift axis safety stroke. If it is active, then the Z-axis moves to a safe height before each movement in the XY plane.

Sets the current Z-axis position as safe height. If the safety lift feature is active, then the positioning system moves to this height before each movement in the XY plane.

Configure zero angle - the zero angle is never crossed by the axis system and it is the start and end point of the 360 ° rotation range

11.4 Control Pad

11.4.1 rotAXYS

With the 6-way control pad, you can manually position all the axes of the rotAXYS positioning system.

Rotate the rotary axis counter-clockwise

Rotate the rotary axis clockwise

Move the radius axis away from the centre of rotation (increase radius)

Move the radius axis toward the centre of rotation (decrease radius)

Move the vertical axis up and down

11.4.2 neMAXYS

With the 6-way control pad, you can manually position all the axes of the neMAXYS positioning system.

Moves X-axis to the left

Moves X-axis to the right

Moves the Y-axis toward the back of the device

Moves the Y-axis toward the front of the device

Move the vertical Z-axis up and down

11.4.3 Changing the speed

When a direction is selected on the control cross, the corresponding axle is positioned at a fixed velocity. This velocity can be changed by simultaneously pressing a key on the keyboard. First press the key on the keyboard, and then click a direction button in the control cross.

The following keys can be used:

Shift key

Increases the velocity by a fixed factor

Control key

Decreases the velocity be a fixed factor for precise, fine positioning

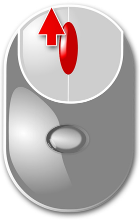
ATTENTION

Danger of damage through collision with the sample holder! For example, if there is a needle in a holder, ensure that moving the rotary or linear axis will not cause collisions when the tool tip is lowered.

11.5 Positioning Map

11.5.1 Introduction

With the positioning map you can position the positioning system in the XY plane by clicking points on the positioning map. The magnification of the positioning map can be changed via the context menu in fixed steps or continuously adjusted with the mouse wheel.

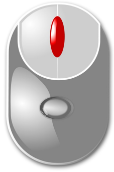


Increase magnification (zoom in)

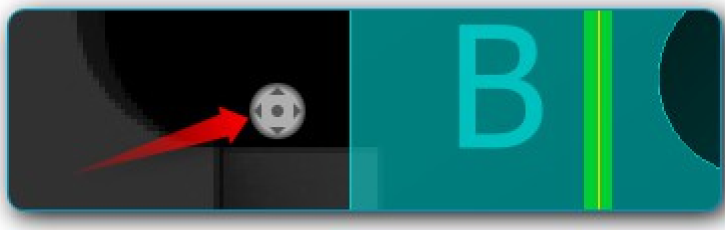
Decrease magnification (zoom out)

For navigation within the positioning map you can also use the mouse. You can move the currently visible area with the two scroll bars on the right and bottom of the map Alternatively, you can also enable free mouse navigation mode. To do this, follow these steps:

(1) Click the middle mouse button within the positioning map



(2) It is now displayed a navigation cross and the free navigation mode is activated



(3) You can now move the mouse to scroll the visible area of the positioning map. The further you move the mouse away from the on-screen navigation cross, the faster the positioning map is scrolled.

11.5.2 Positioning modes

The positioning map can be used in two different modes:

- Free positioning
- Well positioning

Switch between the two modes with the *Positioning Map* button in the toolbar or with the context menu of the positioning map.

The current position of the rotAXYS device is indicated by a green cross on the positioning map (Figure below).

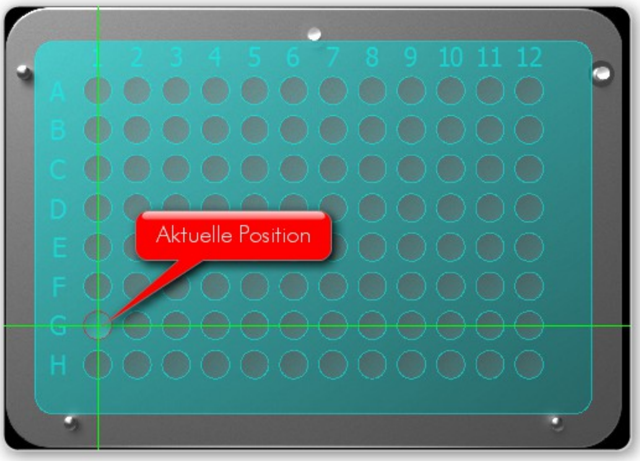


Figure 183: Actual position indicator

IMPORTANT

In order to avoid collisions, the vertical axis always moves automatically into its topmost position before travelling to a position.

11.5.3 Free positioning

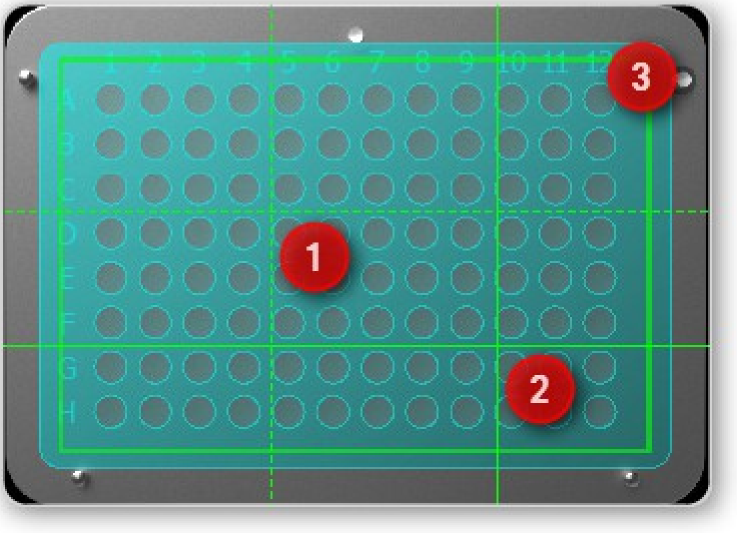


Figure 184: Free positioning mode

With free positioning, you can travel to all positions within a permissible area 3. The permissible area is indicated by a green border (see Figure above). When you move the mouse, the potential target position 1 is indicated by a target cross consisting of two dashed lines. When you click the left mouse button on the target position, the positioning system travels to the selected position. The indicator of the current position 2 then moves to the target position.

11.5.4 Well positioning

In this mode, simply click in one of the existing wells to position. As soon as you have clicked the target well, the system automatically moves to the centre of the selected target well.

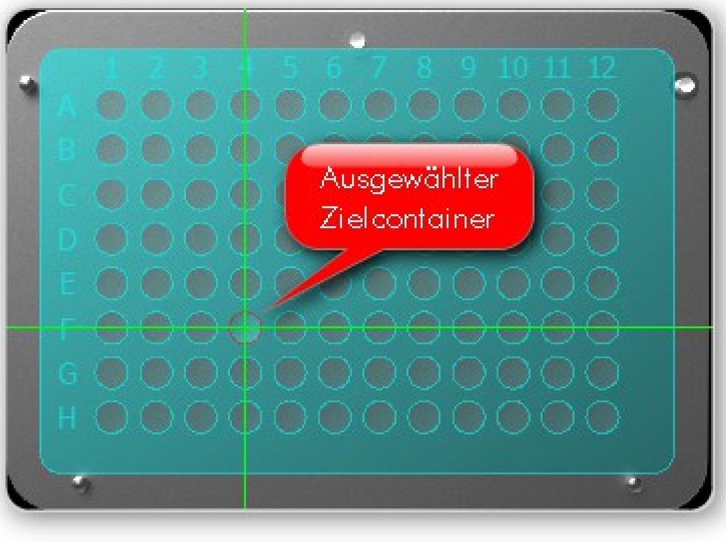


Figure 185: Well positioning

The selected target container is marked with a coloured marker.

11.6 Configuring Well Plates

11.6.1 Introduction

The *Positioning Map* always shows the currently configured well plate. You can configure new well plates in the software and assign it to a rotAXYS device. To do this, select the main menu item *Edit → Configure Well Plate* (see figure below).

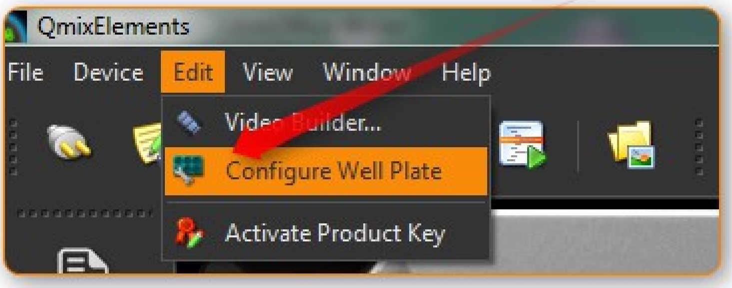


Figure 186: Show well plate configuration dialogue

The configuration dialogue for well plates appears. This configuration dialogue is structured as follows (see figure below):

- 1 list of available well plates
- 2 configuration parameters of current plate
- 3 graphical representation of the current well plate

11.6.2 Selecting an existing well plate

If the list of available well plates 1 already contains a suitable plate, simply select it by clicking on it and confirm the selection by clicking the *OK* button in the configuration dialogue.

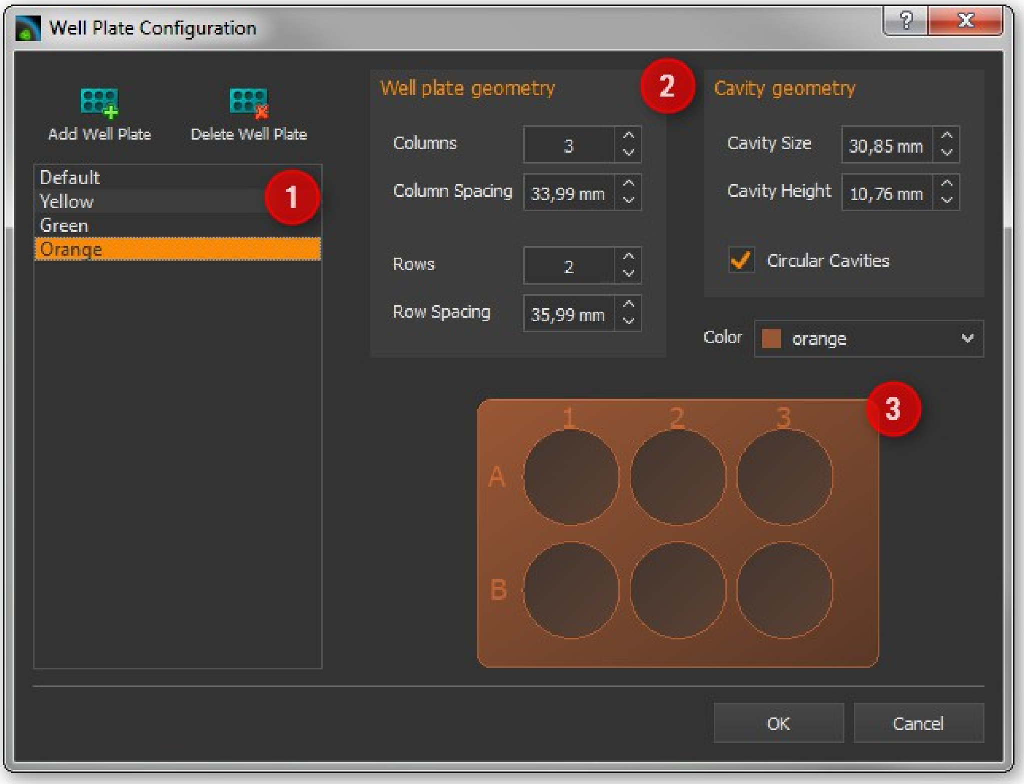


Figure 187: Configuration dialogue for well plates

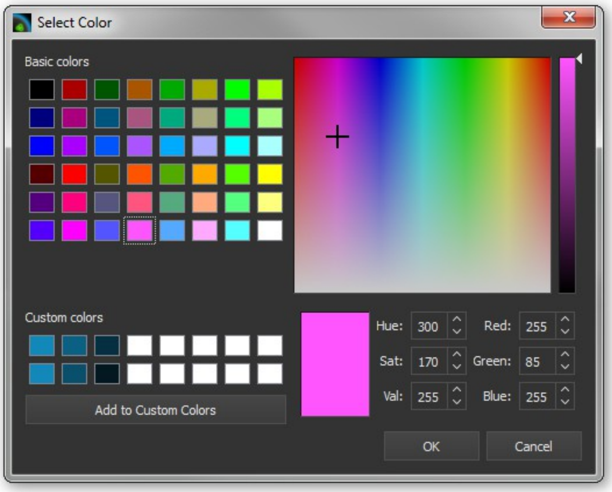
11.6.3 Creating a new well plate

To create a new well plate, perform the following steps:

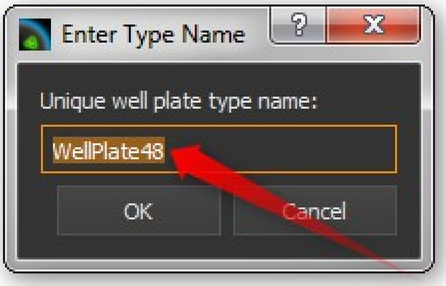
(1) First you need to configure the well plate geometry in the *Well plate geometry* panel 2. You can configure the number of *Rows* and *Columns* and the space between rows (*Row Spacing*) and columns (*Column Spacing*). You will find the required data in the data sheet of the well plate.

(2) Next, configure the geometry of the cavities in the *Cavity geometry* panel. Define the size of the cavities (*Cavity Size*) and their depth (*Cavity Height*). Use the selection field *Circular Cavities* to determine whether the new plate should have round or square cavities.

(3) Finally define the colour of the new well plate. Choose the desired colour in the *Color* selection box. If the right colour is not available, select the entry *More...* The application shows a colour selection dialogue (see figure below) that allows you to select any colour.



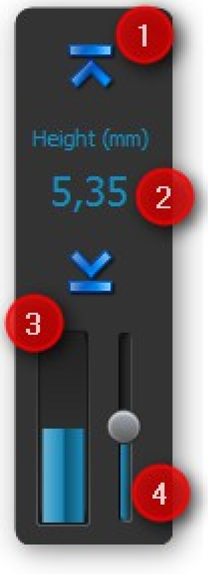
(4) Now click the *Add Well Plate* button to add the new plate to the list of available well plates. In the input dialogue that appears (see figure below), enter a unique name for the new well plate and press *Enter* or click *OK*.



(5) The new well plate will be added to the list and can be selected from now on.

11.7 Z panel for Positioning the Z axis

11.7.1 Overview



The Z panel on the right of the positioning map shows you the current position of the vertical axis, and gives you a number of ways of positioning the vertical axis. These are:

1 Buttons for predefined positions

2 Height indicator

3 Graphical height indicator

4 Positioning slider

11.7.2 Buttons for predefined Positions

The two raise and lower buttons simplify quick positioning of the vertical axis to previously defined positions. When you click one of the two buttons, the vertical axis travels at maximum velocity to the position previously defined for this button.

To assign the current position, right mouse click one of the two buttons, and then select *Assign actual position* from the context menu. (see Figure below).

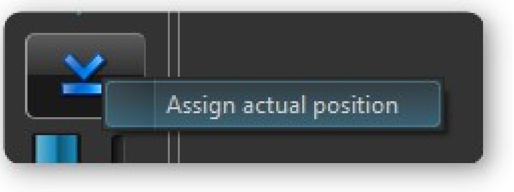


Figure 188: Assign actual position

The assigned position is saved in the configuration data, and is still present when the software is restarted.

11.7.3 Height Indicator

The numerical height indicator shows you the current height of the axis in millimetres. A bar in the graphic display shows the current position of the axis within the range of travel.

11.7.4 Positioning Slider

You can move the vertical axis quickly and easily to a specific height with the positioning slider. Do this by moving the slider to the desired height. As soon as you release the left mouse button, the axis travels to the selected position.

11.8 Status Bar

You can read the current XY position and the current status of the device in the status bar of the rotAXYS working area at all times.

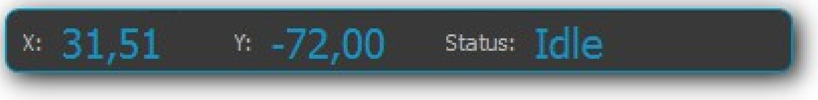
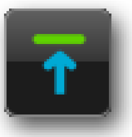


Figure 189: rotAXYS Status Bar

11.9 Z-Axis safety stroke

You can use this button in the toolbar to switch on and off the safety stroke of the Z-Axis. If it is active, then the Z-axis moves to a safe height before each movement in the XY plane. This prevents collisions with sample vessels or other objects on the worktable.



By pressing this button, you define the current position of the Z-axis as the clearance height. If the safety stroke is active, the positioning system will automatically go to this height before any XY positioning.

11.10 Zero angle configuration

The rotAXYS 360 axis system has an operation angle which is theoretically greater than 360 degrees. In order to prevent entanglement of the hose for fluid dosing, the turning range is limited to 360 °.

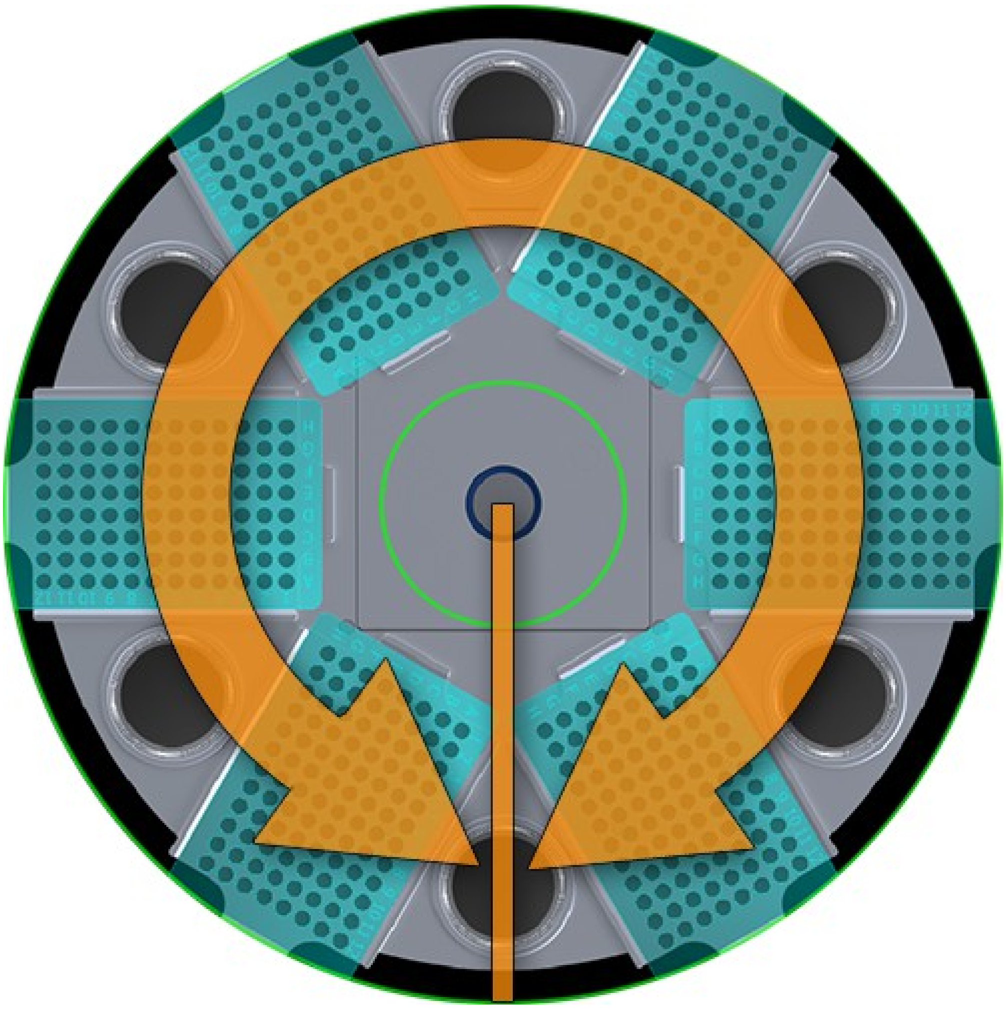
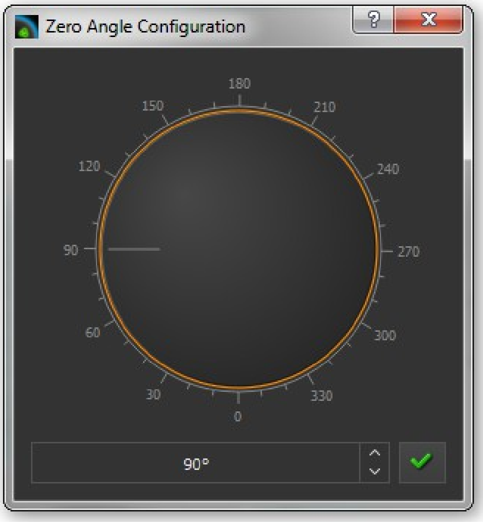
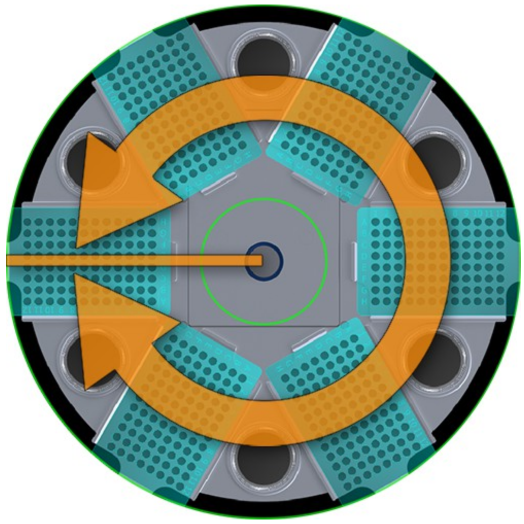


Figure 190: Zero angle configuration

The zero angle ist the angle that will never be crossed by the positioning system. It is the start- and end-point of the 360° rotation range. If this angle is situated between two sample containers that you would like to process, the positioning system will choose the longer route in the opposite direction, so as not to cross the zero angle.

You can move the zero angle to a different angular position in order to circumvent this problem. Press the button for zero angle configuration (image left) to set a new zero angle.





In the dialog which is now displayed (see illustration above), you can freely set the zero angle in the range from 0 - 360 °.

11.11 Positioning with the Space Navigator

11.11.1 Introduction

The Space Navigator is an intuitive input device which allows you to control all axes (rotation, radius and vertical axis) simultaneously.

To use the Space Navigator, connect it to an available USB port on your PC. Latest drivers can be found on our internet page:

http://www.cetoni.de/englisch/service/downloads_software.html

or on disk (CD or USB stick) you received when you purchased the software. Install the driver to use the Space Navigator.

Position the Space Navigator device, so that the cable leads away from you and that the *3D Connexion* label is readable from your position.

Figure 191: Space navigator installation

11.11.2 Positioning rotAXYS with Space Navigator

11.11.2.1 Positioning rotary axis

In order to position the rotary axis, rotate the Space Navigator cap clockwise or counter-clockwise.

Figure 192: Positioning rotary axis with Space Navigator

The speed of the axes can be determined by the strength of the deflection of the Space Navigator cap. The harder you push or twist the cap, the faster the axes are moving. By a slight deflection of the cap, you can very finely and precisely control axis speeds.

HINT

The more you deflect the Space Navigator cap, the higher are the axes speeds.

11.11.3 Positioning radius axis

To move the radius-axis, slide the cap of the Space Navigator to the front or rear. To move the radius-axis to the outer radius, press away the control element from you. To move the radius axis to the centre of the rotation axis, pull the cap to your direction.

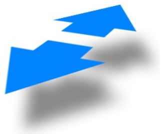
Figure 193: Positioning radius axis with Space Navigator

11.11.4 Vertical axis positioning

To position the Z-axis, press down the cap or pull it to the top. The Z-axis is moved in the direction in which you move the control element of the Space Navigator.

Figure 194: Positioning vertical axis with Space Navigator

Rotates the rotary axis clockwise or counter-clockwise



Moves the radius axis away from or towards to the centre of rotation

Move the vertical axis up or down

IMPORTANT

Don't use the Space Navigator if the software moves the positioning system (e.g., during active script execution), in order to not interrupt movements.

11.11.5 Positioning neMAXYS with Space navigator

11.11.5.1 Positioning X-axis

To control the X-axis movement, slide the Space Navigator's control in the desired direction. Push the controller to the left, to move the X-axis leftwards and push it to the right to move the axis rightwards.

Figure 195: Positioning X-axis with Space Navigator

The speed of the axes can be determined by the strength of the deflection of the Space Navigator cap. The harder you push or twist the cap, the faster the axes are moving. By a slight deflection of the cap, you can very finely and precisely control axis speeds.

HINT

The more you deflect the Space Navigator cap, the higher are the axes speeds.

11.11.5.2 Positioning Y-axis

To move the Y-axis towards the back of the device, press the controller forward. To move the positioning system towards the front of the device, pull the controller towards you.

Figure 196: Positioning Y-axis with Space Navigator

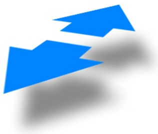
HINT

You can control both directions at the same time if you move the controller forward or backward and to the left or right at the same time.

11.11.5.3 Vertical axis positioning

To position the Z-axis, press down the cap or pull it to the top. The Z-axis is moved in the direction in which you move the control element of the Space Navigator.

 Move X-axis to the left or right

 Move Y-axis forwards or backwards

 Positioning vertical Z-axis

IMPORTANT

Don't use the Space Navigator if the software moves the positioning system (e.g., during active script execution), in order to not interrupt movements.

11.12 Single Axis Control

In addition to the control of multi-axis systems the software also supports control of individual axis devices. This may be necessary if your axis system has additional tool axes (like second Z-axis or rotation axis) which are not part of the XYZ-multi-axis control.

You can show / hide the window for single axis control via the main menu item *Window → Single Axis Control* (figure below).

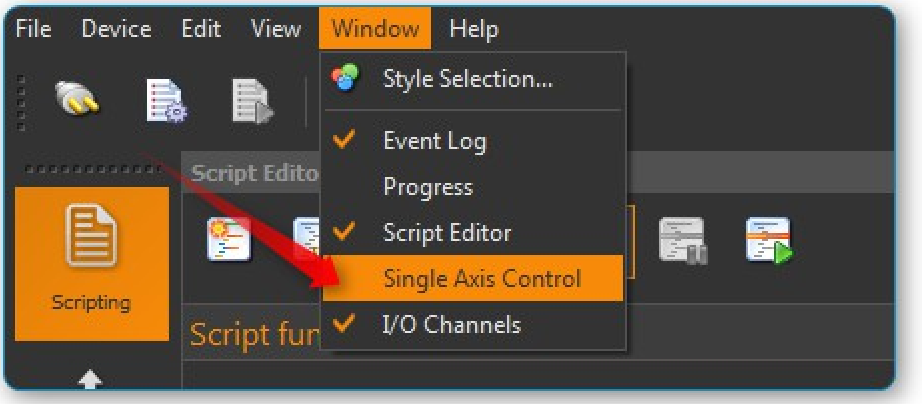


Figure 197: Display of single axis control panel

HINT

The single axis control window is a tool window and can be freely placed in the application window and docked in all four side areas.

The following control elements are available in the single axis control window:

1 Drop-down list *Axis Device* for selection of axis device to control.

2 In the *Position* area a progress bar shows the actual position in the range between minimum and maximum axis position. You can move the axis by pressing the plus / minus buttons. The movements continues as long as the buttons are pressed and stops immediately if a button is released.

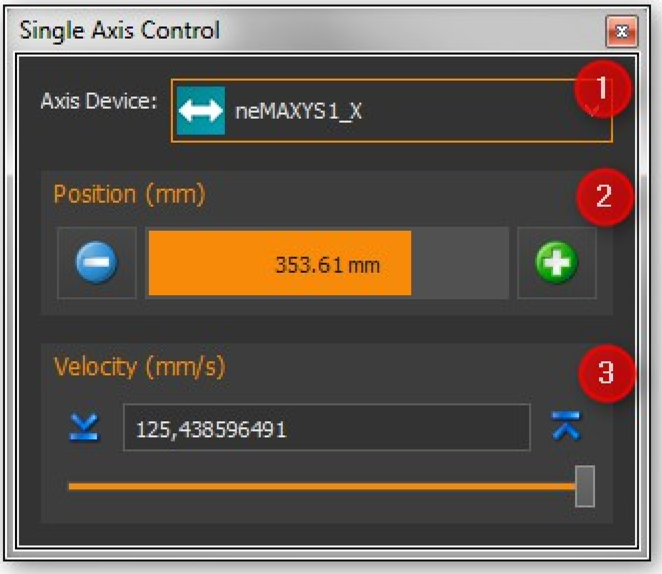


Figure 198: Single axis control panel

3 In the *Velocity* area you can adjust the speed of axis move.

HINT

With the Move Axis script function you can control individual axes from a script program.

11.13 Additional Functions

11.13.1 Calibrating axis system

If the travel path of an axis is blocked (for example in the event of a collision), the stepper motors in the device may "lose" steps, so that the position values are no longer displayed correctly. In this case, the axis system has to be calibrated. To start calibration, select the point in the main menu *Device → rotAXYS -> Find Home* (see Figure below). If you have a neMAXYS system, you can reach the calibration via *Device → neMAXYS -> Find Home*

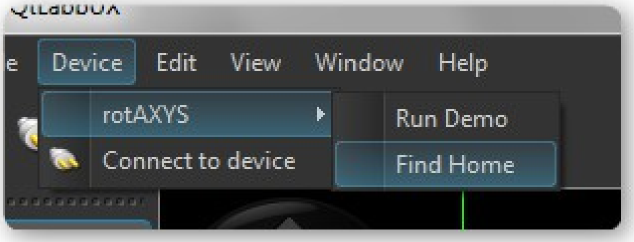


Figure 199: Calibrate positioning system

11.14 Positioning Script Functions

11.14.1 Introduction

The rotAXYS / neMAXYS plugin offers a number of script functions required for integrating the cetoni positioning systems in the Qmix scripting system. The following script functions are available:

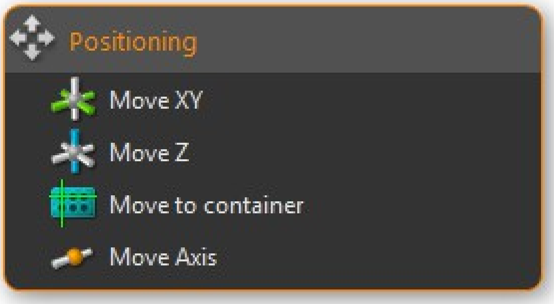
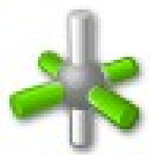


Figure 200: Positioning Script Functions

11.14.2 Move XY



This function is used in a Qmix script to position an XY axis system in the XY plane. You can select an axis system 1 and the target position (X and Y positions) 2 in the configuration area of this function. Alternatively, you can move the axis system manually to a specific position (for example with the positioning map - see Section 11.5), and then assign the current position values to the script function with the *Capture Actual Position* button (see Figure below).

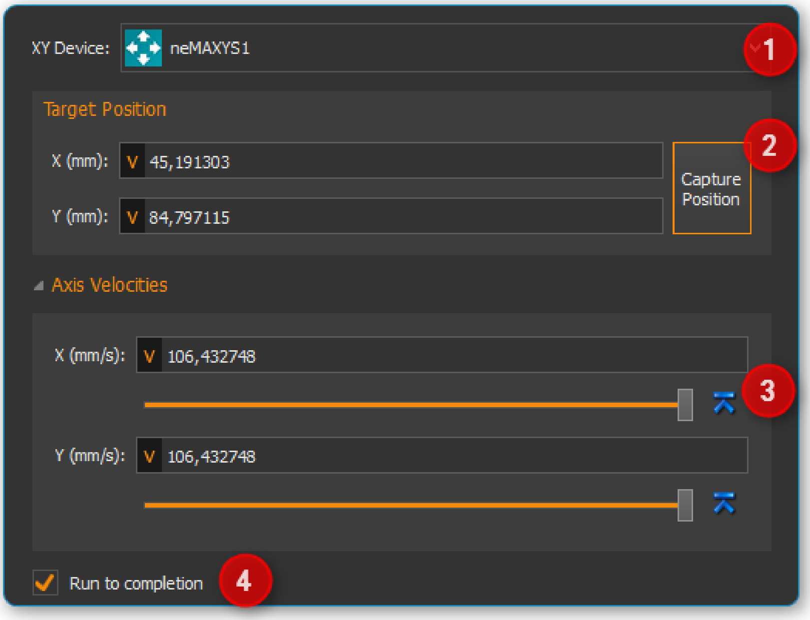


Figure 201: Move XY script configuration

If you make no further adjustments, all positionings are carried out with the maximum possible speed. If you want to change velocities, you need to expand the *Axis Velocities* control by clicking it. This control panel 3.contains all control elements for entering velocity values for each single axis.

Use the *Run to completion* 4 check box to determine whether the next function

- will start immediately after the current positioning move started (*Run to completion* not checked)
- or if the next function is called after the target position has been reached (*Run to completion* checked).

11.14.3 Move Z

This function can be used to position the vertical axis (Z-axis) of an axis system. You can enter the axis system 1 and the position parameters (Z position 2 and velocity 3) in the configuration area of this function. Alternatively, you can move the Z-axis manually to a specific position (for example with the Z panel - see Section 11.7), and then assign the current Z position to the script function with the *Capture Actual Position* button.

Figure 202: Move Z script configuration

If you make no further adjustments, the positioning is carried out with the maximum possible speed. If you want to change velocities, you need to expand the *Velocity* control by clicking it. Another click on this element hides the velocity controls again.

Use the *Run to completion* 4 check box to determine whether the next function

- will start immediately after the current positioning move started (*Run to completion* not checked)
- or if the next function is called after the target position has been reached (*Run to completion* checked).

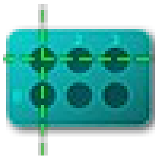
HINT

If ***Run to completion*** *is not activated, you can move several axes simultaneously. You can e.g. start positioning the XY-system, and while the XY-axes are moving, you can perform positioning with the Z-axis.*

IMPORTANT

The input fields for configuring the speeds are hidden by default, and must be displayed by clicking the ***Velocity*** *label.*

11.14.4 Move to container



This function is used to move an axis system in the XY plane to the centre a specific container (beaker, fluid tank, cavity of a well plate).

First you select the positioning device in the *XY Device* combo box 1 (figure below). Then you select the target container in the container selection combo box 2. If the container has several cavities (e.g. the wells of a well plate), enter the column and row of the cavity in the *row* and *column* input fields 3. The numbering starts at zero. That means, for a microwell plate with 12 columns and 8 rows, you can enter values from 0 to 11 in the column input field and values from 0 to 7 in the row input field. If the container has only one single cavity (e.g. beaker or fluid tank) the input fields for *row* and *column* are greyed out.

HINT

You can use script variables in the row and column input fields.
This allows you to process all wells of a well plate in a loop using the loop counter variables.

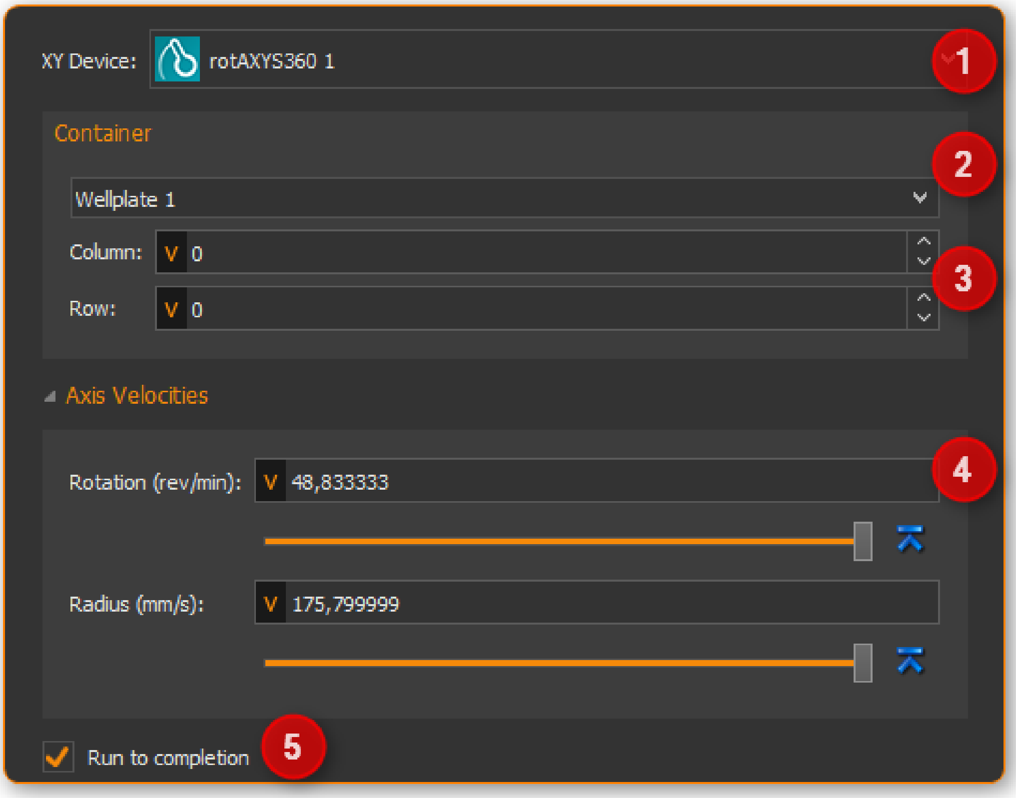


Figure 203: Move to container configuration

If you make no further adjustments, all positionings are carried out with the maximum possible speed. If you want to change velocities, you need to expand the *Axis Velocities* control by clicking it. This control panel 4 contains all control elements for entering velocity values for each single axis.

Use the *Run to completion* 5 check box to determine whether the next function

- will start immediately after the current positioning move started (*Run to completion* not checked)
- or if the next function is called after the target position has been reached (*Run to completion* checked).

11.14.5 Move Axis

This function can be used to position a single axis device. Use the drop-down list *Axis device* 1 to select the axis to me moved. You can enter the parameters (position 2 and velocity 3) in the configuration area of this function. Alternatively, you can move the axis manually to a specific position (for example with the Single Axis Contol Panel – see Section 11.12), and then assign the current position to the script function with the *Capture Position* button.

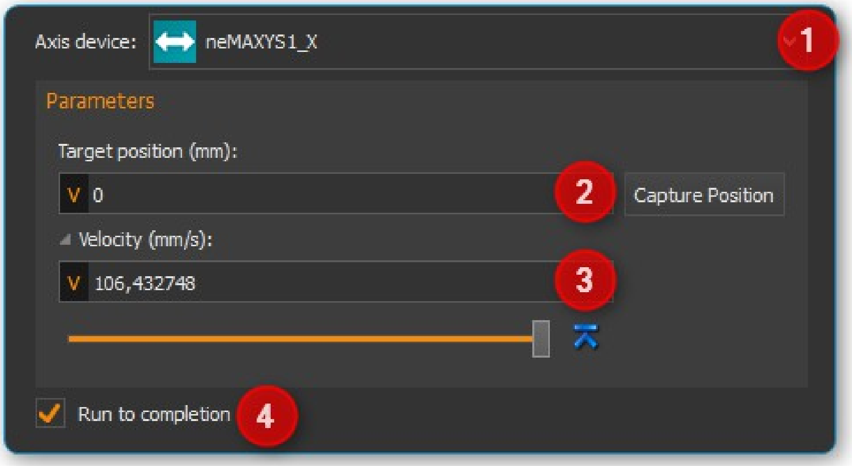


Figure 204: Move Axis script configuration

If you make no further adjustments, the positioning is carried out with the maximum possible speed. If you want to change velocity, you need to expand the *Velocity* control by clicking it. Another click on this element hides the velocity controls again.

Use the *Run to completion* 4 check box to determine whether the next function

- will start immediately after the current positioning move started (*Run to completion* not checked)
- or if the next function is called after the target position has been reached (*Run to completion* checked).

IMPORTANT

*The input fields for configuring the speeds are hidden by default, and must be displayed by clicking the **Velocity** label.*

12 Tubing Pump Plugin

12.1 Introduction

Press the button *Tubing Pump* in the side bar to switch to the operating window (Work bench) of the tubing pump plug-in.

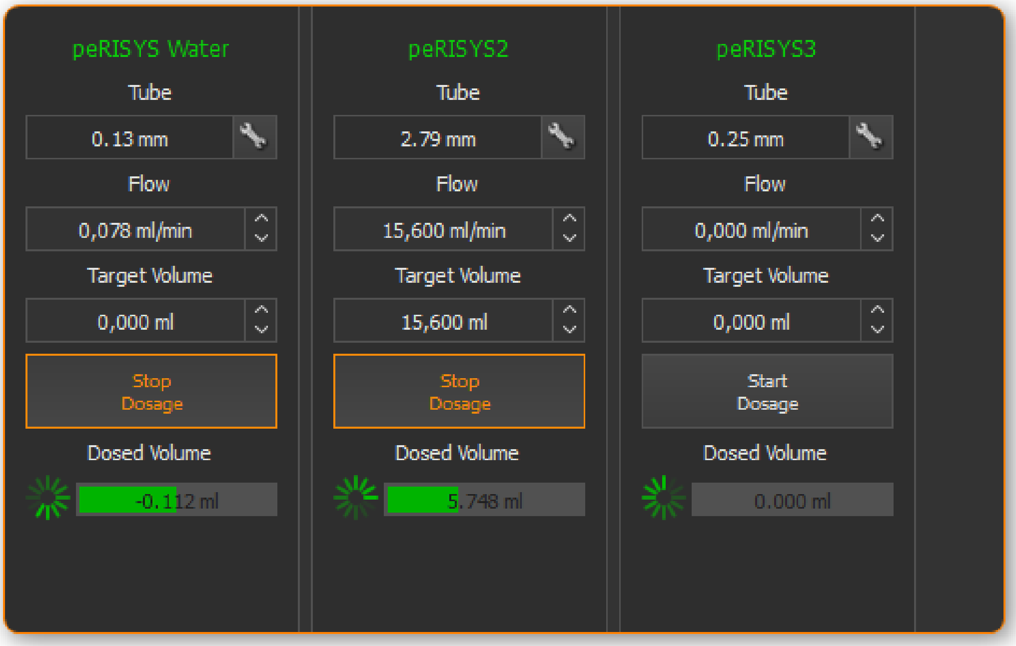
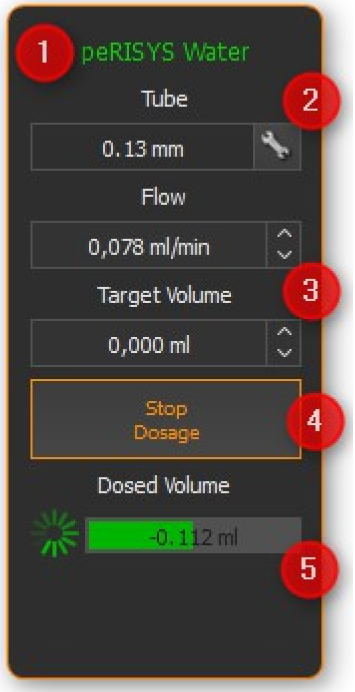


Figure 12.1: Tubing pump work bench.

The work bench shows the operating panels for all tubing pumps.

12.2 Tubing Pump Operating Panel

12.2.1 Operating Parameters



4 Pump name (customizable).

5 Tubing configuration (inner diameter).

6 Target values (flow rate, volume).

7 Start/stop dosing button.

8 Activity and dosed volume indicators.

12.2.2 Pump Name Customization

You can change the description of each pump at any time to reflect the function or fluids of the respective pump. To change the description, directly click the description 1 and type in the new name.

12.2.3 Tubing Configuration

The input box *Tube* 2 shows the inner diameter of the currently configured tubing. Click on the wrench symbol right of the input area to change the tubing diameter configuration. This will open the *Tube Selection* dialog box with a list of pre-configured tubings.

The list is a table that shows the inner tube diameter in the first column (*Tube*). The second column (*Milliliters per pump head revolution*) indicates the actuated volume per revolution of the pump head in milliliters.

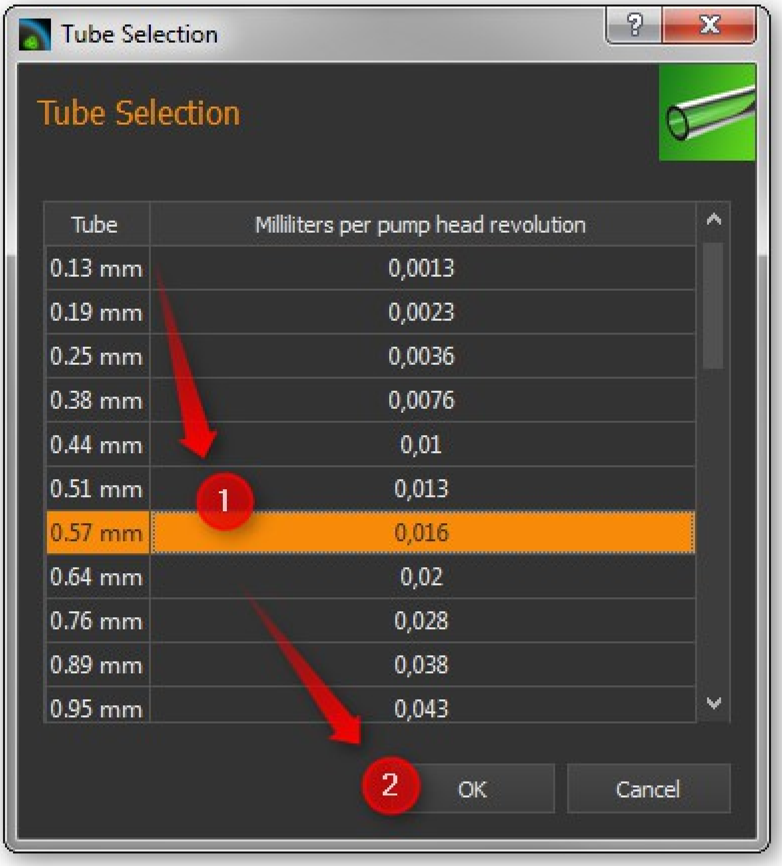


Figure 12.2: Tubing selection dialog.

Select the appropriate type of tube from the list 1 and confirm your selection 2.

12.3 Manual Dosing

Proceed as follows to configure a manual dosing task:

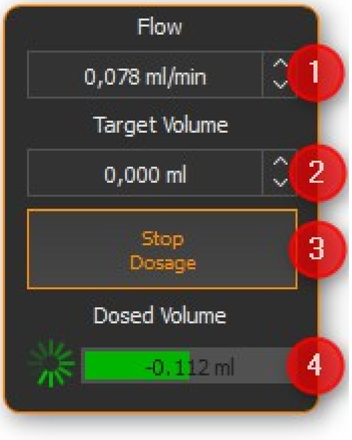


Figure 12.3: Manuel dosing.

1 Enter your target flow rate in the *Flow* input box. A negative flow value causes a change in the rotating direction of the pump head. So you can switch between dispensing and aspiration by switching the sign of the entered flow value.

2 Now enter the volume to be dosed in the input box *Target Volume*. If you set the target volume to zero, the pump will be working in flow mode, i.e., the pump will work until manually stopped. A negative volume causes a change in the rotating direction of the pump head.

3 Start the pump by clicking *Start Dosage* and stop it by clicking the same button again.

4 The *Dosed Volume* info box will show the progress of the pumping process.

HINT

The software treats a dosing volume set to zero as unlimited *continual flow. This is, once started, the pump will continue to operate until it is manually stopped by the user.*

HINT

You can switch between dispensing and aspiration by switching the sign of the entered flow or volume value.

12.4 Script Functions

12.4.1 Introduction

The plugin offers a number of script functions which can be used to program automatic sequences or for time-controlled dosing of liquids. The following script functions are available:

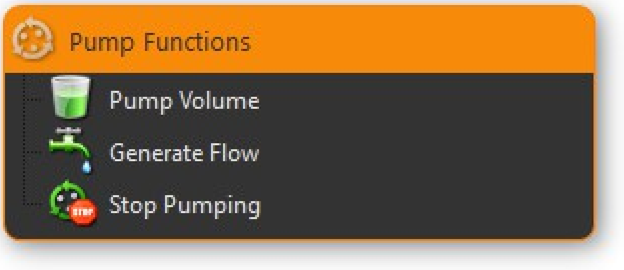


Figure 12.4: Pump script functions

12.4.2 Pump Volume

With this function, you can pump a specific volume at a precisely defined flow rate. You can set all the parameters in the configuration area, for example the dosing module 1, the volume to be dosed, and the flow rate 2.

You can also activate or deactivate the *Run to completion* 3 parameter in the configuration area. When *Run to completion* is activated, the script execution is not continued until the complete volume has been dosed and the dosing process has ended. If this parameter is not active, the dosing is started, and then the next script function is executed immediately. This enables you, for example, to start a number of dosing modules almost simultaneously.

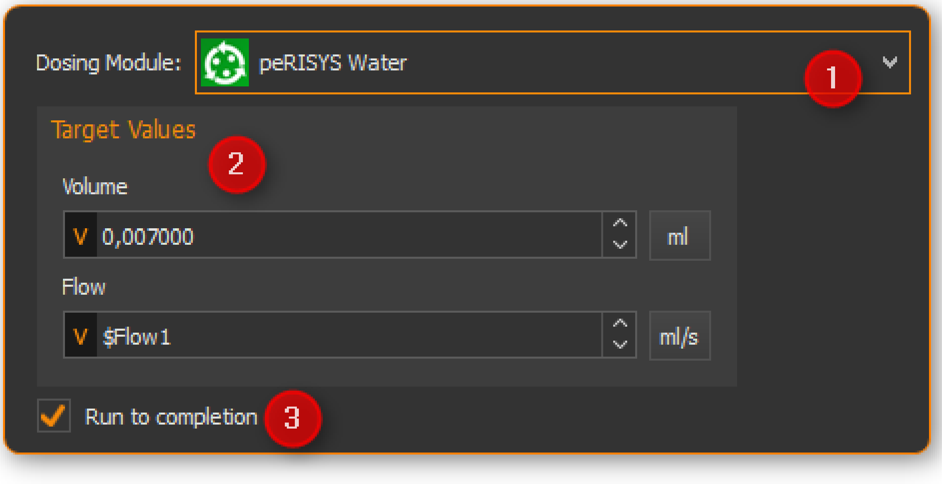
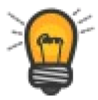


Figure 12.5: Pump Volume Script Parameter



HINT

All the pump functions support the use of variables. That means, in all input fields marked with a coloured V in the script configuration panel (e.g. flow rate and volume) you can enter variables.

12.4.3 Generate Flow

This function is used to generate a constant flow rate. In the configuration area, you can select the dosing module and set the flow rate. If the *Run to completion* parameter is active, the next script function is not executed until the module has stopped or reached one of the limit positions.

12.4.4 Stop Pumping

You can immediately stop an active dosing process of a pump with this function.

13 LED Array Plugin

13.1 Introduction

Press the button *LED Array* in the side bar to switch to the operating window (workbench) of the LED Array plug-in (figure below).

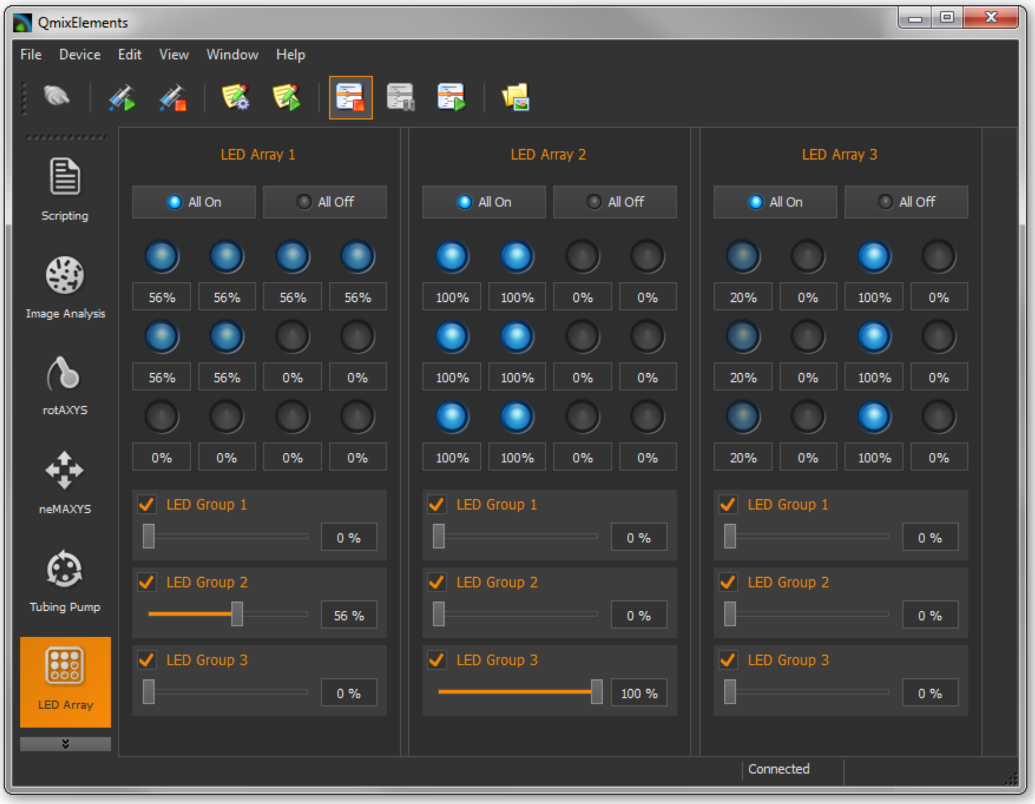


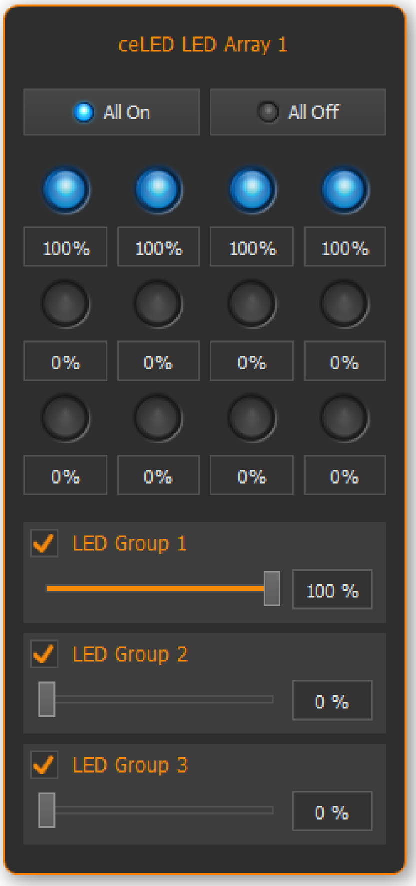
Figure 13.1: LED Array work bench.

The workbench contains a separate control panel for each connected LED array device:

13.2 Hardware Version

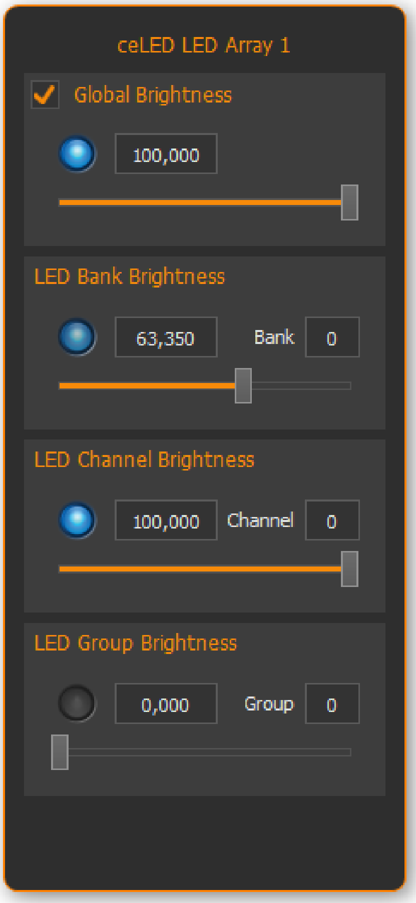
Depending on the LED array hardware version a corresponding LED array control panel is displayed for every LED array. The following two hardware versions are supported:

LED Array Hardware V1



Hardware version 1 supports 12 independent LED array channels with a resolution of 100 steps to adjust the brightness.

LED Array Hardware V2



Hardware version 2 supports a virtually infinite number of independent LED channels with a resolution of 4096 steps to adjust the brightness.

13.3 LED Array Control Panel V1

13.3.1 Control Elements

1 Caption (customizable).

2 all LED channels on / off

3 adjust the brightness of individual LED channels (0 - 100%)

4 adjust the brightness of multiple LED channels (LED group) at the same time

13.3.2 Caption Customization

You can change the caption of each LED array control panel at any time. To change the caption, directly click the caption label 1 and type in the new name. This name will be saved and reloaded the next time you start the software.

13.4 Setting brightness of individual LED channels

You can set the brightness of each LED channel by entering the value directly in the field below the LED 2 (0 - 100%) or by clicking the right mouse button on an LED and dragging the brightness slider 1 (see figure below).

Figure 13.2: Setting brightness of individual LED channels

Click with the left mouse button on an LED to switch individual LED channels on and off.

Figure 13.3: Switching LED channel on / off

As all LED channels are normal analog output channels in the QmixElements software, you can also adjust the brightness of individual LED channels via the *I/O Channels* window (figure below).

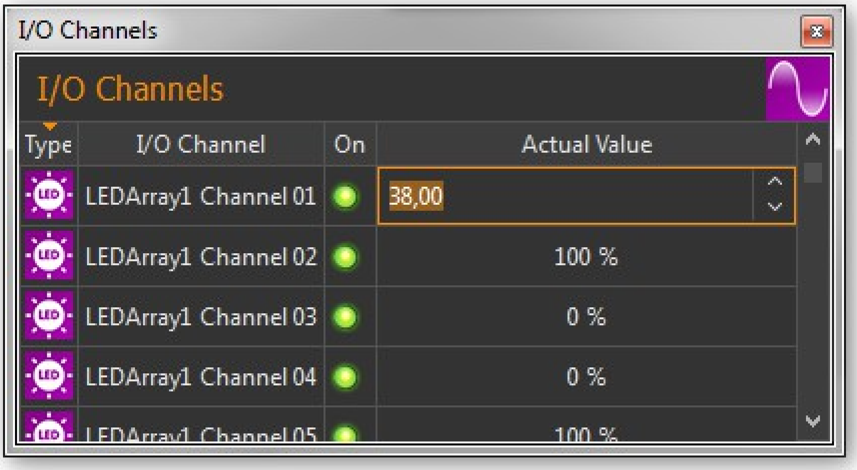


Figure 13.4: Setting LED brightness in I/O Channel window

13.5 LED Group Control

13.5.1 Introduction

When multiple LED channels should work synchronously, you can combine these channels into groups and control them together. The control panel contains three sliders to adjust brightness of three different LED channel groups.

13.5.2 Configuring LED channel groups

To select the channels to be combined into a group, click the right mouse button in the control area of a certain group to show its context menu. Then select the menu item *Configure Group Channels* (see figure below).

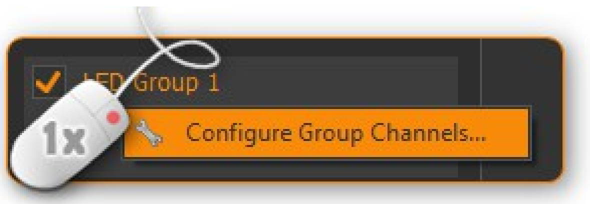


Figure 13.5: Show LED group configuration dialog

In the LED group configuration dialog (figure below) you can select all LED channels to be grouped together. Check each channel to be part of this group 1 and confirm your selection by clicking *OK* 2.

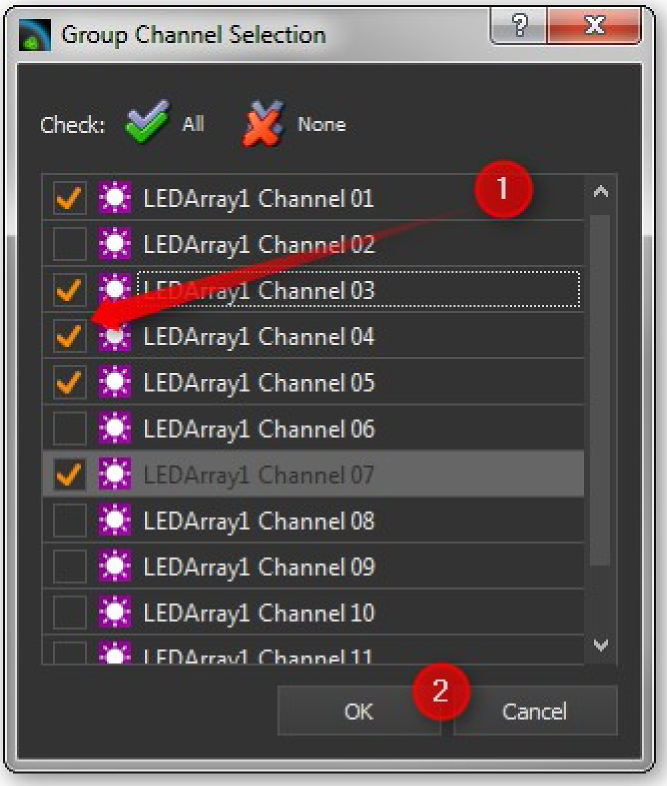


Figure 13.6: LED group configuration dialog

13.5.3 Controlling LED groups

You can change the brightness of an LED group with the slider 1 or by entering the brightness value in the input box 2 (see figure below).

Figure 13.7: LED group control

With the check mark in the upper left corner 3, you can switch all LED channels in this group on /off simultaneously.

13.6 Configuring Standby-Timer

The LED Array has a standby timer functionality. This means, all sectors of the LED Array are automatically switched off after the last action of the user in the software and after the standby time is over. Every change in parameters in the software resets the standby timer and reactivates the array.

IMPORTANT

The standby timer value is saved in the LED Array. I.e. even if the connection to the PC is lost, the LED array is switched off after the standby time is over.

To configure the standby time, click with the right mouse button on a blank area in the LED array control panel to display the context menu. Then select the menu item *Configure Standby Timer* (figure below).

Figure 13.8: Show standby timer configuration window

In the configuration dialog displayed (figure below), you can configure the hours, minutes and seconds of the standby timer. Your changes are accepted by clicking on *OK* and the standby time will be saved in the device.

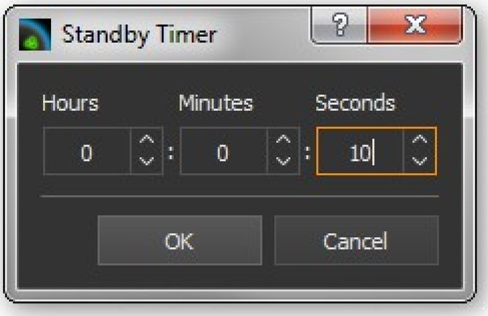


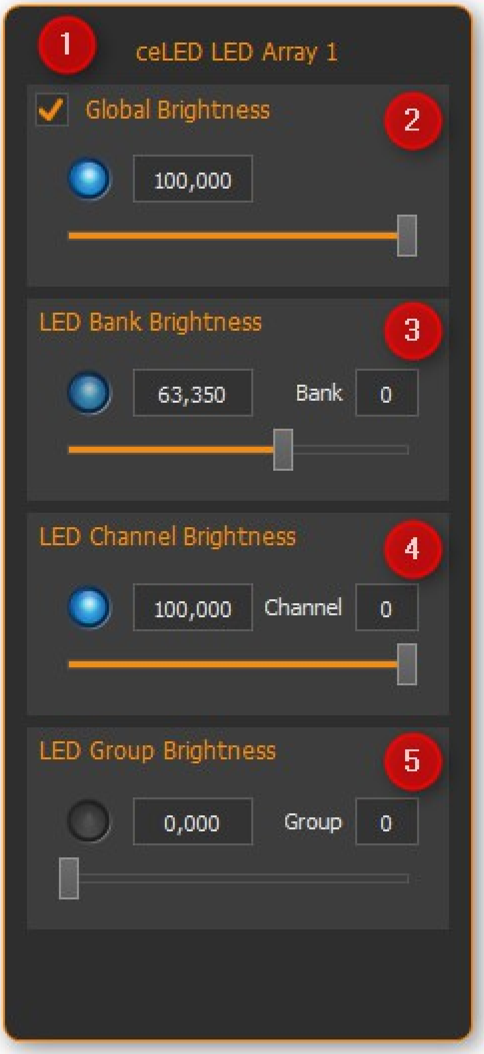
Figure 13.9: Standby timer configuration window

IMPORTANT

Enter a value of 0 for Hours, Minutes and Seconds to deactivate the standby timer.

13.7 LED Array Control Panel V2

13.7.1 Control Elements



1 Caption (customizable)

2 adjust the global brightness of all channels and switch the Enable signal

3 adjust the brightness of a complete bank

4 adjust the brightness of individual LED channels (0 – 100%)

5 adjust the brightness of a user-specific LED group consisting of multiple channels

13.7.2 Caption Customzition

You can change the caption of the LED array at any time. To change the caption, click the caption label 1 and enter a new name for the LED array. This name will be saved and reloaded the next time you start the software.

13.8 Setting the Global Brightness

You can set the global brightness of all LED channels at the same time by using the slider 2 or the input field of *Global Brightness*. With the check mark in the top left corner 1 you can switch the global Enable signal of the LED array. The array can be switched on and off this way without changing the brightness of the individual channels.

Figure 13.10: Globale brightness and Enable Signal

IMPORTANT

The Enable signal must have been activated for the LEDs of the array to light up.

13.9 Setting the Brightness of LED Banks

LED banks are device-specific and hardware-specific groups of individual LED channels in groups. LED banks group LED channels that also form a physical group in the hardware, e.g. all LEDs on one board or all LEDs of a certain type (e.g. warm white or cold white). These groups are anchored in the firmware and cannot be changed by the user.

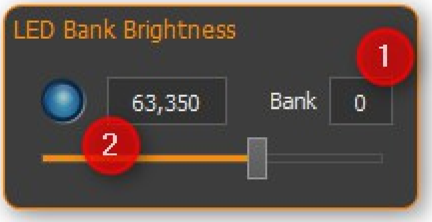


Figure 13.11: Control elements for LED bank brightness

To change the brightness of a bank, select the bank in the *Bank* 1 input field and then set the brightness of the selected bank using the slider or input field 2.

13.10 Setting the Brightness of Individual LED Channels

In *LED Channel Brightness* you can set the brightness of individual LED channels. For this purpose select the channel in the *Channel* 1 input field. You can change the brightness by using the input field 2 or the slider.

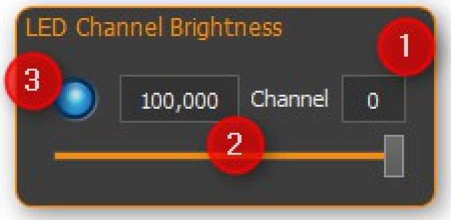


Figure 13.12: Changing the brightness of individual LEDs

To switch a channel on or off, click the left mouse button on the relevant LED 3.

As all LED channels are analogue output channels in the QmixElements software you can also change the brightness of individual channels via the I/O Channels window (see figure below).

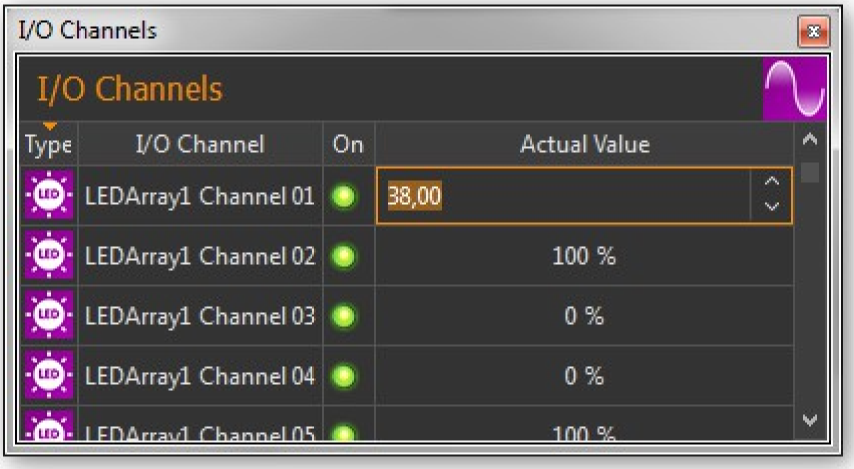


Figure 13.13: Window of I/O channels

13.11 LED Channel Groups

13.11.1 Introduction

When multiple LED channels should work synchronously, you can combine these channels into groups and control them together. The group channels can then be used to jointly control all channels of an LED group.

13.11.2 Configuring the LED channel groups

To configure LED channel groups, use the right mouse button and click on *LED Group Brightness.* Then select the menu item *Configure LED Groups* from the context menu.

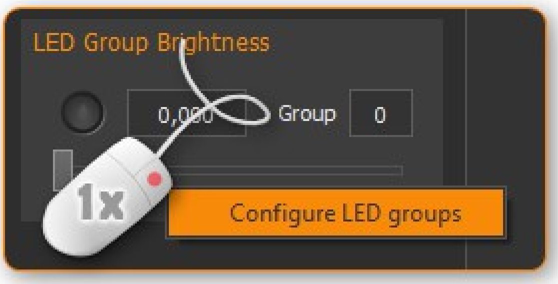


Figure 13.14: Activate LED group configuration

The dialogue for the LED group configuration will be shown now (see figure below).

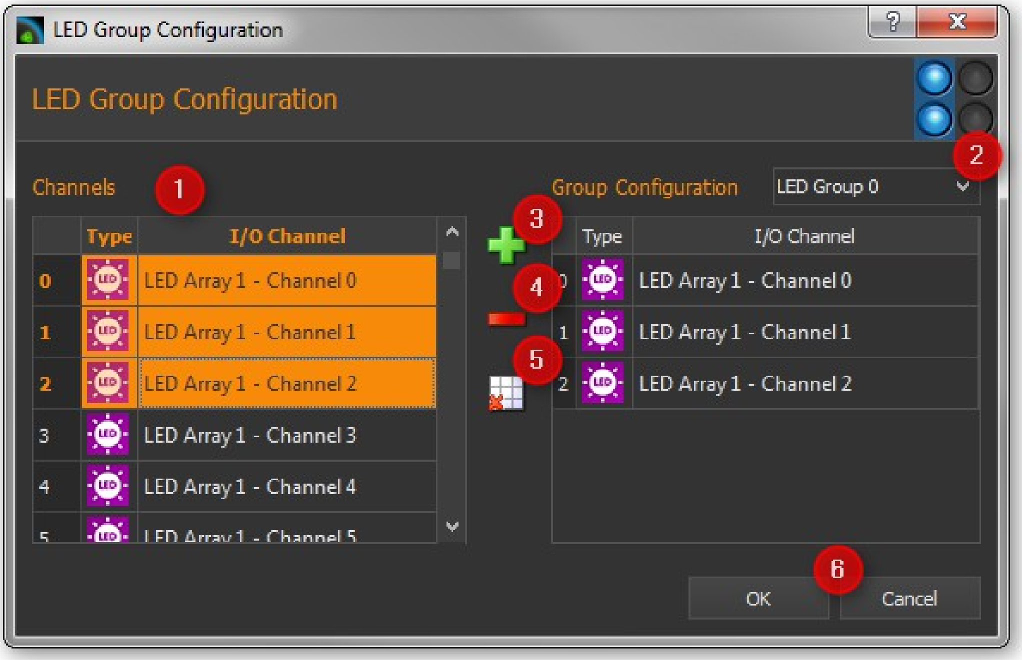


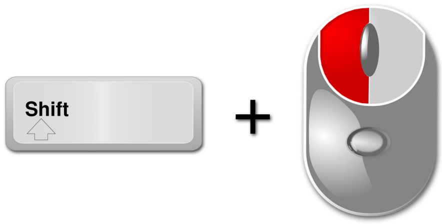
Abbildung 13.15: Konfigurationsdialog für LED Kanalgruppen

To add LED channels to a group, proceed as follows:

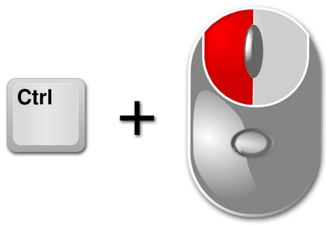
(1) First, go to the group list box 2 and select the group to be configured.

(2) Now, select from the channel list 1 the channels to be added to the group by clicking with the mouse button.

- Select individual channels by clicking.
- Select multiple connected channels by clicking with the mouse on the first channel on the first channel. Keep the *shift key* pressed while clicking on the last channel.



- Multiple independent channels can be selected by keeping the control key pressed while clicking.



(3) Now, add the selected channels to the group by clicking the *Plus button* 3. To delete individual channels from the group, select the channels from the group list and then click the *Minus* button 4. To delete all channels from the group, click the *Clear LED Group* 5 button..

(4) After having configured all groups click *OK* 6. The group configuration will then be transmitted to the device. If you want to permanently save the group configuration in the device, click *Yes* in the message window displayed (see figure below).

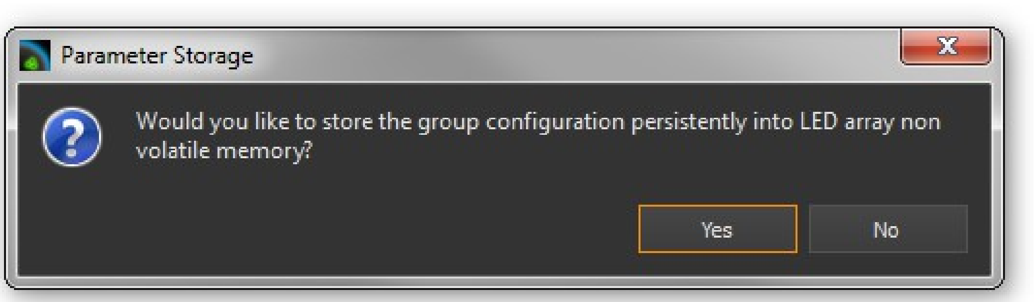


Figure 13.16: Save group configuration permanently in device

13.11.3 Setting the brightness of LED groups

To change the brightness of a group, select the group from the *Group* 1 input field and then set the brightness of the selected group using the slider or the input field 2.

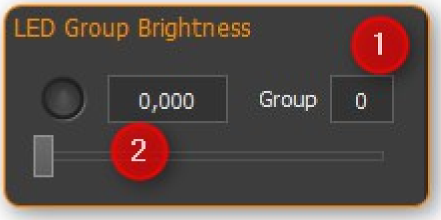


Figure 13.17: Control elements for the LED group brightness

13.12 Script Functions

13.12.1 Introduction

The LED Array Plugin provides various script functions that can be used to program automated exposure sequences or for time-controlled exposure. The script functions are available in the categories of *LED Array Functions* and *I/O Functions*.

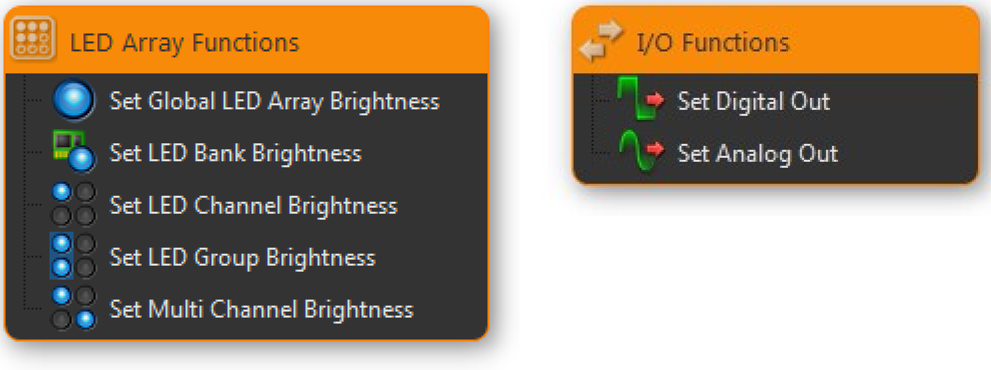


Figure 13.18: LED Array script functions

13.12.2 Global Brightness Function – *Set Global LED Array Brightness*

The global brightness of all LED channels of an LED array can be set jointly by this script function. To configure the script function, proceed as follows (see figure below):

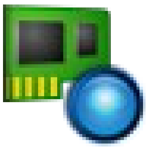
- 1 First, select the LED device from the list of devices.
- 2 Then, set the brightness (0 – 100%).

Figure 13.19: Set Global LED Array Brightness script function

HINT

This function supports script variables. Variables can be used in the Brightness field.

13.12.3 Bank Brightness function – *Set LED Bank Brightness*



You can set the brightness of entire LED banks by this script function. To configure the script function, proceed as follows (see figure below):

- 1 Select the LED Array device.
- 2 Select the bank the brightness of which is to be changed.

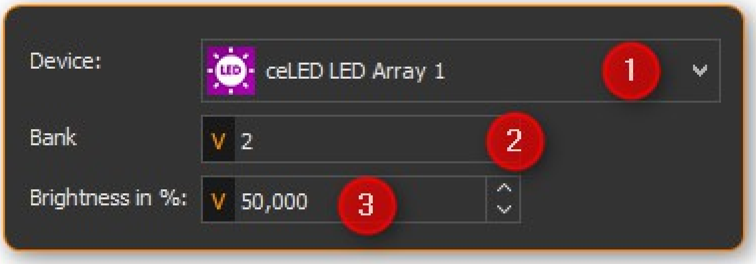


Figure 13.20: Set LED Bank Brightness script function

- 3 Set the brightness (0- 100%).

HINT

This function supports script variables. Variables can be used in the Bank and Brightness field.

13.12.4 LED brightness function – Set LED Channel Brightness

This script function can be used to set the brightness of individual LED channels. To configure the script function, proceed as follows (see figure below):

- 1 Select the LED Array device.
- 2 Select the channel the brightness of which you want to set.

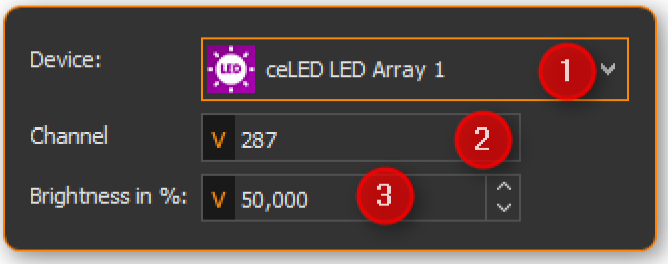


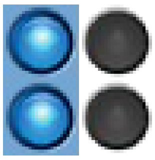
Figure 13.21: Set LED Channel Brightness script function

- 3 Set the brightness (0- 100%)

HINT

This function supports script variables. Variables can be used in the Channel and Brightness field.

13.12.5 Group brightness function – Set LED Group Brightness



You can use this function to set the brightness of an LED group in a script-controlled manner. To configure the script function, proceed as follows (see figure below):

- 1 Select the LED Array device.
- 2 Select the group the brightness of which you want to set.

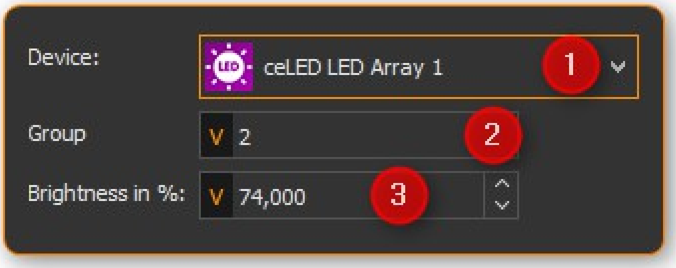
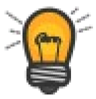


Figure 13.22: Set LED Group Brightness script function

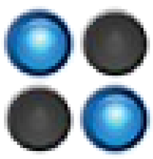
- 3 Set the brightness (0- 100%).



HINT

This function supports script variables. Variables can be used in the Channel and Brightness field.

13.12.6 Multi-channel brightness function – *Set Multi Channel Brightness*



You can use this function to set simultaneously the brightness of multiple LED channels in a script-controlled manner. To configure the script function, proceed as follows (see figure below):

- 1 Select the LED Array device.

- ❷ Set the brightness (0- 100%).

- ❸ Check the every channel of the list which is to be controlled.

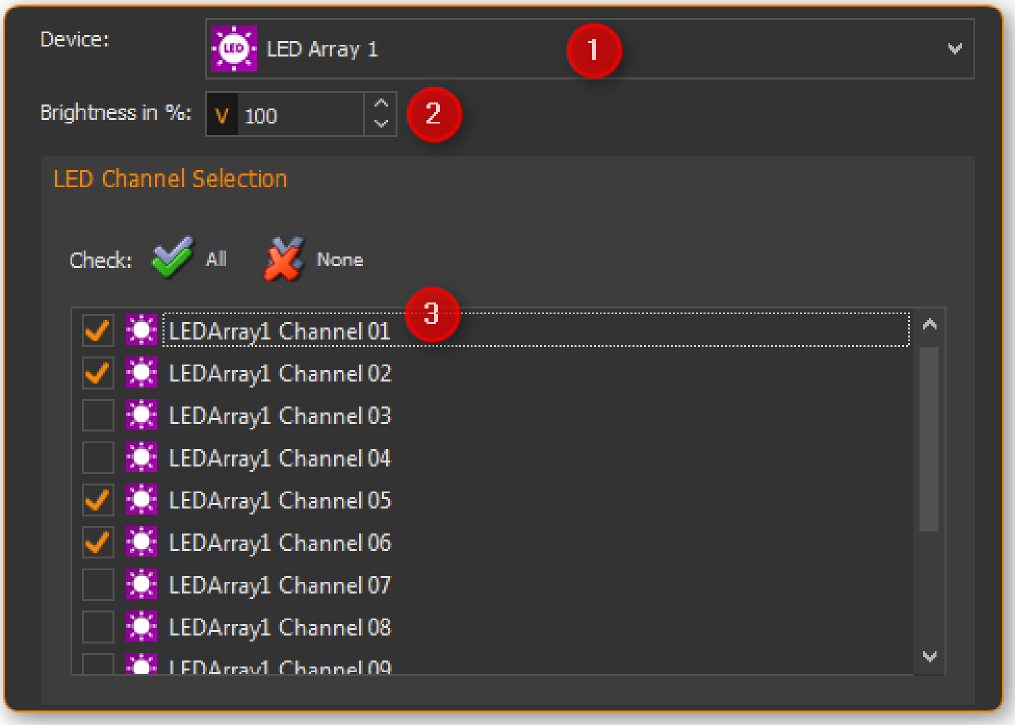


Figure 13.23: Set LED Group Brightness script configuration

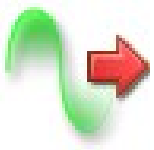
IMPORTANT

If a large number of channels have been selected, the data transfer for all channels may take some time so that not all channels will be switched exactly synchronously. If such delay is undesired, use LED groups.

HINT

This function supports script variables. Variables can be used in the Brightness field.

13.12.7 Setting the analog output function - *Set Analog Out*



All LED channels are normal analog output channels in the QmixElements software. To adjust the brightness of individual channels, you can use the *Set Analog Out* function from the *I/O Functions* category.

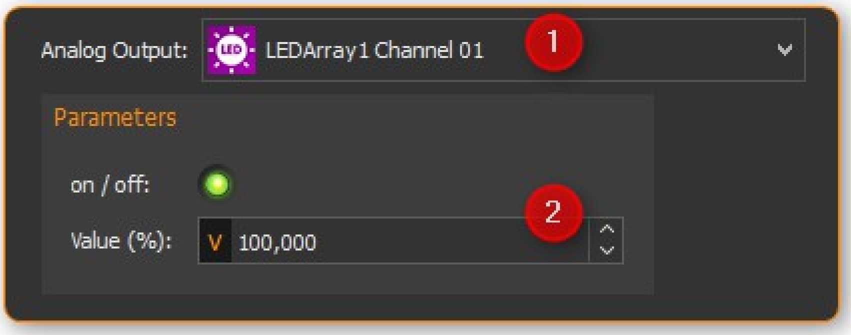
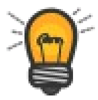


Figure 13.24: Set Analog Out script configuration

Select the relevant LED channel from the configuration menu (*Analog Output*) 1 and enter the brightness value 2 (0 – 100%) in the *Value* field.



HINT

This function supports the use of variables, i.e. you can enter in the Value field 2 *the name of the variable instead of a value which contains the brightness value at the run time of the script (see figure below).*

14 Image Analysis Plugin

14.1 Introduction

This plug-in allows you to view, edit, and save images using the most common image file formats (PNG, JPG). In addition you can use it to:

- zoom in and out of images,
- easily navigate within an image,
- measure the dimensions of details of an image.

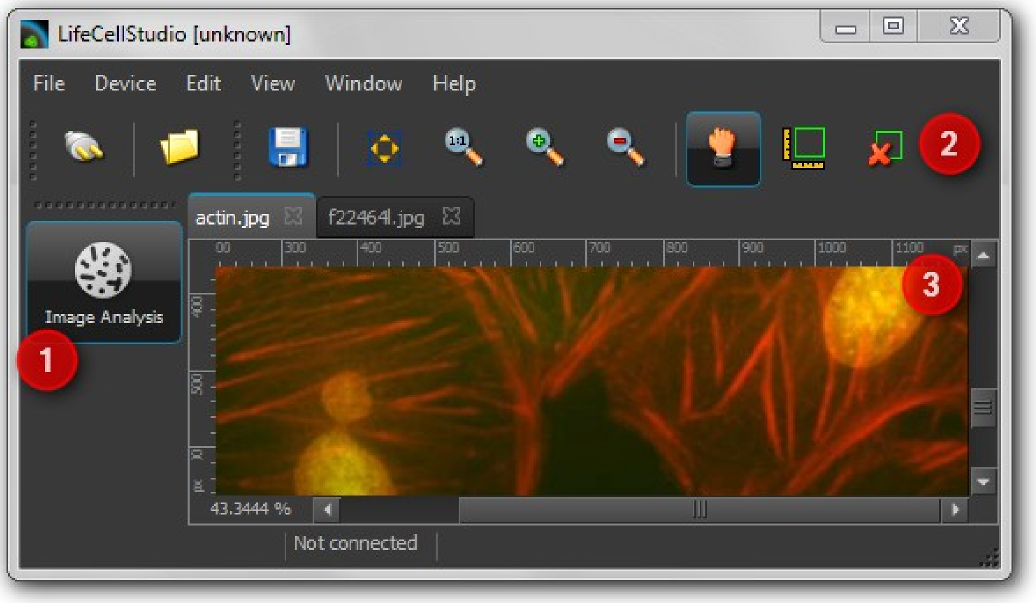


Figure 14.1: Image Analysis work space.

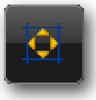
The work space of the *Image Analysis* plug-in can be activated using the button *Image Analysis* in the side bar 1. If no picture has been loaded or taken yet, this work space will be empty.

The user interface of the plug-in consists of two main elements: the toolbar 2 and the image viewer 3.

14.2 Toolbar

Open image file.

Save current image.



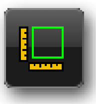
Adjust display size of image to completely fill image viewer area.

Resize image to match original pixel size.

Zoom in to image.

Zoom out of image.

Pan tool to move the currently displayed image section.



Draw rectangular measuring box.

Remove all measurement overlays.

HINT

You can also open image files easily via drag & drop. Simply drag an image file from your file system over the image analysis workbench and drop it there.

14.3 Image Viewer

14.3.1 Overview

The image viewer contains the following elements:

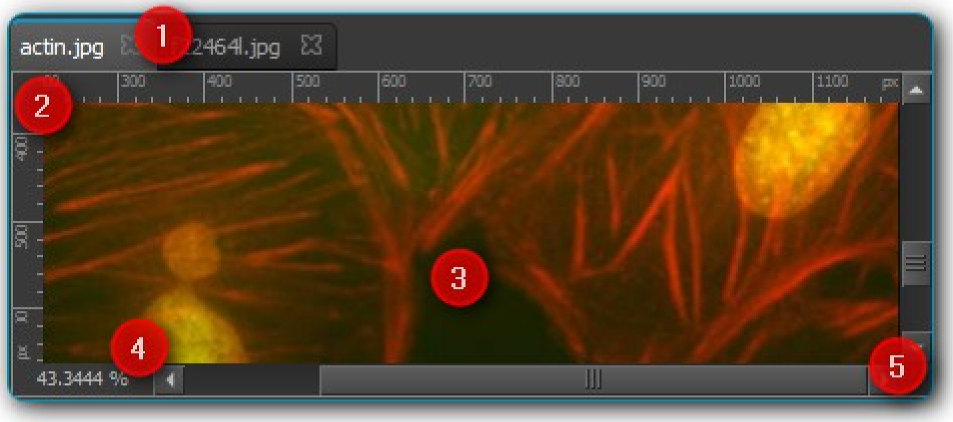


Figure 14.2: Image viewer.

6 Tabbed list of available images.

7 Ruler with customizable dimensions.

8 Display area.

9 Zoom level status bar.

10 Scroll bars.

A new tab displaying the file name will be added for each opened image, e.g., when you open an image file or take a screen shot using the camera. The active tab is highlighted.

To close a tab, left-click on the cross icon that is displayed on the right hand side of every tab (see figure below).

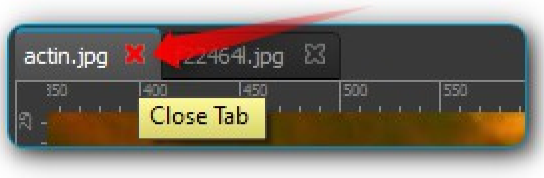


Figure 14.3: Closing a tab.

To easily access the most frequently used functions, use the mouse and right-click within the image viewing area. This will open a context menu with most of the same buttons that are shown in the toolbar.

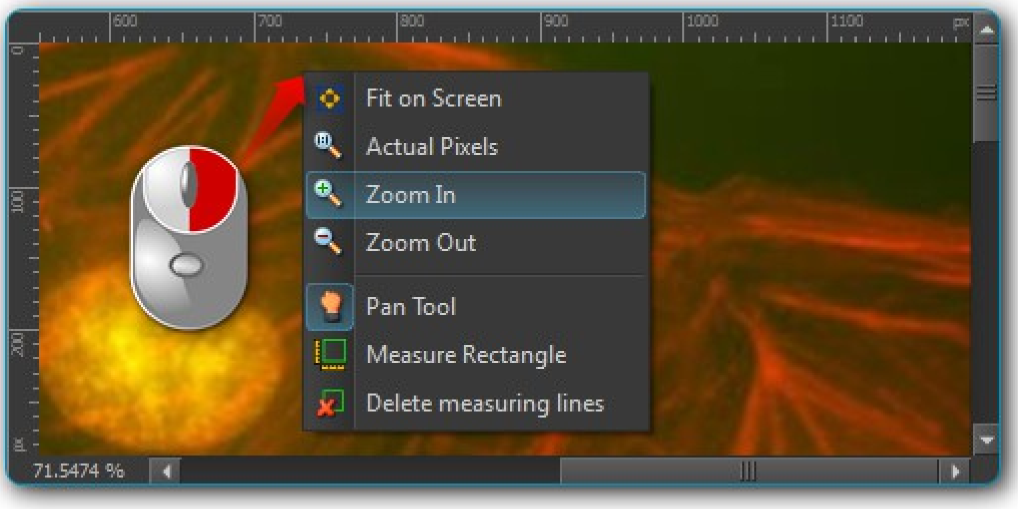


Figure 14.4: Context menu of the image viewer.

14.3.2 Select Scale Unit

You can select between different scale units for the ruler to measure objects displayed in the image viewer window. To change the scale unit right-click anywhere within the ruler. This will open a context menu with the list of available scale units (see figure below).

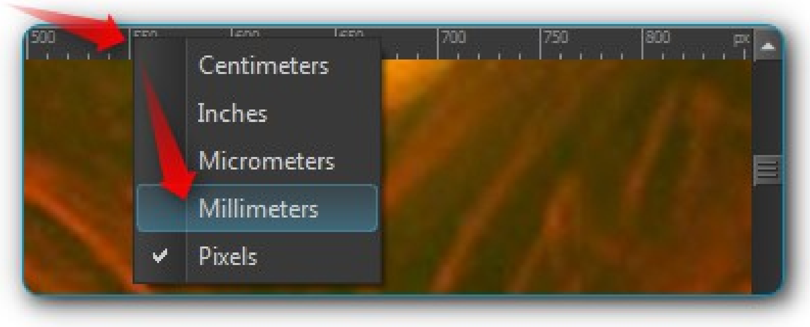
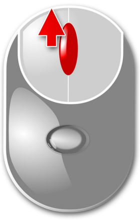


Figure 14.5: Scale unit selection.

14.3.3 Select Zoom Level

Use the *Zoom In* or *Zoom Out* buttons in the toolbar to adjust the zoom level. Alternatively, simply place the cursor into the image viewer area and use the mouse wheel to re-scale the panoramic image.



Increase zoom level.

Decrease zoom level.

14.4 Image Navigation

14.4.1 Navigation via Scrollbars

If the image is larger than the viewing area (e.g., due to a large zoom level), scrollbars will be displayed below and to the right of the image viewer. Use these scrollbars to adjust the displayed section of the image.

14.4.2 Navigation via the Pan Tool

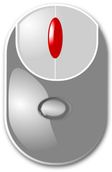
Activate the Pan Tool in the tool bar to move the displayed image section manually.

Click-and-drag into the image viewer to move the image in the viewing area.

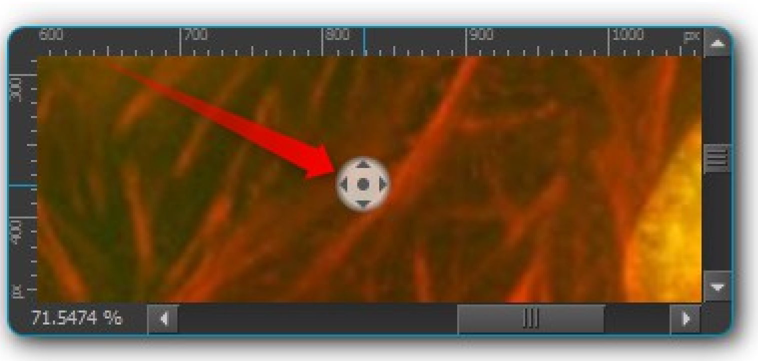
14.4.3 Navigation via Mouse

You may also use the mouse to interact with the panoramic view of an image:

(5) Middle-click into the image.

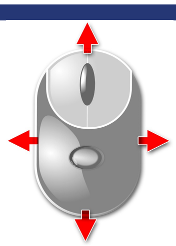


(6) This will activate a navigation cross, which indicates that mouse panning is activated.



(7) By moving the cursor away from the navigation cross, the image will move into the same relative direction. The speed of the movement depends on the distance between the cross and the cursor – the greater the distance, the faster the adjustment.

(8) Mouse panning can be deactivated by another middle-click or a left-click into the image.



14.4.4 Navigation via the Space Navigator

When the *Space Navigator* is installed, loading the relevant plug-in allows for yet another way for image manipulation.

The *Space Navigator* is an intuitive tool to move an image within the viewing area in both X- and Y-axes concomitantly.

To be able to use the *Space Navigator,* it must be installed at a free USB port of your computer. The required drivers have already been installed together with the main Qmix application.

Place the *Space Navigator* so that its cable points away from and the *3D connexion* label towards you.

Figure 14.6: Positioning the Space Navigator.

To adjust the digital zoom of the currently active image, press down or pull up the navigation knop of the *Space Navigator* (figure below).

1. To adjust the section of the image displayed in the image viewing area, pivot the navigation knop of the *Space Navigator* into the desired direction.

Figure 14.7: Zoom level adjustment with the Space Navigator.

The speed with which the adjustment is carried out can be regulated by adjusting the degree of pivoting – the stronger the navigation knop is pushed or pulled, the faster the image will move.

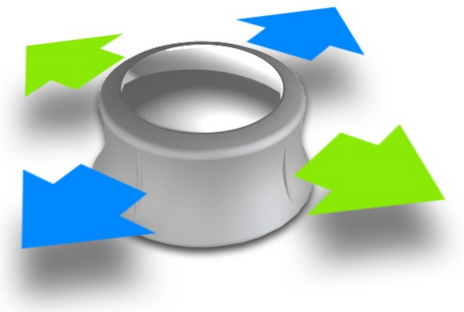
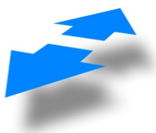


Figure 14.8: Moving an image using the Space Navigator.

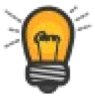
 Move image section left & right.

 Move image section up & down.

 Zoom in/out.

14.5 Measurement of Objects

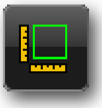
14.5.1 Draw a Measuring Box



TIP

Before drawing a measuring box, select the desired scale unit.

You can draw and adjust a measuring box using the following steps:

(1)  Click the toolbar button *Measure Rectangle* to activate the measuring box tool.

(2) Left-click into the image viewing area to define the upper left-hand corner of the measuring box 1.

(3) While holding the left mouse button, drag the cursor to size the box as required 2.

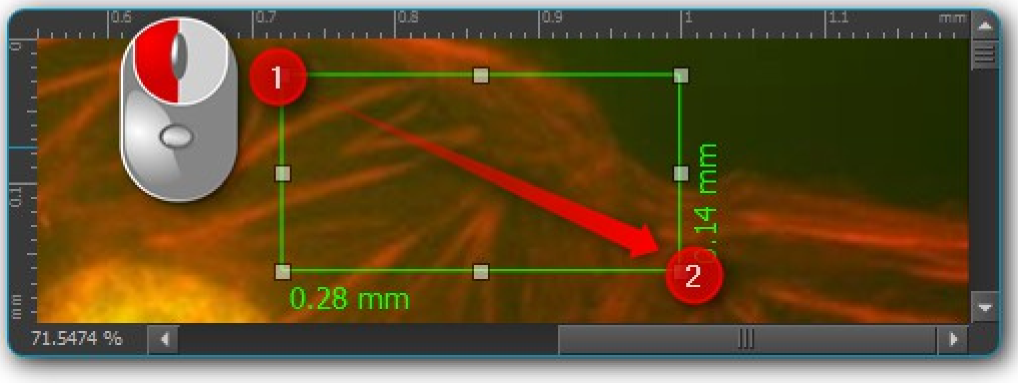


Figure 14.9: Drawing a measuring box.

14.5.2 Moving the Measuring Box

To move an entire box to a new location within the viewing area, first place the cursor within the box so that it changes into a crossed arrow 1. Left-klick to drag the measuring box to its new position 2.

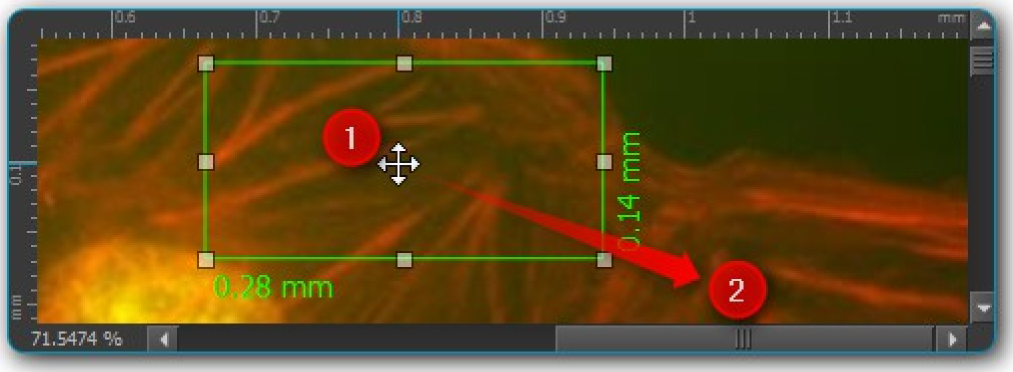


Figure 14.10: Moving a measuring box.

14.5.3 Adjusting the Size of a Measuring Box

To change the size of a measuring box, move the cursor to one of the check marks at the corners or the sides of the box (figure below). The cursor will change to a double-sided arrow 1, which indicates that you can now resizing the box by dragging the respective mark to a new position.

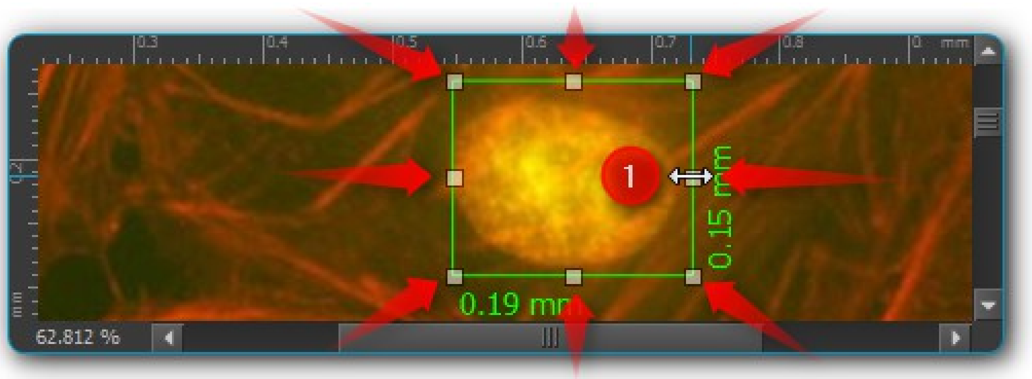


Figure 14.11: Adjusting the size of a measuring box.

14.5.4 Deleting a Measuring Box

You can delete individual measuring boxes. First click into the respective box to activate it – the activated box can be identified by the presence of the check marks. Press the Delete button of your keyboard to remove the box.

Delete all measuring boxes within an image by clicking the button *Delete measuring lines* in the toolbar or via the context menu.

15 Camera Plugin

15.1 Introduction

The camera plugin allows the user to integrate a broad range of cameras into the software. The plug-in supports, *e.g.*, Firewire 1394 cameras (DCAM standard), USB cameras, as well as all standard Windows DirectShow cameras.

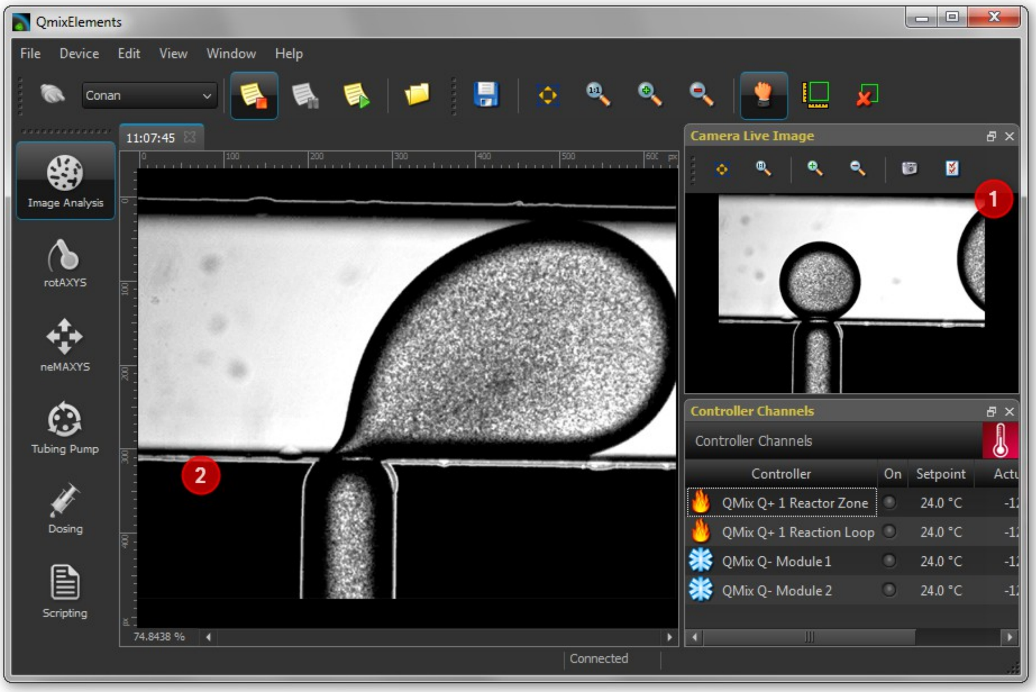


Figure 15.1: Camera live image and image viewer.

The figure above shows the typical work environment of the plug-in, including the *Camera Live Image* in a detachable tool window 1 as well as the *Image Viewer* 2 displaying captured images.

15.2 Camera Live Image

15.2.1 Overview

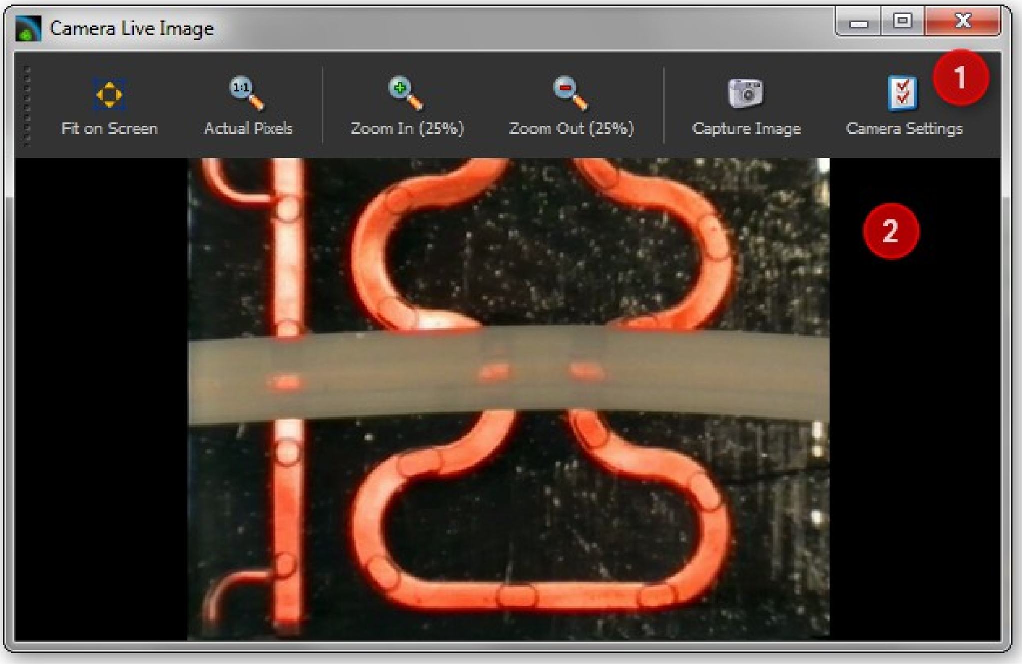


Figure 15.2: Camera Live Image window.

The camera's live image is displayed in a tool window that the user may move freely on the screen desktop via drag-and-drop of the window's title frame. Thus the user can place the live image on a second monitor and may observe the live image whilst maintaining complete oversight and control over the software settings on the first monitor.

1 Toolbar

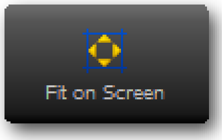
2 Camera live image

15.2.2 Toolbar

A variety of tools are available via buttons in the top part of the live image window so that you can adjust the displayed image or camera settings (see figure below).

Figure 15.3: Toolbar of the Camera Live Image window.

These buttons have the flowing functions:



Resize image to fit window to fit the image into the available display space.

Resize image to its native pixel resolution relative to the screen.

Increase digital zoom

Decrease digital zoom

Capture the currently displayed image.

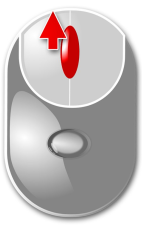
Record video

Opens dialog for configuration of camera settings

15.2.3 Adjust Zoom Level

The digital zoom may be adjusted using one of three alternatives:

- Toolbar – using the toolbar buttons will change the zoom between preset levels.
- Context menu – a right-click into the live image opens a context menu that contains the relevant items to adjust the zoom level.
- Mouse wheel – changes the zoom level smoothly



Increase magnification level (zoom in).

Decrease magnification level (zoom out).

15.2.4 Manual Capture

Click on the toolbar button *Capture Image* to capture the currently displayed camera live image*:*

The captured image will be saved to the software's image viewer for further adjustments.

15.2.5 Manual Video Recording

If you want to record the current camera live image as a video, simply press the *Record Video* button. The video recording will remain active until you stop it by clicking the *Record Video* button again.

Video recording inactive

Video recording active

In the *Event Log* of the application you will get status information about the recording state and about the file name of the saved video file.

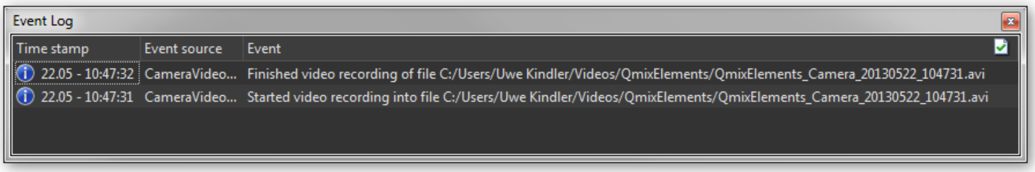


Figure 15.4: Status information video recording

The file name of each video file is extended with a time stamp that indicates the start of the recording. E.g. the recording of the video file *QmixElements_Camera_20130522_104731.avi* started 2013-05-22 at 10:47 o'clock.

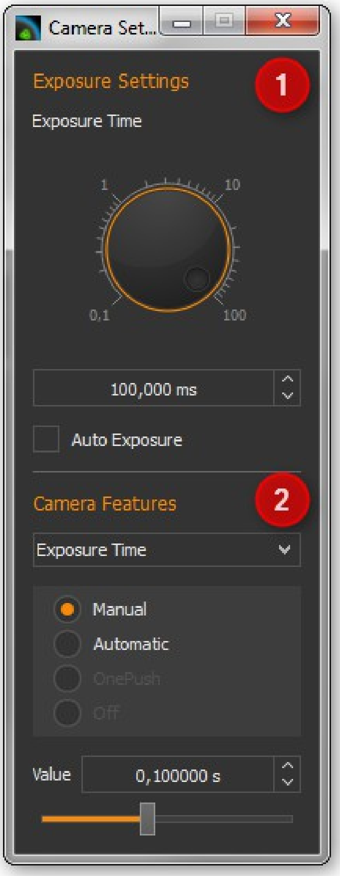
IMPORTANT

The XVID codec is used for video recording. If you want to play back the recorded videos with your media player, it needs to support this codec or the XVID codec should be installed on you computer system.

15.3 Camera Settings

15.3.1 Introduction

The quality of the images depends on the lighting conditions and the chosen camera settings. The following will provide an overview of the most important parameters that may be changed via the software.



The window pane to adjust the camera settings may be opened via the *Camera Settings* toolbar button:

The window has to main areas:

1 *Exposure Settings* – to set the exposure settings.

2 *Camera Features* – to adjust general camera settings.

IMPORTANT

Depending on the used camera type (Direct Show, USB, or Firewire) or model, the actual camera settings dialog may look differently.

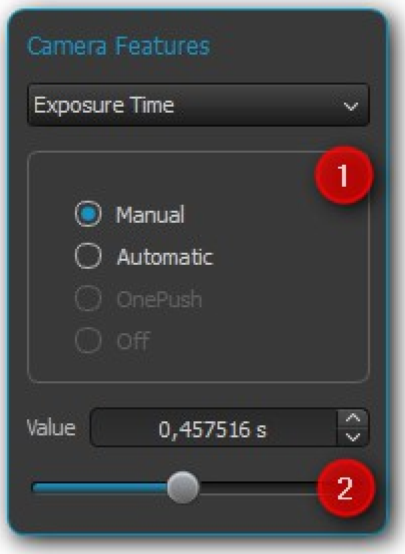
> **IMPORTANT**
> *Some cameras may not be supported by the Camera Settings dialog and will show their own configuration panel instead.*

15.3.2 Setting Exposure Parameters

With *Exposure Settings* you may set the exposure time of the image sensor inside of the camera. It may operate either in manual mode or in automatic mode. The *Auto Exposure* mode will adjust the exposure time according to the brightness of the image.

15.3.3 Setting Camera Parameters

All settings of the camera may be adjusted via the *Camera Features* controls. The feature selection and their respective value ranges may vary according to the type of camera used.



If supported by your camera, you may toggle between manual and automatic modes 1. In manual mode the values may be changed either numerically via the input box or adjusted via the sliding controller 2.

IMPORTANT

Depending on the type of camera used, the available parameters and their value ranges may differ.

15.4 Camera Script Functions

15.4.1 Introduction

The camera plug-in contains script functions for capturing still images and for recording videos:

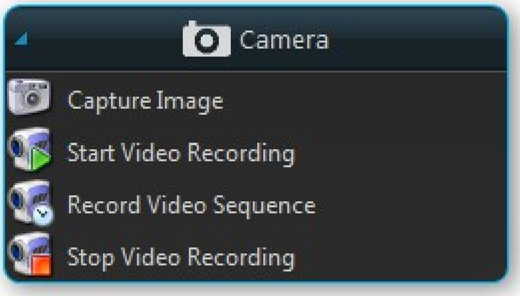


Figure 15.5: Camera script functions.

15.4.2 Still Image Capture

The function *Capture Image* is to take individual shots of the current camera view. The *Image File* section is to set the file path and file name of the captures image files 1.

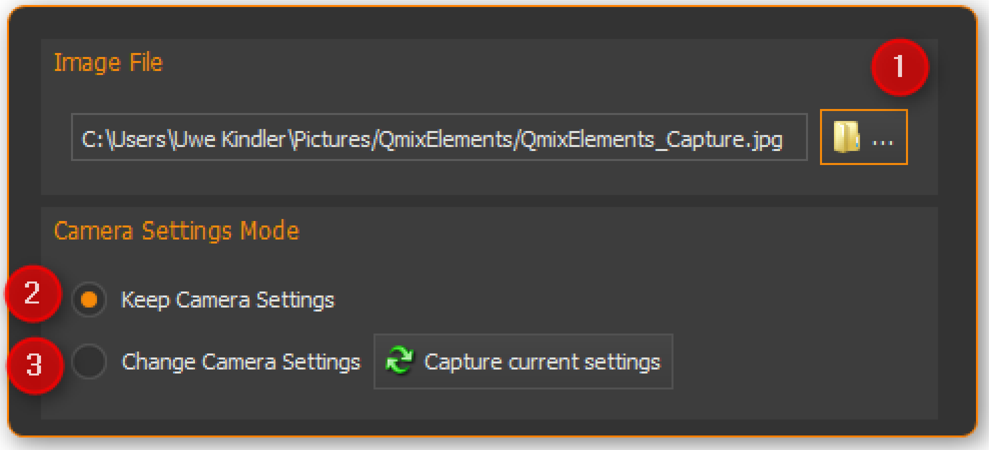


Figure 15.6: Capture Image configuration.

Subsequently you may set whether the camera settings should be used unaltered

❷ (*Keep Camera Settings*) or whether to reset them to pre-defined and saved values ❸ (*Change Camera Settings*) before capturing an image.

The current camera settings may be saved as function parameters by pressing the button *Capture current settings* to be able to reset the camera to those parameters just prior to capturing another image at a later point. Thus you can take images of different objects with completely different camera settings.

HINT

You may select from different image formats by setting the appropriate file extension (.png or .jpg).

Every image file name will be appended with a time stamp at the time of capture. This is, when you have defined the file name as *Photo.jpg* the resulting file will be saved as *Photo_20120921_154502.jpg* according to the syntax *name_YYYYMMDD_hhmmss.jpg.*

HINT

You may capture images at set intervals to subsequently compile them to a stop-motion movie using the Video Builder plug-in.

15.4.3 Start Video Recording

With this function you can start recording of the camera live image into a video file. The record will remain active until it is stopped with the [Stop Video Recording](#) function. The video recording will stop automatically when the script execution ends.

In the configuration area you can configure all the video recording parameters. Enter a unique name for this recording session in the input field *Recording Session Name* 1. This name is required to stop the recording later with the [Stop Video Recording](#) function.

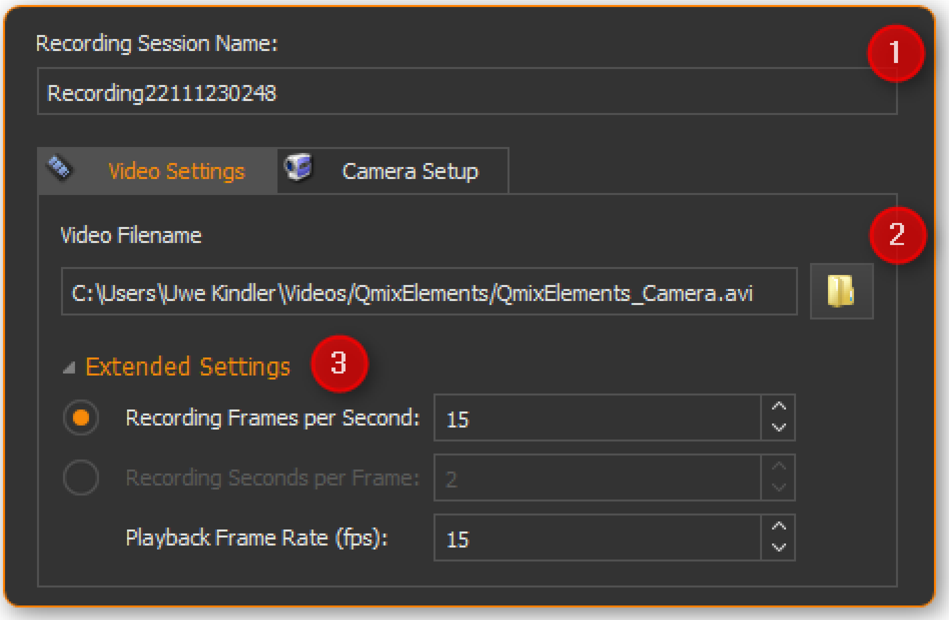


Figure 15.7: Configuration of Start Video Recording function

15.4.3.1 Configuring video settings

In the *Video Settings* tab, you configure all video settings. Click on the button with the folder icon 2 to select a file name and a storage location for the video file in the *Video Filename* field.

IMPORTANT

Every video file name will be appended with a time stamp that identifies the start of its recording. This is, when you have defined the file name as Video.avi the resulting file will be saved as Video_20130522_154502.avi.

By clicking *Extended Settings* 3 you can display additional settings. In the field *Recording Frames per Second* set the number of images to be recorded per second. To get a real-time video playback later, you should use the current camera frame rate.

HINT

If you create a new recording function, the current camera frame rate is automatically entered into the ***Recording Frames per Second*** *field.*

If you want to monitor slower processes, you can also choose a smaller frame rate for recording. If you check the check box *Recording Seconds per frame*, you can choose very slow frame rates (several seconds or minutes per image).

This mode is suited for creating time-lapse recordings of very slow processes. So you can for example, take a picture every 30 minutes for several days and then play back these captured images in a video with a frame rate of 24 frames per second.

In the input field *Playback Frame Rate (fps)*, set the frame rate at which the film is played back in media player later. In the fields of video production or motion picture a frame rate of 24 images per second is the standard frame rate. If you want to play back the movie in real time, that means with its real recording speed, you should choose a frame rate here, that matches the recording frame rate.

15.4.3.2 Configuring camera settings

Select the *Camera Setup* 1 tab to display the controls for configuring the camera settings:

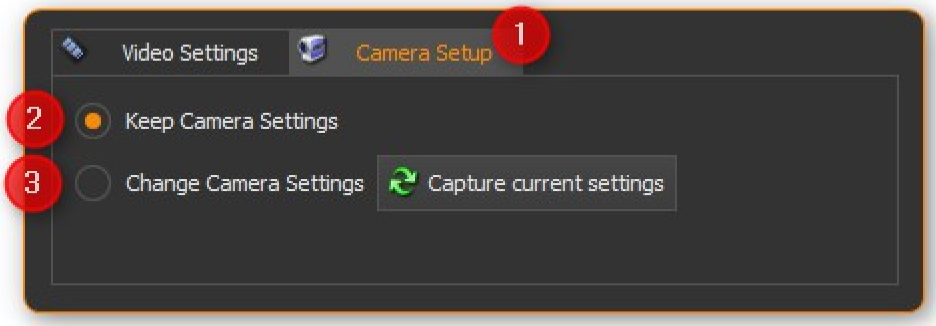


Figure 15.8: Configuring camera settings of Start Video Recording function

Here you may set whether the camera settings should be used unaltered 2 (*Keep Camera Settings*) or whether to reset them to pre-defined and saved values 3 (*Change Camera Settings*) before capturing an image.

The current camera settings may be saved as function parameters by pressing the button *Capture current settings* to be able to reset the camera to those parameters just prior to capturing another image at a later point. Thus you can take images of different objects with completely different camera settings.

15.4.4 Record Video Sequence

Use this function to record a video sequence with a fixed duration. You can configure the desired recording duration and the recording stops automatically after the expiry of the recording time.

All video settings and camera settings for this script function are identical to the settings of the *Start Video Recording* function. Additionally you may set the recording duration for this function.

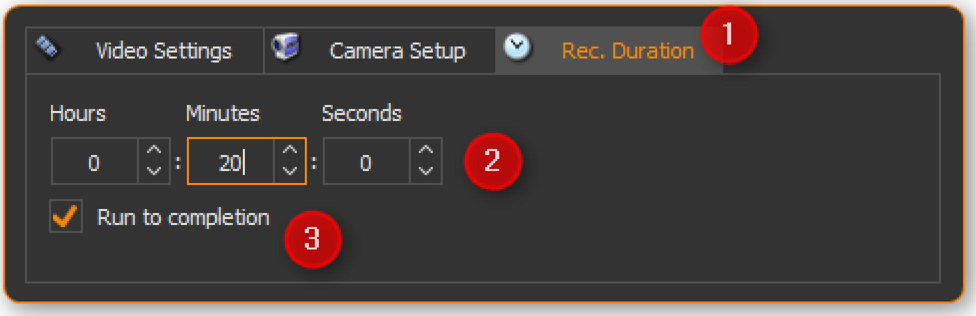


Figure 15.9: Configuration of Record Video Sequence function

Select the tab *Rec. Duration* 1. Now you can set the recording duration accurate to the second 2. Use the selection field *Run to completion* 3 to determine, when the recording function is finished and when the next script function will be executed:

- *Check box active* – the next function is executed when the recording is completed
- *Check box inactive* – the recording is started and then script execution immediately continues with the next script function

HINT

The recording of a video sequence can be stopped at any time by calling the Stop Video Recording function.

15.4.5 Stop Video Recording

A running video recording that was started with the functions *Start Video Recording* or *Record Video Sequence* can be terminated at any time with this function. Simply enter the name of the recording session you want to quit into the input field *Recording Session*.

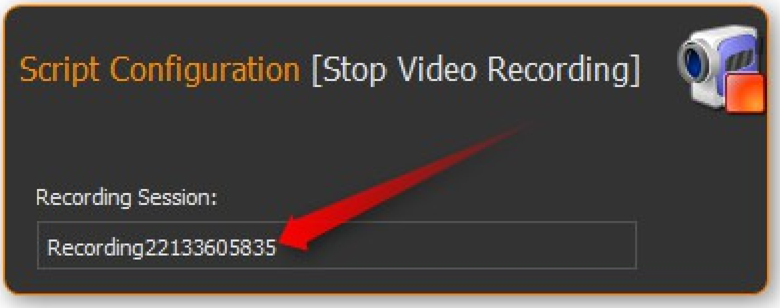


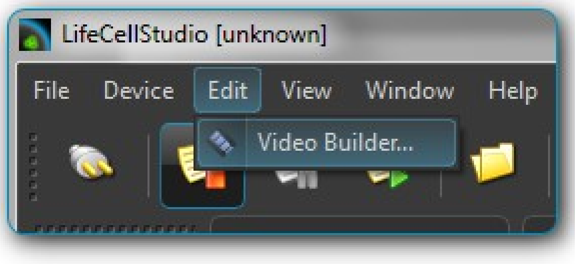
Figure 15.10: Configuration of Stop Video Recording function

16 Video Builder Plugin

16.1 Introduction

The *Video Builder* plug-in allows you to quickly compose videos from individual images taken by timing or scripting via the camera or the microscope plug-ins. This is useful, for instance, for time laps recordings of slow microfluidic processes or for long-exposure microscope pictures.

To open the *Video Builder* plug-in, select the relevant item from the main menu via *Edit* → *Video Builder*.



The *Video Builder* dialog contains the following elements (see figure below):

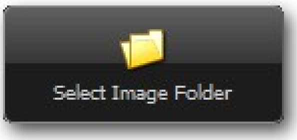
- 3 Tool bar,
- 4 List of image folder(s),
- 5 Video parameter settings,
- 6 Thumbnail of the currently processed picture,
- 7 Progress bar of the processing step.

Figure 16.1: Video Builder dialog box.

16.2 Video Assembly

The following description shows you how to assemble a video from individual images.

16.2.1 Step 1 – Select Image Files



To select the image files that you want to process, first locate the relevant folder and open it via *Select Image Folder* in the tool bar.

This opens a dialog box (see figure below) that allows you to navigate to the folder containing your image files 1. Open the folder by clicking on *Choose* 2.

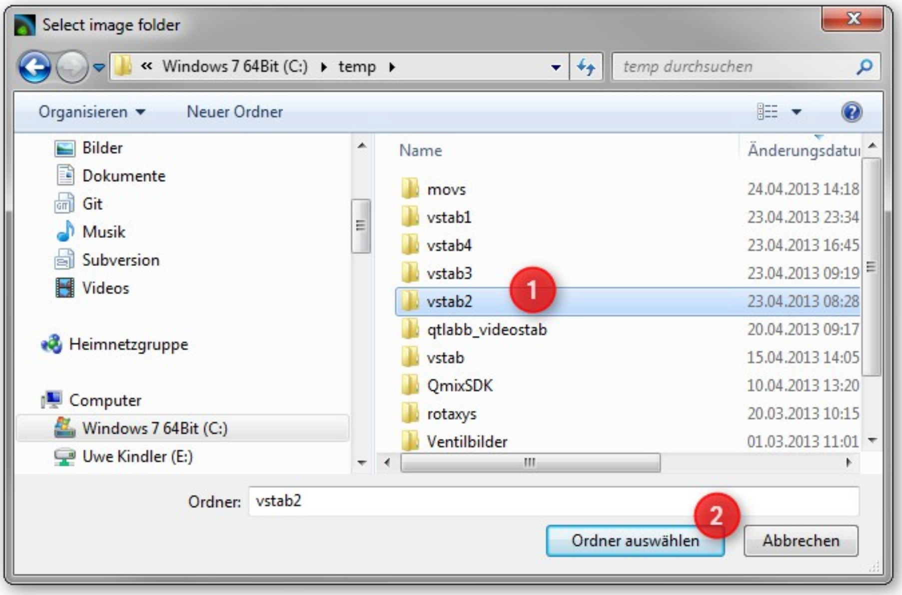
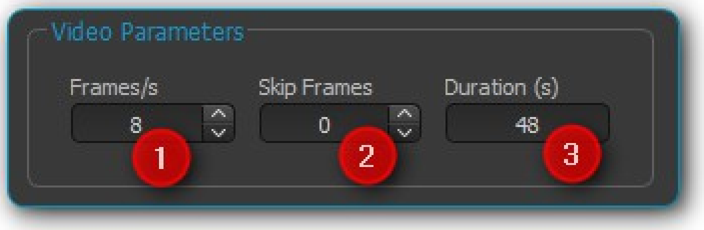


Figure 16.2: Select directory containing individual image files.

16.2.2 Step 2 – Video Assembly Parameters

The box *Video Parameters* is to configure the parameter settings for the image assembly process.



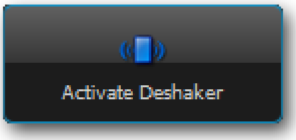
You can set the following parameters:

1 *Frames/s* – configures the frame rate, i.e., how many images are to be shown per second.

2 *Skip Frames* – set the number of images that are excluded from the video. If this value is set to zero, all image files will be used; if set to one, for instance, every other image will be skipped and thus only every second image will be used to assemble the video.

Based on the number of available images and the two parameters just set, the third box (*Duration (s)* 3) will show the length of the final video.

In addition, the software offers the possibility to de-shake (stabilize) the video sequence:



HINT

Video sequences (e.g. of images captured from microscope camera) can be stabilized using the de-shaker function.

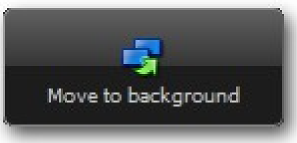
IMPORTANT

The de-shaking step will increase the time for building a video.

16.2.3 Step 2 – Start Video Assembly

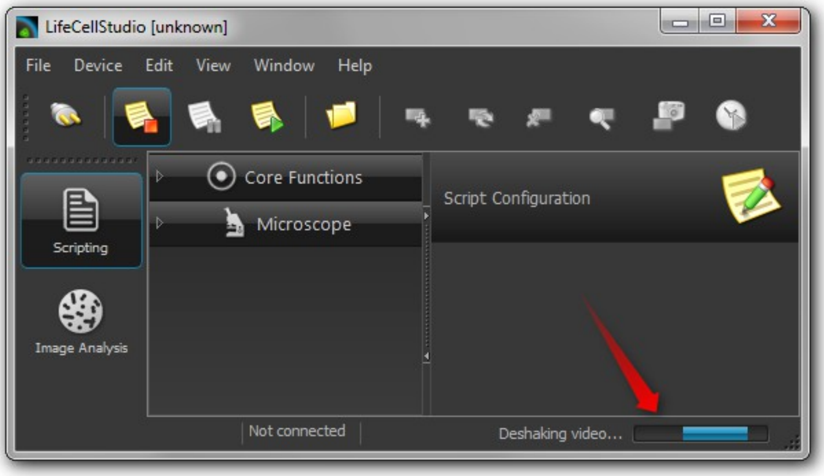
Start the assembly process by clicking on the *Build Video* button.

This will open a dialog box that is to define the target directory and the name of the video file. Processing will start immediately thereafter.



The assembly process may take several minutes, in particular when a large number of images is to be processed. Processing may be continued as a background task by activating the *Move to background* feature of the Qmix Elements plug-in. This will close the *Video Builder* dialog and allows you to continue to work with the Qmix Elements application.

If a process is being carried in the background, this will be indicated in the status bar of the main application window (see figure below).



You will be notified via the application's *Event Log* as soon as the video assembly process has been completed (see figure below). Now you may start the process with a new set of pictures or with different settings.

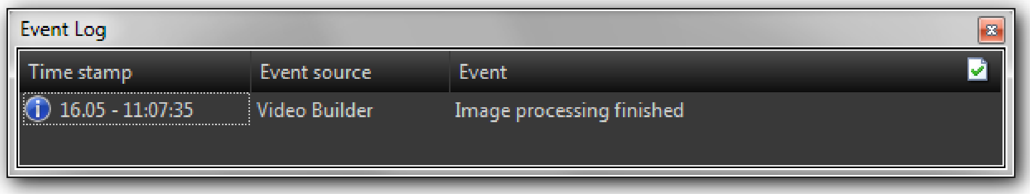


Figure 16.3: Event Log message about process completion.

17 Spectroscopy Plugin

17.1 Installation

The spectroscopy plugin is not part of the standard QmixElements installation package, but must be installed as an add-on. The versions of the spectroscopy plugin have to match the installed QmixElements Software. For example, if you want to install the spectroscopy plugin with version number 20150520, you need to have the QmixElements software, version number 20150520, installed on your machine.

IMPORTANT

The version number of the QmixElements software and the Spectroscopy plugins must match.

Please close all other programs before starting the installation.

IMPORTANT

Install the spectroscopy plugin + the device driver before connecting your spectrometer to the PC via USB for the first time.

For installation launch *QmixElements_Spectroscopy_Setup.exe*. The installation assistant will guide you through the installation of the software and hardware drivers.

IMPORTANT

When using Windows you have to be logged in with administrator privileges to be able to install the hardware drivers.

The installation assistant will now guide you through the installation of the software.

17.2 Introduction

By clicking the *Spectroscopy* 4 button in the sidebar you will get to the *spectroscopy plugin* (see figures below).

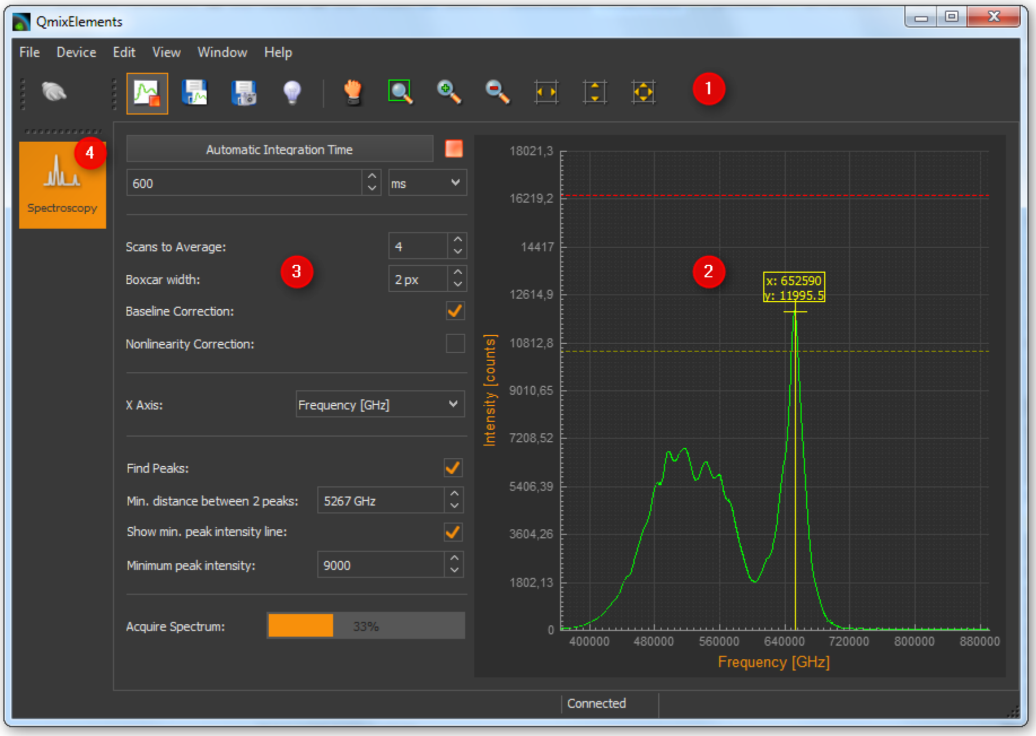


Figure 17.1: Overview of spectroscopy workbench

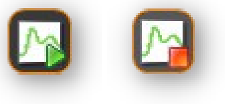
1 Toolbar

2 Spectrum live display

1 Parameter control elements

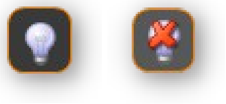
2 *Spectroscopy* button

17.3 Toolbar

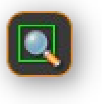

Start / Stop showing spectrums in live-display

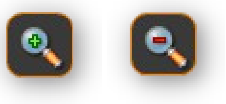

Save spectral data as txt-file

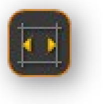

Save spectral data as png-file

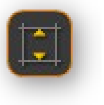

Acquire / remove a background spectrum

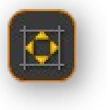

Panorama tool for shifting the spectrum live display


Zoom selection tool (enlargement)


Enlarge / minimize spectrum live display


Rescale x-axis to fit it to visible screen space


Rescale y-axis to fit it to visible screen space


Rescale both axes to fit them to the screen

17.4 Spectrum Live Display

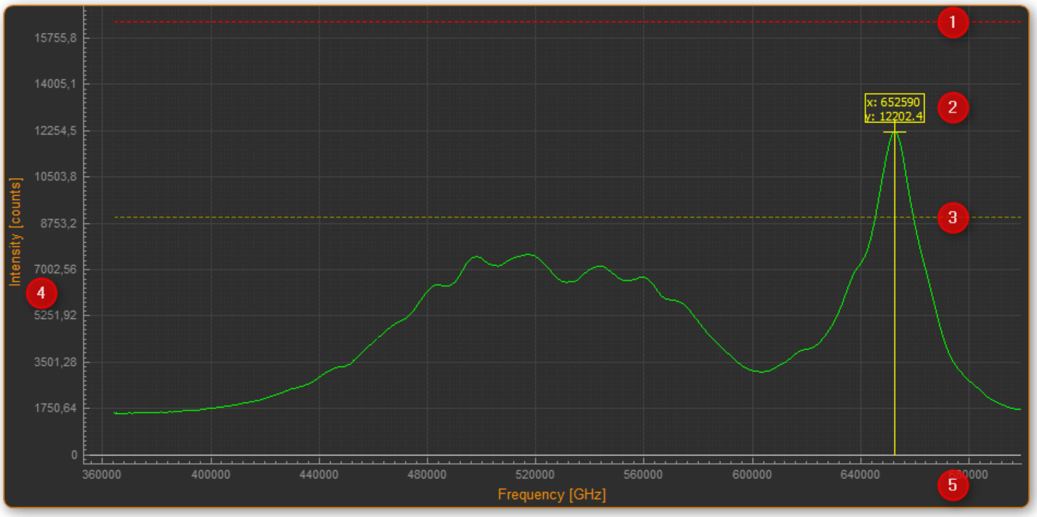


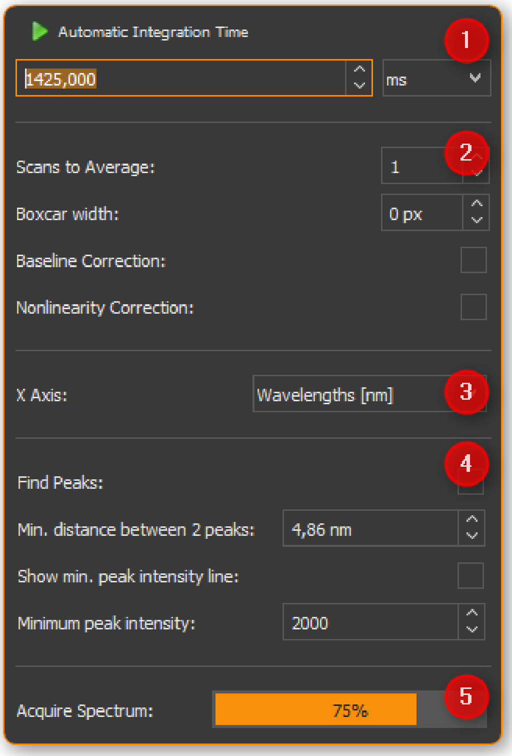
Figure 17.2: Spectrum Live Display

The saturation line 1 marks the saturation value of the CCD-sensor, meaning the maximum photon detection capacity has been reached. Any values above the saturation point are no longer displayed and the spectrum is cut off. The peak marker 2 shows the x and y coordinates of the respective spectrum peak. The minimum peak intensity line is displayed in yellow 3. Below this line spectrum peaks are no longer marked. Currently, the x-axis 5 uses GHz as a unit. The unit can be changed using the x-axis control element. The y-axis 4 uses intensity (counts) as a unit.

17.5 Parameter Control Elements

17.5.1 Overview

The control elements control the acquisition and pre-processing of spectrums. Here you can also set the parameters for peak recognition.



1 Integration time setting

2 Spectrum pre-processing

3 Unit selection for x-axis

4 Find peaks

5 Acquisition progress

Using the spectrum live display you can immediately see and evaluate the effect of parameter changes.

17.5.2 Integration Time Setting

The integration time is equivalent to the exposure time of the spectrometer CCD.

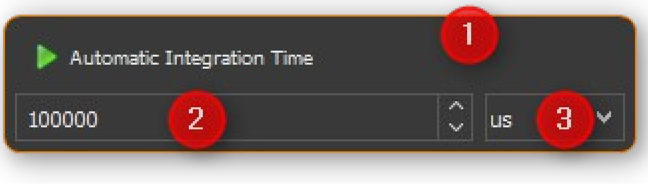


Figure 17.3: Integration time settings

For automatic setting of the integration time please click the *Automatic Integration Time* 1 button. The integration time increases in steps until the highest spectrum peak has reached approximately 85% of the saturation value. While the calculation is in progress the green play symbol is replaced by a red stop symbol. You can click on this symbol to abort the calculation process. An individual integration time can be assigned in Spinbox 2, the desired time unit can be set in the selection field 3. As the integration time grows the spectrum's intensity values will increase.

TIP

The greater the integration time the longer the spectrum acquisition takes to complete. Set a smaller integration time if the automatic integration time calculation is taking too long.

If the product of integration time and *scans to average* is 10 seconds or larger, a window will appear advising the future length of integration time. You can either adopt the current parameter settings or maintain the original values.

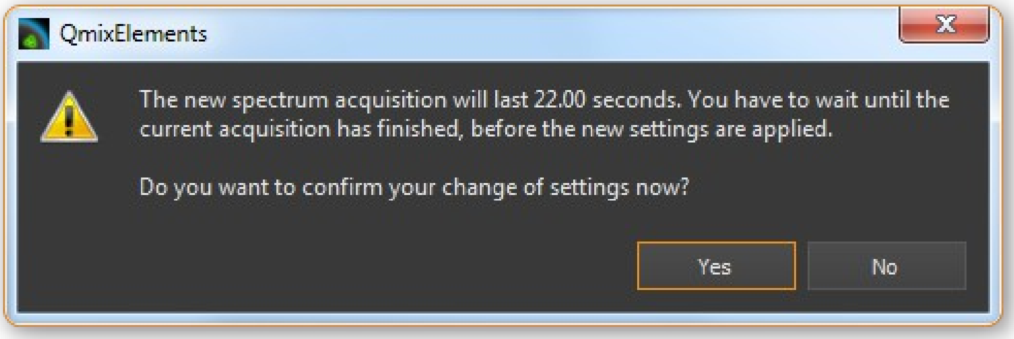


Figure 17.4: Pop-up window in case of excessive integration time.

IMPORTANT

Any actions triggered are executed only after the previous spectrum acquisition is complete. In case of long integration times this could cause the impression of the software no longer responding.

17.5.3 Spectrum Pre-processing

The pre-processing of live spectrums can contribute to a better signal-to-noise-ratio (SNR). This reduces interference while improving spectrum quality. Spectral smoothing makes it easier to recognize significant peaks. The removal of non-linearities and the baseline correction of spectrums will improve their quality.

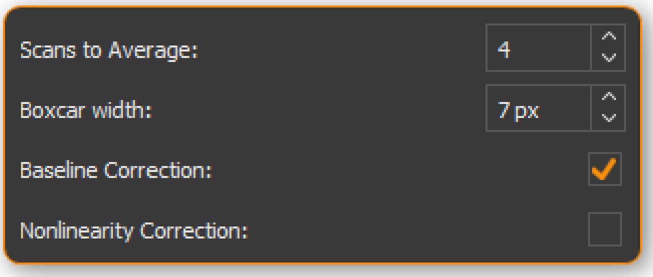


Figure 17.5: Spectrum pre-processing

Scans To Average:

- Number of discrete spectrum captures before an average spectrum is displayed (SNR increases with increasing captures)

Boxcar Width:

- Width of the rectangular function used to smoothen the spectrum
- Averaging with neighboring CCD sensor elements
  (e.g. boxcar width = 3: averaging each data value with its 3 neighboring values to the left and right)
- As the boxcar width increases, so does the SNR → the spectrum smoothens out
- Excessive boxcar width: reduction of spectral resolution

Baseline Correction:

- for STS spectrometers
- Live spectrum shifted down on the plot → lowest intensity value touches baseline

Nonlinearity Correction:

- Corrects non-linearity between photon stream and output signal of the CCD sensor

17.5.4 Unit Selection for the X-Axis

You may select the following units for the x-axis:

Wavelengths [nm]:

- Wavelength of the x-axis in nanometers

Pixels [px]:

- Each pixel value corresponds to one CCD sensor element
- Starts at 0, ends at n-1 (n = maximum number of sensor elements)

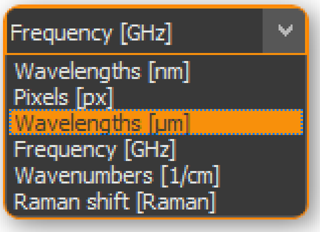


Figure 17.6: Select unit for x-axis

Wavelengths [µm]:

- Wavelength of the x-axis in micrometers

Frequency [GHz]:

- x-axis frequency in gigahertz (GHz)

Wave numbers [1/cm]:

- x-axis in inverse centimeters (1/cm)

Raman shift [Raman]:

- x-axis in 1/cm
- depending on the laser excitation wavelength

17.5.5 Finding Peaks

A spectrum's peak values function as its “fingerprint”. A substance is identified by the horizontal position of peak values. In addition, intensity values of main and secondary peaks can be used as indicators of individual concentrations in mixed substances. The following functions let you mark and select peaks.

Find Peaks:

- Marking peaks in the spectrum
- Displaying peak coordinates

Min. distance between 2 peaks:

- Minimum distance (in x-direction) between two peak markings
- The unit of minimum distance and x-axis is always the same

TIP

Since the minimum peak distance cannot be transferred to the plot with complete accuracy, please feel free to play with the distance value in order to find your optimum result.

Show min. peak intensity line:

- Dashed yellow line: vertical limit for peak marks
- Peaks above the lines get marked, peaks below the line do not (find peaks function is active)
- The vertical location of the line corresponds to the minimum peak intensity

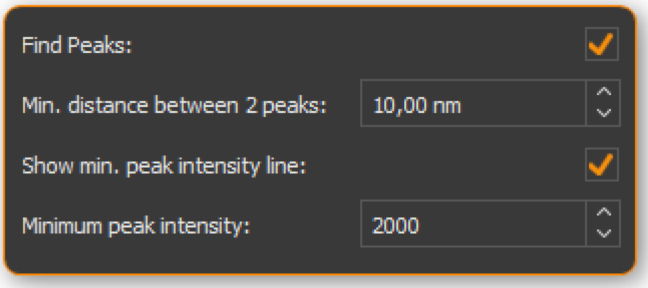


Figure 17.7: Finding peaks

Minimum peak intensity:

- Minimum intensity, up to which no peaks will get marked
- Shown in live display as active *peak intensity line*

TIP

An excessive number of peak marks can impair your computer's performance. Only a limited number of peaks will be displayed.

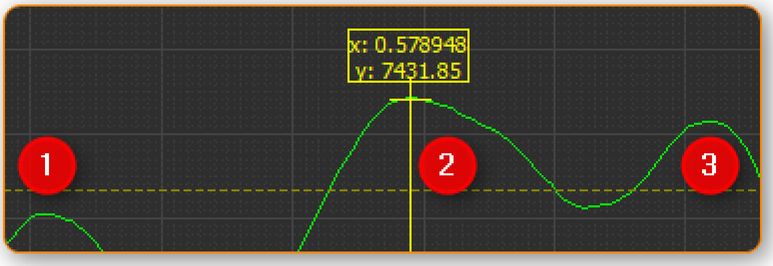


Figure 17.8: When peaks get marked

Application example: A peak will not get marked if it is located below the minimum peak intensity line 1. If the peak is located above the line 2, the mark will be displayed. The last peak does not get marked 3, since the distance between it and its predecessor is shorter than the minimum peak distance. If the minimum peak distance were exceeded, the last peak would get marked as well.

TIP

If an excessive number of peak marks affect your computer's performance, please reduce the number of peak marks. To achieve this increase the minimum distance and the minimum peak intensity or smoothen the spectrum.

17.5.6 Acquisition Progress

A progress bar is displayed below the control elements, if the integration time rises to 500 milliseconds or above.

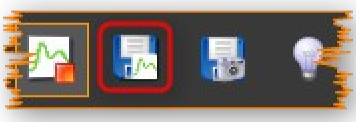
IMPORTANT

Initially, the progress bar will stay at 100%, if the "scans to average" value is larger than 1. Once the calculated average spectrum is *displayed the progress bar will restart at 0%.*

If the integration time drops below 500 milliseconds, the progress bar disappears.

17.6 Saving Spectral Data

Save your spectral data as a txt file by clicking on the disc symbol with the spectrum graph in the toolbar.



A dialog window will open, suggesting a file name.

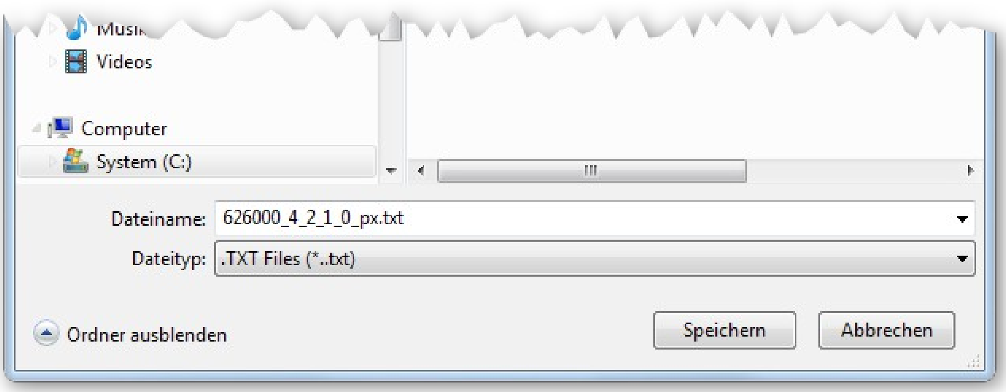


Figure 17.9: Saving spectral data

The file name consists of the parameters currently set.

`IntZeit_ScnToAvg_BoxWid_BasCor_NonCor_Einheit.txt`

Parameter	Meaning	Value in dialog window
IntZeit	Integration time in microseconds	626000
ScnToAvg	Scans To Average	4
BoxWid	Boxcar Width	2
BasCor	Baseline Correction active (1)/inactive (0)	1

NonCor	Nonlinearity Correction active (1)/inactive (0)	0
Unit	X-axis unit	px

17.7 Applying Background Spectrum

If a background spectrum is deducted from the current spectrum, stray light influences will be reduced. Begin by using the control elements to set the parameters in the way you would actually use them in your measurements. Deactivate your light or laser source and acquire a background spectrum by clicking on the light bulb symbol.

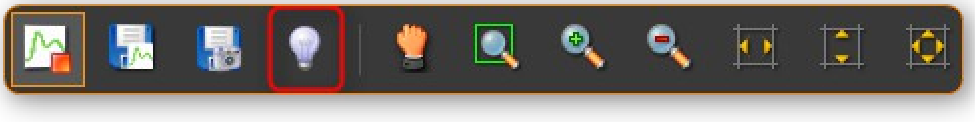


Figure 17.10: Apply background spectrum

Now the same background spectrum will be automatically deducted from each newly acquired spectrum.

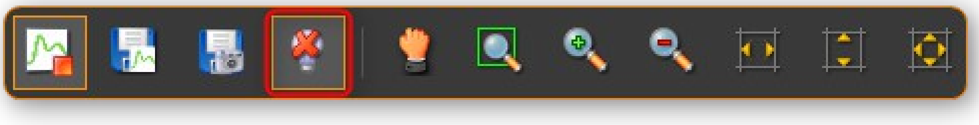


Figure 17.11: Remove background spectrum

If you don't like the result, you can click the crossed out lightbulb symbol to remove the background spectrum.

18 CSV Data Logger Plugin

18.1 Introduction

The data logger plug-in provides the user with a powerful tool to record any of the data provided by connected devices in a user-defined time interval. Data are written into a file of the CSV format (CSV: comma/character-separated values). This text file format is commonly used to save and exchange simply-structured data.

HINT

CSV files can be opened and worked with in spreadsheet applications, such as Microsoft Excel if the correct value-separating and decimal sign has been used.

18.2 Configuration Dialogue

18.2.1 Open the Configuration Dialogue

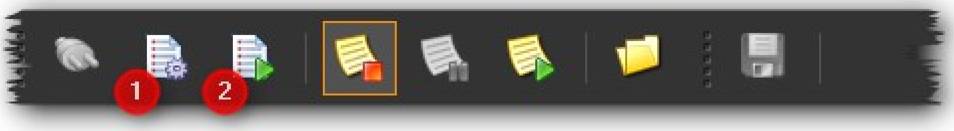


Figure 18.1: Toolbar for data logging.

When the data logging plug-in has been loaded, the toolbar will display two additional buttons for the configuration of the logging of data 1 and to start/stop the logging process 2.

18.2.2 Overview Data Logger Plug-in

Once the data logging configuration has been activated, the following configuration dialogue will be displayed:

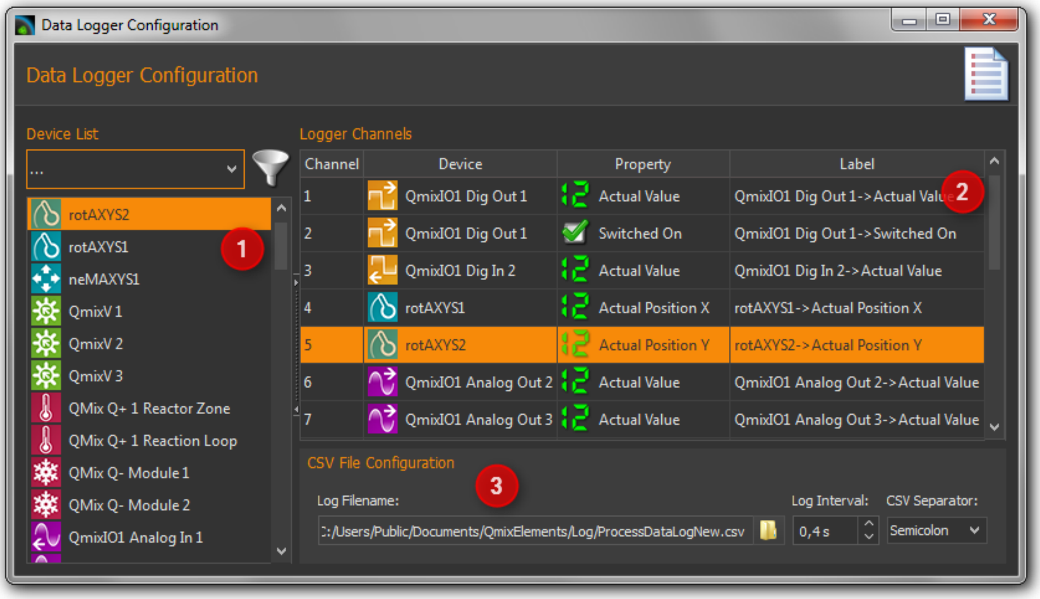
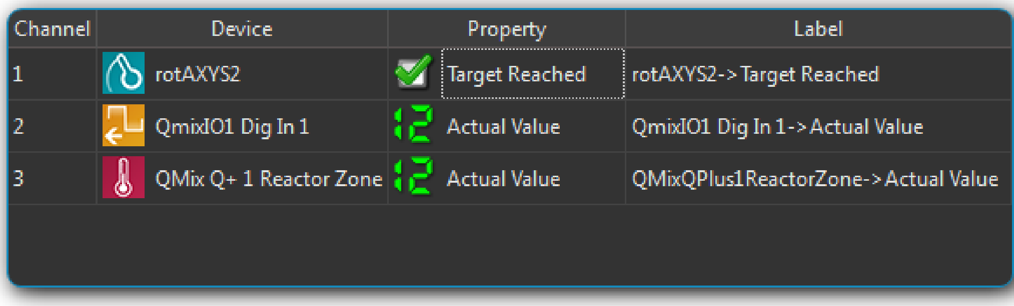


Figure 18.2: Configuration dialogue for data logging.

The configuration dialogue contains the following elements:

- 3 *Device List* – displays all devices or modules that provided recordable data. The filter selector above is to limit the list to specific device types, e.g. valves.
- 4 *Logger Channels* – lists all channels that may be recorded by the logger.
- 5 *CSV File Configuration* – allows the user to set various settings for the data logging file.

18.2.3 Overview Table Logger-Channels



Channel	Device	Property	Label
1	rotAXYS2	Target Reached	rotAXYS2->Target Reached
2	QmixIO1 Dig In 1	Actual Value	QmixIO1 Dig In 1->Actual Value
3	QMix Q+ 1 Reactor Zone	Actual Value	QMixQPlus1ReactorZone->Actual Value

Figure 18.3: Logger channel table columns

The table *Logger Channels* shows the configuration of the process data logger. Each row in that table corresponds to one column in the recorded csv file. The following columns may be recorded:

- *Channel* – shows the channel number of the corresponding channel.
- *Device* – contains the device name for which the data will be recorded and its device icon.
- *Property* – this is the name of the device property/process data value that will be recorded. Its type (numeric or boolean) can be identified by the displayed icon.

   Numeric value

   Boolean value

- *Label* – allows you to define a customized description for the selected channel. This description will be used as the column header in the csv file.

In order to add a channel to the data logging process, simply follow the steps below.

18.3 Configuration of the Logging Process

18.3.1 Step 1- Adding of Channels

Drag-and-Drop the device for which you want to log the data from the *Device List* into the *Logger Channels* list. The new channel will be inserted into the list at the desired position (see figure below).

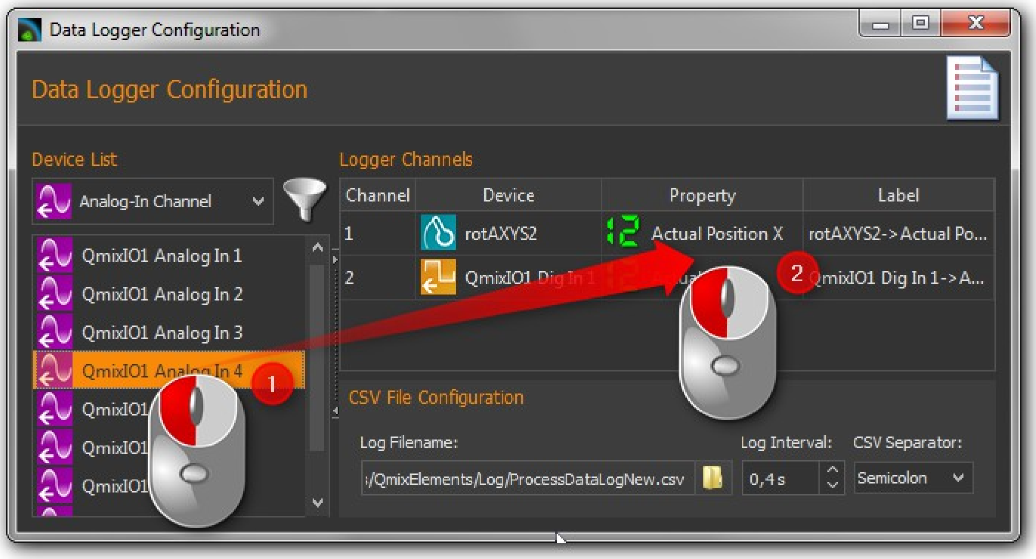
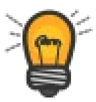


Figure 18.4: Adding channels using drag-and-drop.



HINT

To simplify the device selection, the device list can be filtered according to device type.

18.3.2 Step 2- Select Device Property

In the *Logger Channels* list you now need to select the *Property* of the device that you want to record. For this, double-click into the respective filed within the column *Property* and select the device property from the opening list (see figure below).

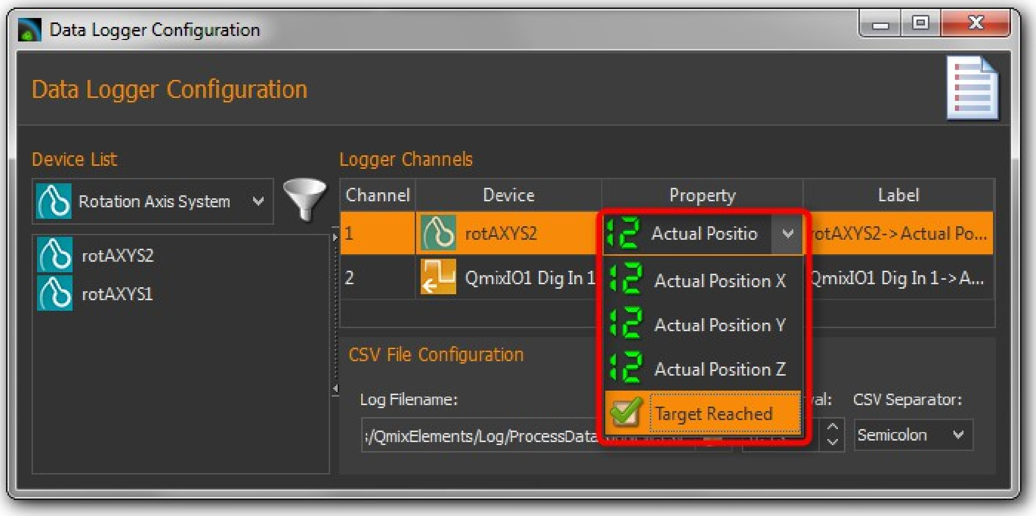


Figure 18.5: Selecting the device property that is to be recorded.

18.3.3 Step 3 – Channel Description

In the column *Label* you can customize the description for each channel. This label will be used as the column header of the csv file for the selected channel.

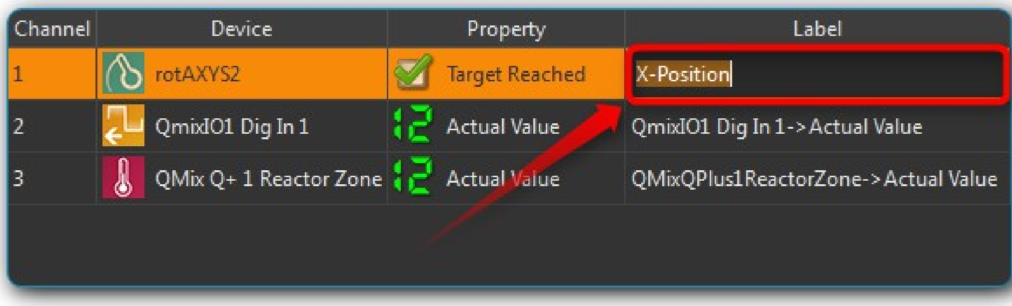


Figure 18.6: Customizing the channel label.

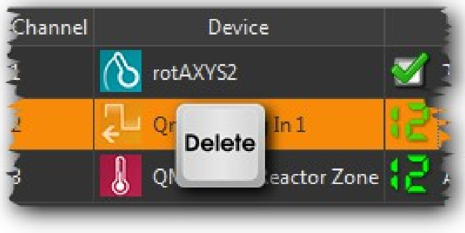
To do this, double-click into the respective table cell that is to be changed and insert the new description (see figure above).

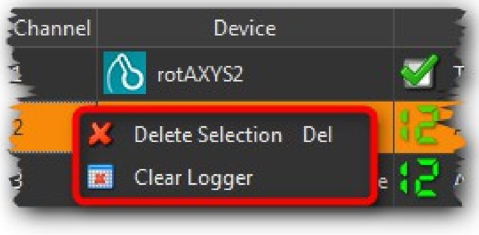
IMPORTANT

Upon choosing a new device property, a new channel description will be assigned automatically. That is, you should change the channel label only once the correct device property has been selected.

18.3.4 Deleting Channels

Highlight the desired channels using the mouse to delete one or more channels from the list, and then use either the *Delete* key or the *Delecte Selection* item of the right-click context menu:





To delete the entire channel list, use the context menu item *Clear Logger.*

18.3.5 Step 4 – Configuration of CSV Properties

In the *CSV File Configuration* section you can set the global properties of the CSV logger as well as the format of the recorded data (see figure below).

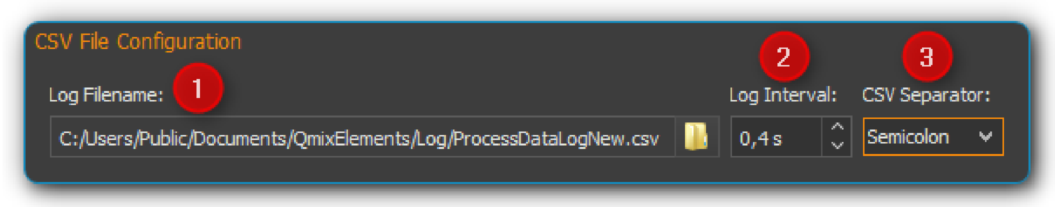


Figure 18.7: Configuration of global csv properties

18.3.5.1 Select File Name and Folder

Set the file name and location of the log file via *Log Filename* 1. For this, click on the folder symbol on the right, select the target folder and give a file name.

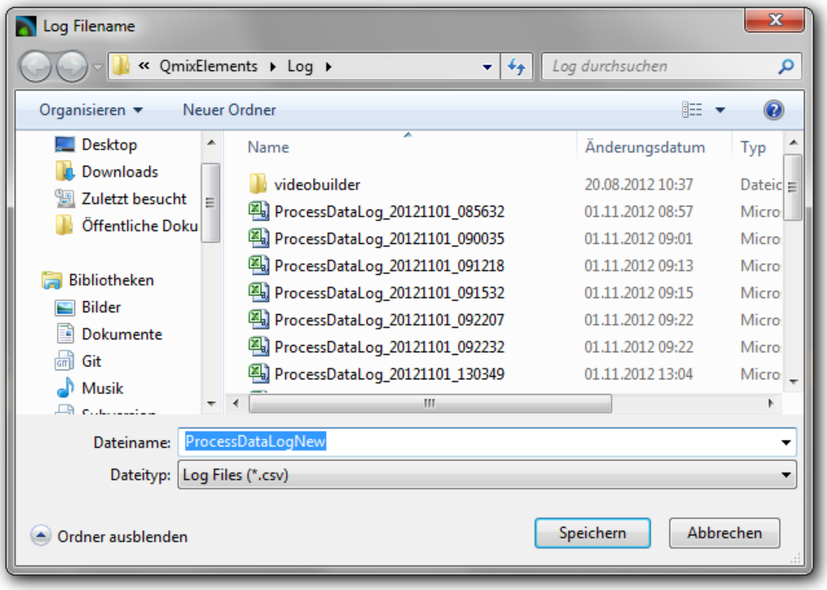


Figure 18.8: Setting file name and folder for the log file.

18.3.5.2 Setting the Recording Interval

Set the time interval at which the data of all channels is to be recorded via the field *Log Interval* 2. The time unit for the interval is seconds and you may set it in 0.1 second increments.

IMPORTANT

Choose the recording interval as large as possible and as small as necessary. This will minimize amount of data that will be recorded.

18.3.5.3 Set the Separating Character

The character that will be used to separate individual data (columns) needs to be set via the selection field *CSV Separator* 3. Depending on the software that is to be used for data evaluation, the character that needs to be used may change.

HINT

To obtain a CSV file that can be imported into Microsoft Excel, the semicolon (;) should be used.

IMPORTANT

All configuration settings of the process data logger will be saved upon closing the configuration dialogue and are available when the application will be restarted.

18.4 Start/Stop of the Logging Process

Use the relevant toolbar button to start and stop the data logging process.

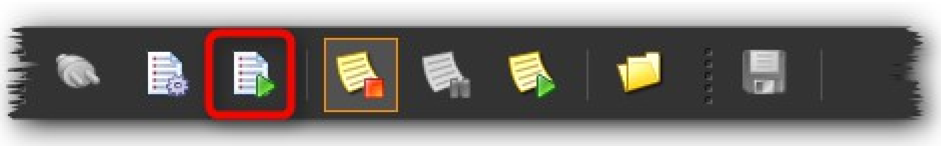


FIgure 18.9: Start/stop of data logging.

A new data file will be created each time the logging process is started. A time stamp with date and time will be added as a suffix to file name (*_YYYYMMDD_hhmmss)*. That means, if the file name *ProcessDataLog.csv* has been assigned by the user, starting the logging process on November 05, 2012 at 10:32 am and 9 seconds will create a logging file with the name *ProcessDataLog_20121105_103209.csv.*

This ensures, that a new logging file with a unique time stamp will be created each time the logging process is started.

18.5 Log File Data Format

The recorded CSV files have the following structure:

- Each CSV file consists of multiple data sets organized in rows and separated by line breaks.
- Each data set consists of a number of data fields (columns) that are separated by a specific character (e.g., semicolon).
- The first column always contains the relative time point (in seconds) of the corresponding data set.
- The first row shows the channel labels as configured by the user.

	A	B	C	D
1	Time (s)	rotAXYS2->Actual Position X	QmixIO1 Dig In 1->Actual Value	QMixQPlus1ReactorZone->Actual Value
2	0	144,5	0	-130,95
3	0,1	144,5	0	-130,95
4	0,2	144,5	0	-130,95
5	0,3	144,5	0	-125,882
6	0,4	144,5	0	-125,882
7	0,5	144,5	0	-125,882
8	0,6	144,5	0	-125,882
9	0,7	144,5	0	-125,135

Figure 18.10: CSV log file opened in Microsoft Excel.

To obtain the absolute time stamp for a data set, you may simply add an extra column to the spreadsheet and calculate the time by adding the data set's relative time to the absolute time given in the file names time stamp.

HINT

The absolute time stamp at which logging started is given in the file name of the log file.

19 Process Data Graph

19.1 Introduction

In addition to the ability to record data in CSV files, the data logger plug-in allows you to visualize process data. This allows for real-time visualization of, e.g., process data changes. Use the push button *Logging* 1 (see figure below) in the sidebar to display the process data graphs.

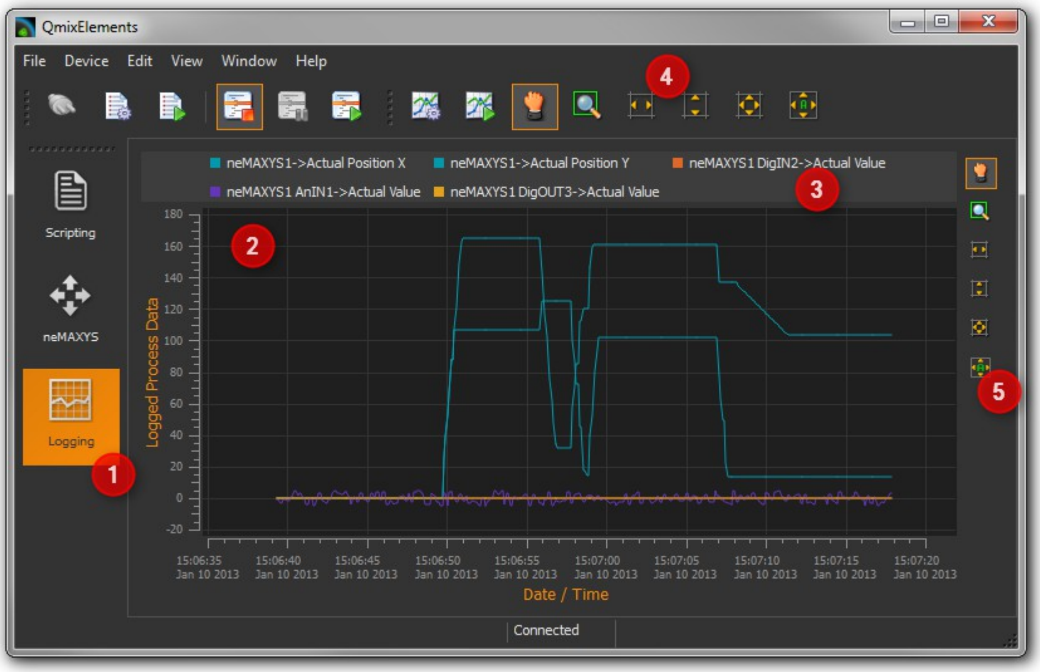


Figure 19.1: Process Data Graph

The main elements of the process data graph area are as follows:

6 *Logging selector button* – Click this to show the process data graphs.

7 *Graph canvas* – This displays the curves of all process data sets that are being recorded.

8 *Legend* – The legend lists all data sets that are displayed with their respective colors. Here you can toggle between whether or not a curve is being displayed.

9 *Toolbar* – Here you find buttons to configure the data logging, to start and stop the recording and to navigate the display.

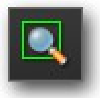
10 *Tool buttons lineup* – This contains essentially the same tool shortcuts as the toolbar.

19.2 Toolbar

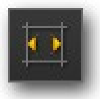
Opens the configuration dialog of the graphic process data logger.

Toggles the recording of process data.

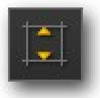
Panning tool to move the currently displayed section of the graph.



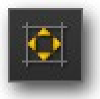
Draws a zoom-in frame to enlarge a desired area of the graph.



Auto-scales the X axis to fit all process data on the screen.



Auto-scales the Y axis to fit all process data on the screen.



Auto-scales both X and Y axes to fit all process data on the screen.

Activates auto-scaling: during a recording, both x- and y-axes are continuously rescaled to fit all process data on the screen.

19.3 Configuration Dialog

19.3.1 Overview

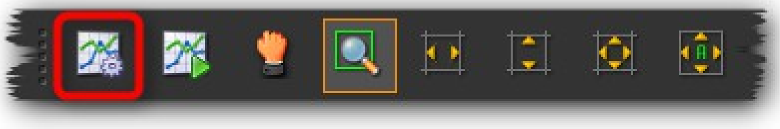


Figure 19.2: Toolbar data logging

Use the toolbar button shown above to open the configuration dialog.

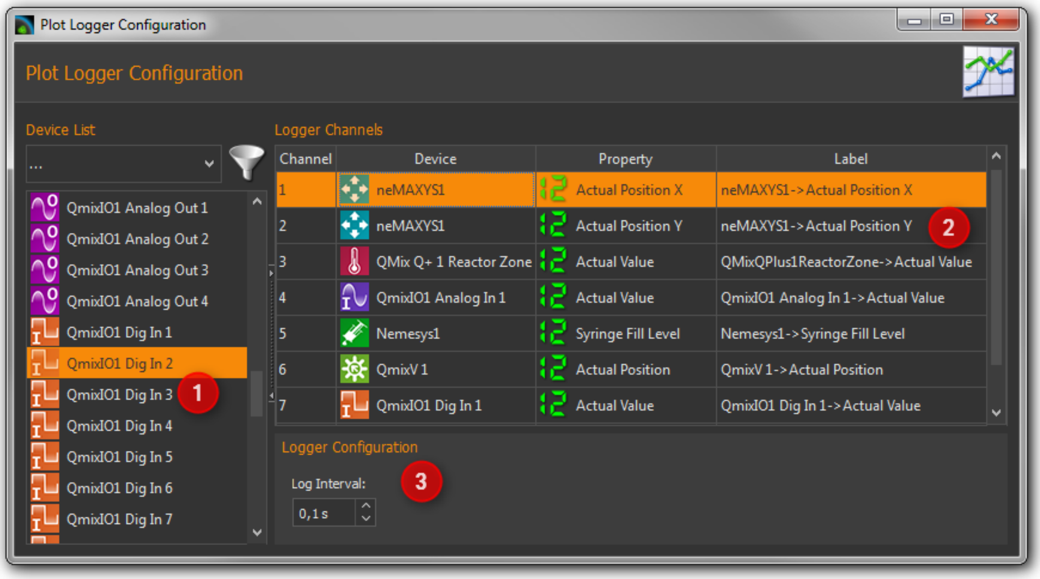


Figure 19.3: Configuration dialog of the graphic data logger

This opens the *Plot Logger Configuration* dialog that contains the following main sections:

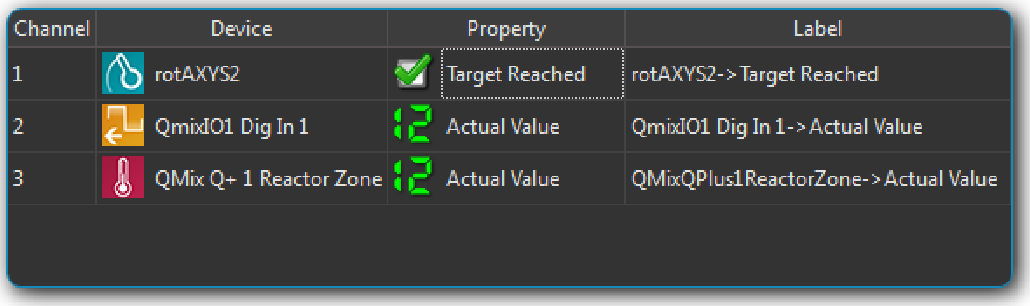
1 *Device List* – shows all devices that return data that may be logged. The filter selection box allows to pre-select a specific device type (e.g., valves).

2 *Plot Curves* – lists all data series or curves that are being recorded and displayed in the diagram.

3 *Logger Configuration* – in this section you find various settings to

configure the data recording.

19.3.2 *Plot Curves* Table



Channel	Device	Property	Label
1	rotAXYS2	Target Reached	rotAXYS2->Target Reached
2	QmixIO1 Dig In 1	Actual Value	QmixIO1 Dig In 1->Actual Value
3	QMix Q+ 1 Reactor Zone	Actual Value	QMixQPlus1ReactorZone->Actual Value

Plot Curves tabulates the selected configuration of the graphic data logger. Each row represents exactly one curve in the diagram. The following columns are shown:

- *Channel* – returns the channel number.
- *Device* – lists the device name for each respective channel including its icon.
- *Property* – shows the property of the respective device that is to be recorded. The data type is identified via a data-type specific icon.

 Numerical value

 Boolean value

- *Label* – allows you to define a user-specific name for each channel. This label will also be used in the legend of the plotted graph.

To add and configure process data channels to the display logger, please proceed as detailed in the following sections.

19.4 Configure data logging

19.4.1 Step 1 – Adding Channels

To add a channel you first have to add the relevant device to the *Device List* of the *Plot Logger Configuration.* To do this, move the relevant item from the device list to the *Plot Curves* table using *Drag-&-Drop*. The new channel will be added at the position where you release the mouse button (see figure below).

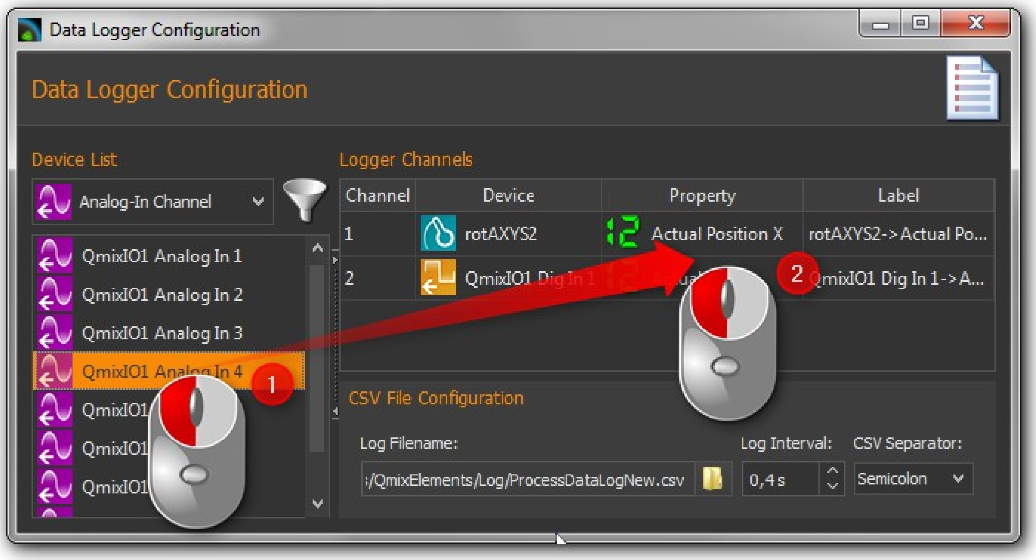


Figure 19.4: Adding a channel to the channel list

TIP

To simplify the device selection process, the device list may be filtered for a relevant device type.

19.4.2 Step 2 – Selecting the Device Property

Select the device property that you want to record by double clicking into the *Property* field of the respective channel from *Plot Curves* table. This will display a drop-down list with all available device properties from which you may select the desired item (see figure below).

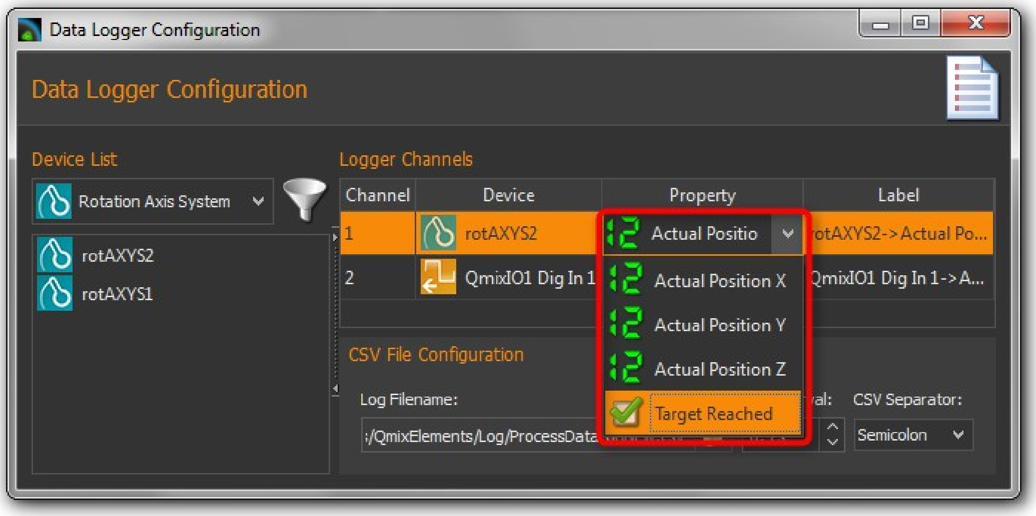


Figure 19.5: Selecting the device property to be recorded

19.4.3 Step 3 – Changing the Channel Label

You may give a recorded property a customized name by changing the description in the column *Label*. This label will also be used to identify the respective curve in the diagram. To do this, double click into the respective field (see figure below) and type the new description.

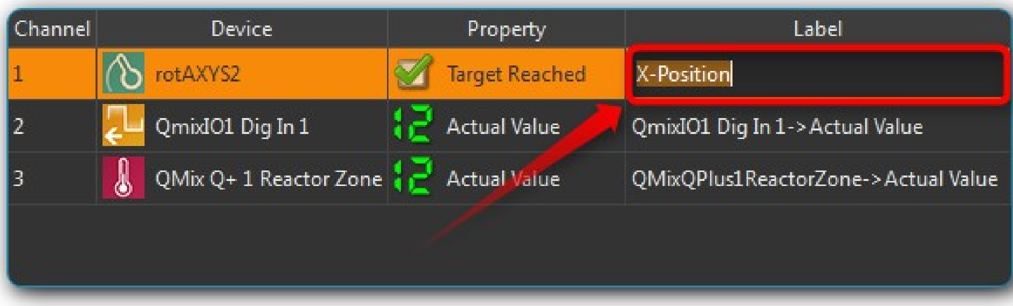


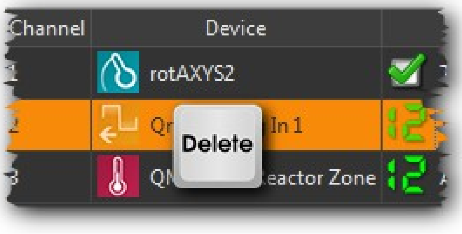
Figure 19.6: Changing the channel label

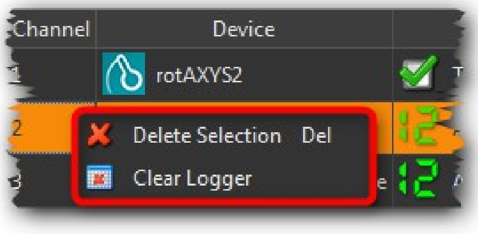
IMPORTANT

When a different device property is being selected, a new channel label will be assigned automatically. Therefore, the channel label should be changed after the device property has been selected.

19.4.4 Deleting Channels

In order to delete one or multiple channels from the *Plot Curves* list, first you have to mark the respective channels using the computer mouse. Now you may use either the keyboard's *Delete* key or select the relevant item (*Delete Selection*) from the right-click context menu.





You may also delete the entire list in a single step by using the *Clear Logger* item of the context menu.

19.4.5 Step 4 – Defining the Recording Interval

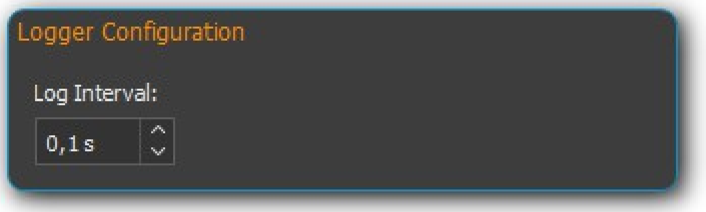


Figure 19.7: Configuration of the data logger interval

The *Log Interval* input box in the *Logger Configuration* section is to define the time interval at which data points for all channels are to be recorded. The minimum resolution is 0.1 seconds.

IMPORTANT

Choose a log interval that is as large as possible and as small as necessary in order to minimize the amount of data that needs to be recorded and transmitted by the system.

The configuration will be saved and reloaded automatically upon exiting the *Plot Logger Configuration* dialog.

19.5 Start/Stop Data Logging

The data logging process may be started/stopped via the relevant button in the toolbar.

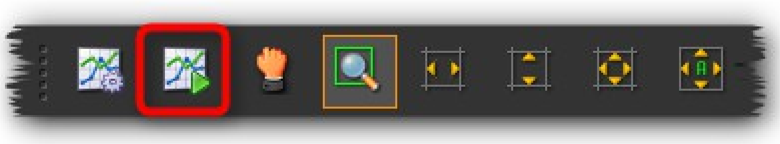


Figure 19.8: Start/stop data logging process

19.6 Diagram Navigation & Use

19.6.1 Overview

The processdata graphing plug-in offers a number of possibilities to customize the way data are displayed. This includes resizing parts of a curve and showing or hiding individual curves.

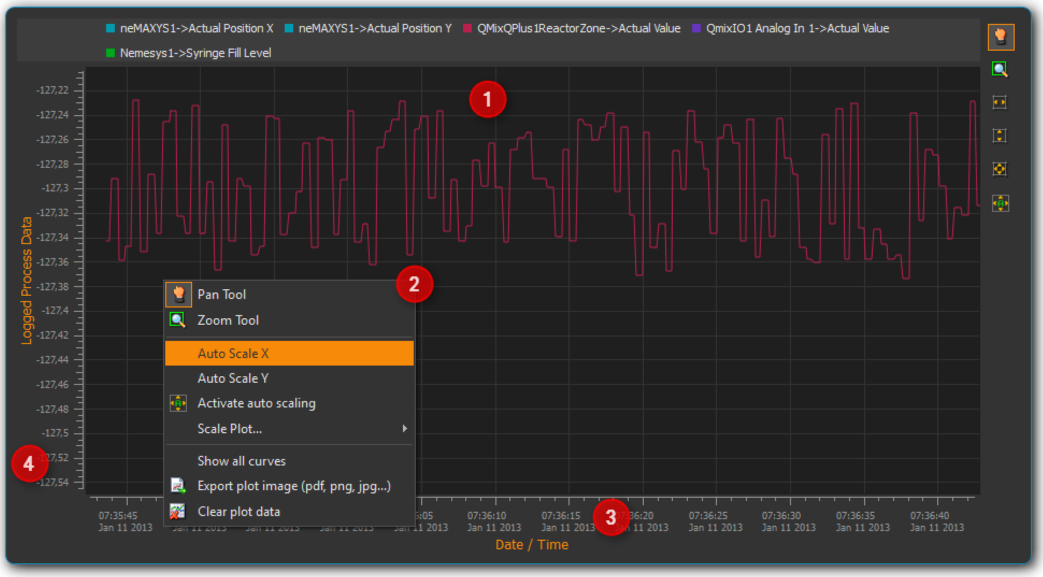


Figure 19.9: The process-data diagram section

The diagram consists of a plot area 1 plus both an X-axis (time) 3 and a Y-axis (process data) 4. The time axis shows date and time as absolute values. The process-data axis shows the respective measurement data; it is without units as it potentially represents a variety of very different values and measurement units.

A right mouse click within the plot area will open a context menu 2 with a number of additional functions.

19.6.2 Changing the Displayed Section

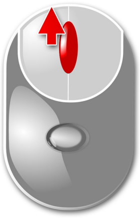
The *Pan Tool* provides you with a simple way to move the displayed section of the plot area. It may be activated via its toolbar button and the displayed section may then be moved around using the mouse whilst keeping the left button pressed.

IMPORTANT

Panning of the displayed plot section will deactivate the auto-scaling of the diagram axes.

19.6.3 Zooming via the Mouse Wheel

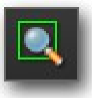
Turning the mouse wheel whilst the pointer is within the plot area will allow you to adjust the displayed section of a graph by increasing (zooming in) or decreasing (zooming out) its zoom level.



Increase zoom level (zoom in)

Decrease zoom level (zoom out)

19.6.4 Defining a Display Section



The *Zoom Tool* allows you to directly select a specific area of the plot and increase its resolution. To do this, please proceed as follows (see figure below):

1 Using the mouse, left-click-and-hold into the plot area to set the first corner of the zoom frame.

2 Move the mouse pointer to define the size of the frame as desired.

3 Releasing the mouse button will finalize the size of the frame. The selected area will be scaled to the current graph size (zoom in).

Figure 19.10: Setting a zoom frame

19.6.5 Auto-Fit & Auto-Scale

The toolbar and the context menu both contain a number of tools to adjust what is displayed in the diagram, in particular to ensure that all or specific data are visible.

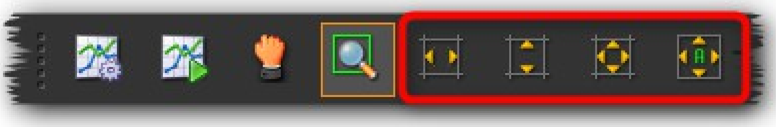
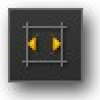
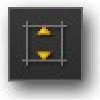


Figure 19.11: Toolbar buttons for automated scaling

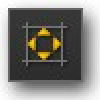
The following possibilities exist:



Rescales the X axis to display all current time data values for a given process data resolution.



Rescales the Y axis to display all current process data values within a given time period.



Rescales both X and Y axes to display all currently available data.

(Re-)activates auto-scaling: as long as data are being recorded, both X and Y axes will be adjusted dynamically to ensure all data are being displayed.

You may also activate auto-scaling for X and Y axes individually via the context menu:

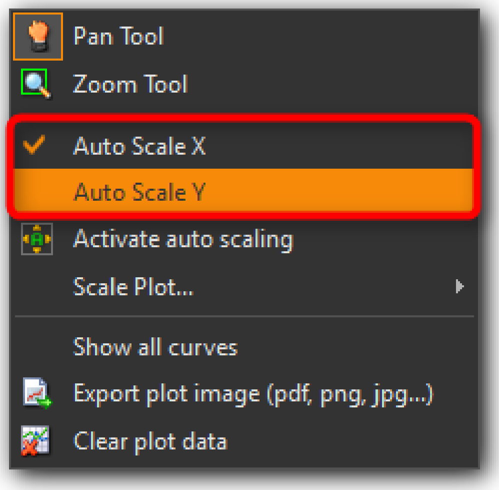


Figure 19.12: Auto-scaling toggle for X and Y axes

IMPORTANT

Zooming or panning within the displayed plot section will deactivate auto-scaling.

19.6.6 Show/Hide Individual Curves

To improve scaling and visibility, you may show or hide individual curves. To do this, right-click the desired item in the plot legend and select the desired function to either hide the corresponding curve only (*Hide Curve*) or all other but the corresponding curve (*Show only this curve*) as indicated in the figure below.

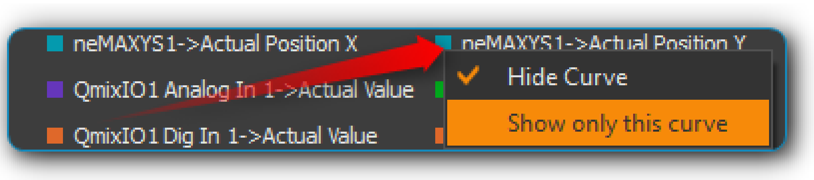
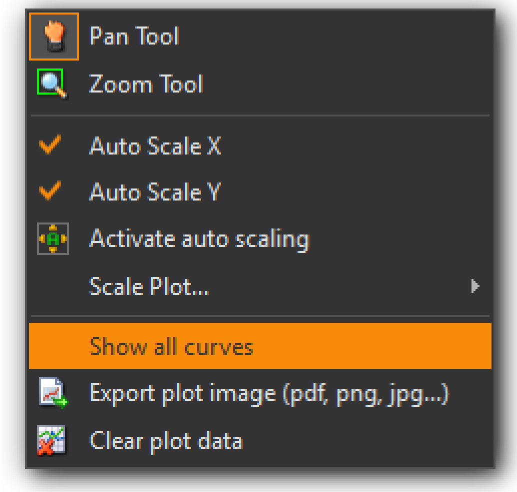


Figure 19.13: Context menu legend item

To revert to displaying all curves, activate the context menu from within the plot area and select the menu item *Show all curves* (see figure below).



19.6.7 Select Curve Color

To choose a different curve color, right click an item in the plot legend. From the context menu select the menu item *Select Color* (see figure below).

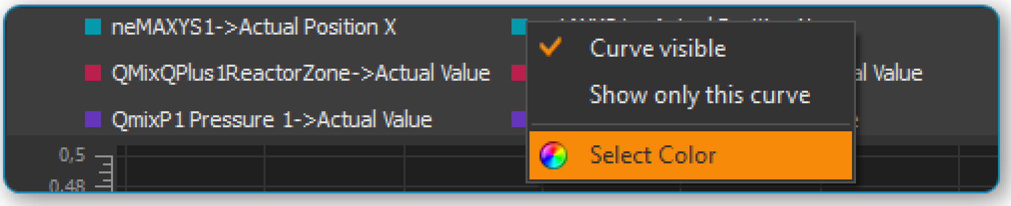


Figure 19.14: Context menu legend item – Color Selection

In the color selection dialog which is now shown (figure below), you can choose any color.

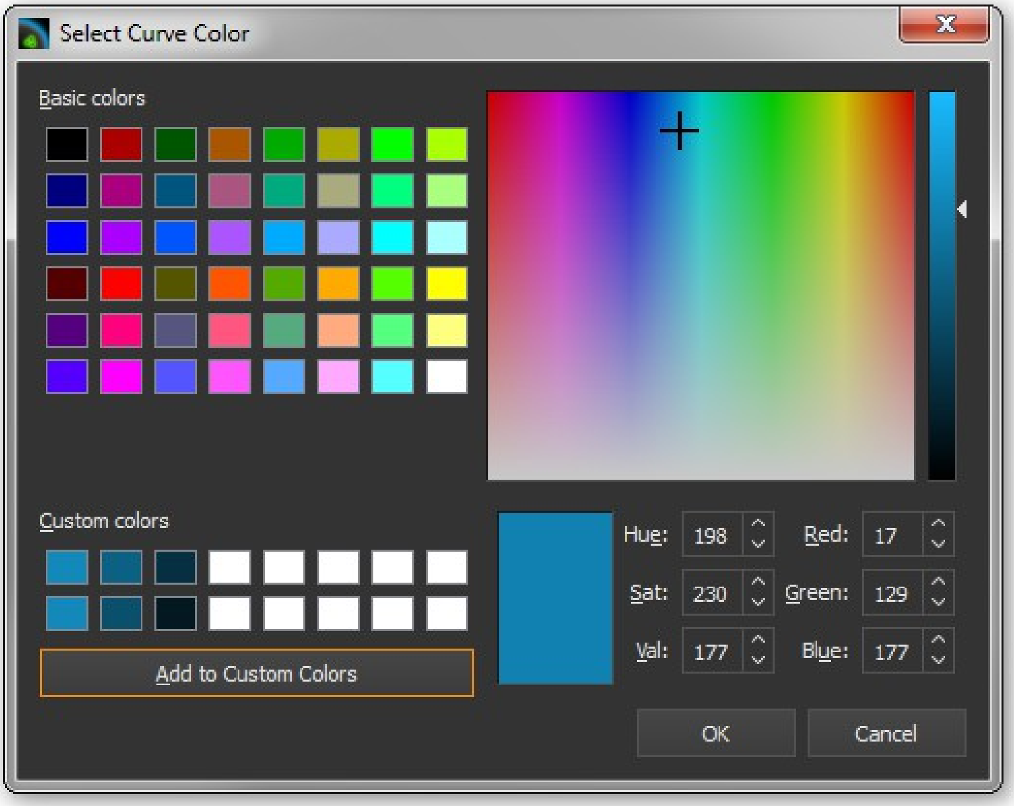


Figure 19.15: Color Selection Dialog

19.6.8 Exporting a Diagram Image

You may export a picture of the current diagram using the right-click context menu and selecting *Export plot image*.

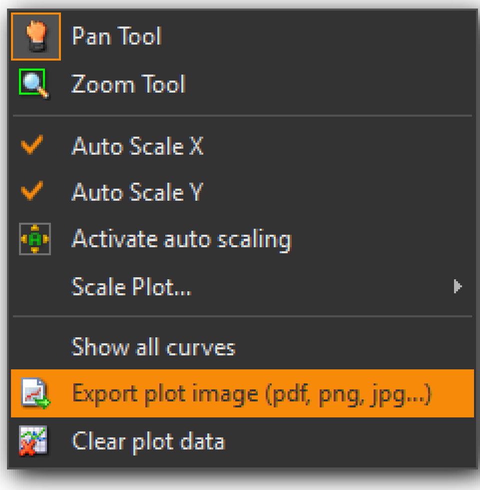


Figure 19.16: Diagram image export

This will open a dialog box (see figure below) to define the location (folder) where the image is to be saved:

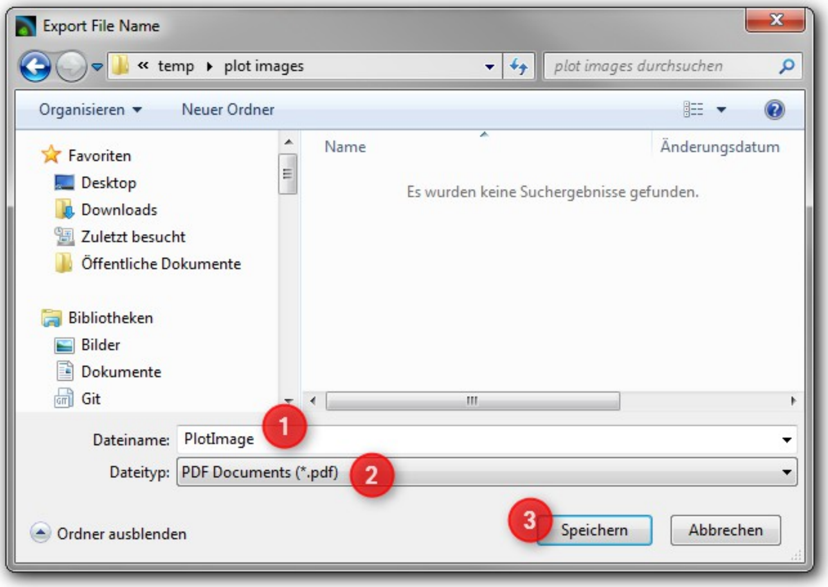


Figure 19.17: Diagram image export dialog

Please enter a name for the image file 1 and select the desired file type 2. The export function supports standard image file formats (*png, jpg...*) as well as scalable vector graphic formats (*pdf, svg...*).

To close the dialog and to start the image export, click *Save* 3.

19.6.9 Deleting of Diagram Data

You may clear the plot area and delete all data recorded since the start of the present recording using the context menu item *Clear plot data*. Recording will resume from this point.

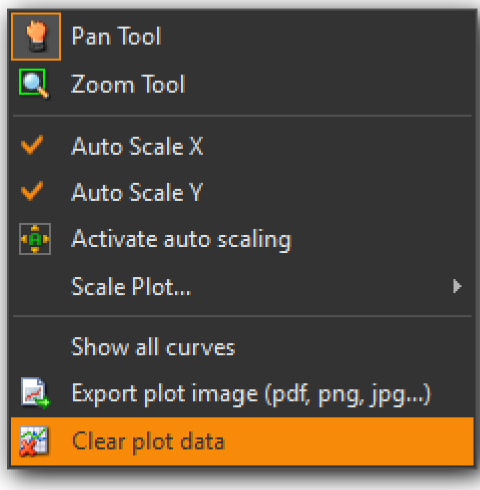


Figure 19.18: Delete diagram data