SM148E

PIMikroMove

Software Manual

Version: 2.13.0 Date: 18.04.2019

This document describes the following product:

- **PIMikroMove**

Physik Instrumente (PI) GmbH & Co. KG, Auf der Roemerstrasse 1, 76228 Karlsruhe, Germany
Phone +49 721 4846-0, Fax +49 721 4846-1019, Email info@pi.ws, www.pi.ws

Original instructions
First printing: 18.04.2019
Document number: SM148E, ASt, Version 2.13.0

Contents

1 Introduction

PIMikroMove is a general purpose graphical user interface for PI motion controllers. You can connect as many controllers or controller networks as you have connected to your PC. Some of the available menus and their items depend on the connected controller type so that not all of the menus, items and windows described in this manual will always be available in your current instance of PIMikroMove.

In this Chapter

1.1 About This Document

1.1.1 Objective and Target Audience of this Manual

This manual assumes that the reader has a fundamental understanding of basic servo systems as well as motion control concepts and applicable safety procedures.

For updated releases of this manual, or if you have any questions, contact our customer service department (mailto:service@pi.de).

1.1.2 Other Applicable Documents

The devices which are mentioned in this documentation are described in their own manuals.

For the latest versions of the user manuals contact our customer service department (mailto:service@pi.de).

1.1.3 Symbols and Typographic Conventions

The following symbols and typographic conventions are used in this manual:

NOTICE

Dangerous situation
Failure to comply could cause damage to equipment.

➢ Precautionary measures for avoiding.

INFORMATION

Information for easier handling, tricks, tips, etc.

Symbol	Meaning
Start > Settings	Menu path in the PC software (example: to open the menu, the ***Start*** and ***Settings*** menu items must be clicked in succession)
`POS?`	Command line or a command from PI's General Command Set (GCS) (example: Command to get the axis position).
Device S/N	Parameter name (example: Parameter where the serial number is stored)
5	Value that must be entered or selected via the PC software

1.2 Initial Installation

INFORMATION

Additional components are required when installing PIMikroMove. PIMikroMove supports a broad range of controllers from PI. To handle the necessary communication with a controller, the program relies on the GCS DLL for the specific controller, and, for some controllers, also on the PIStages database. For every new controller you want to connect to PIMikroMove, you have to ensure its GCS DLL is installed (and database, if required) by running the general setup procedure. If you simply start PIMikroMove without the necessary DLL(s) and database you will get an information message "No GCS DLLs found. PIMikroMove needs these libraries to connect to the PI controllers".

PIMikroMove is provided on the data carrier, e.g. CD, of supported controllers and can be installed via a setup procedure:

1. Connect the data carrier with the host PC.
2. If the Setup Wizard does not start automatically, start it from the root directory of the data carrier by doubleclicking the icon or the ***setup.exe*** file.
3. Follow the on-screen instructions. You can choose between "Complete" and "Custom" installation. "Complete" installs all components: programs, drivers, manuals, programming libraries and samples. With "Custom" you can choose which components you want to install. This option should only be selected by advanced users. "Complete" is recommended.

1.3 Installing Updates

INFORMATION

All customized settings (e.g. settings for user stages or position settings) are retained when PIMikroMove is updated.

INFORMATION

It is strongly recommended to always use the latest versions of PIMikroMove, GCS DLL and PIStages database. The software is continuously kept up-to-date so that the version on the product's data carrier used for initial installation may be out of date.

➢ Use the PI Update Finder to find and download updates for the software.

➢ Always install the latest version of the software.

Prerequisites

- If the PI Update Finder is not on the product's data carrier: You have downloaded the PI Update Finder (http://update.pi-portal.ws) (PIMikroMove provides the ***Help > Download the PI Update Finder*** menu sequence) and unpacked the Zip file into a directory on the host PC.
- You have the Technical Note "PI Update Finder" (A000T0028) at hand. This document is on the product's data carrier or in the Zip file for the PI Update Finder.
- The PC is connected to the Internet.
  - If the PC is **not** connected to the Internet:
    You have the Technical Note "Updating PI Software" (A000T0032) at hand. This document is on the product's data carrier or in the Zip file for the PI Update Finder.

Updating software

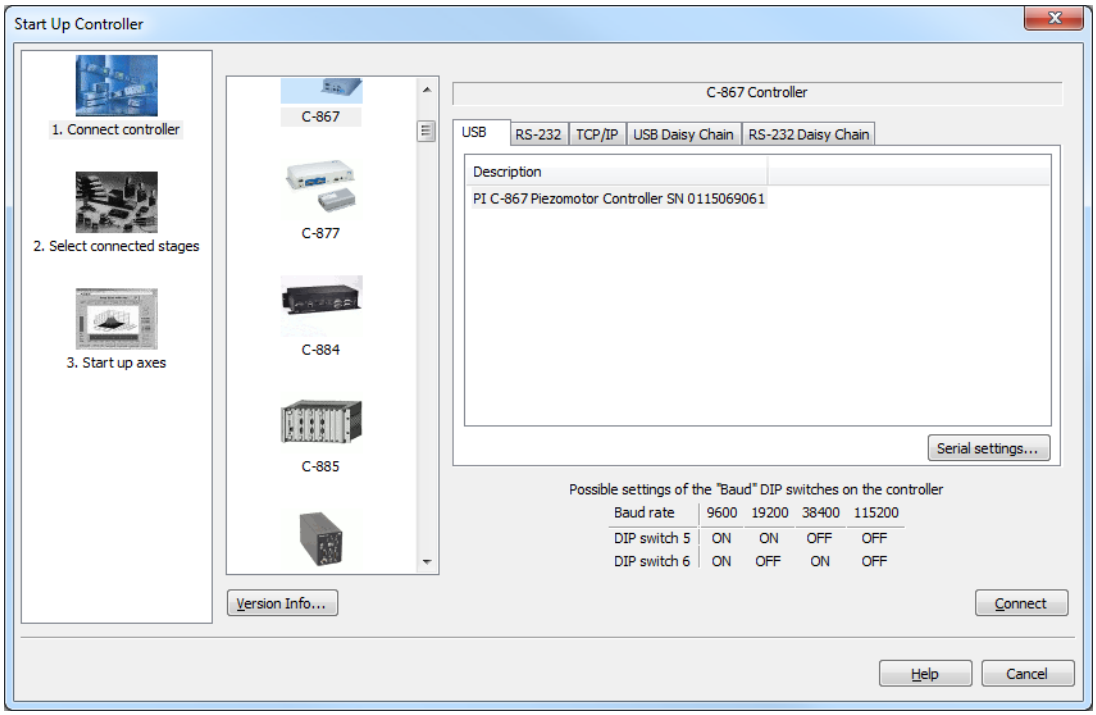
1. Download the update (Zip file) using the PI Update Finder:
   - Follow the instructions in the Technical Note for the PI Update Finder (A000T0028).
   - If the PC is **not** connected to the Internet: Follow also the instructions in the Technical Note "Updating PI Software" (A000T0032).
2. Install the update:
   - Follow the instructions in the documentation (Releasenews file or readme file) which accompanies the update Zip file.
   - To update PIMikroMove, run the downloaded ***PIMikroMove_Setup.exe*** file.

1.4 Getting Started

Set up and installation of the motion system is described in the user manuals of the controller(s) and mechanics.

To start PIMikroMove use the appropriate item on the ***Start*** menu (by default under ***Start > Programs > PI***).

When PIMikroMove starts, the ***Start up Controller*** (p. 169) window is displayed at the first step, ***Connect controller***. Here is where you connect controllers to PIMikroMove.



Only one controller may be selected at a time, but the dialog can be reopened to connect further controllers to the current session. In the main window of PIMikroMove, use ***New*** connection on the toolbar or the ***Connections > New...*** menu sequence to do this. See "Connect Controller" (p. 170) for more information, especially regarding daisy chain connections.

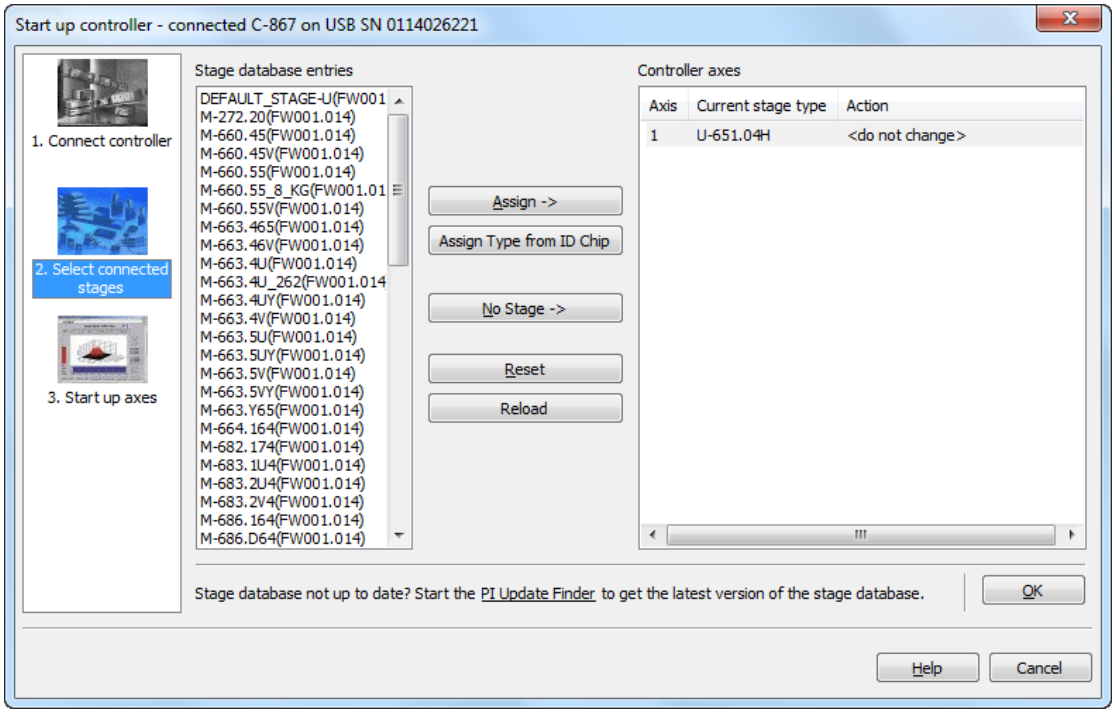
INFORMATION

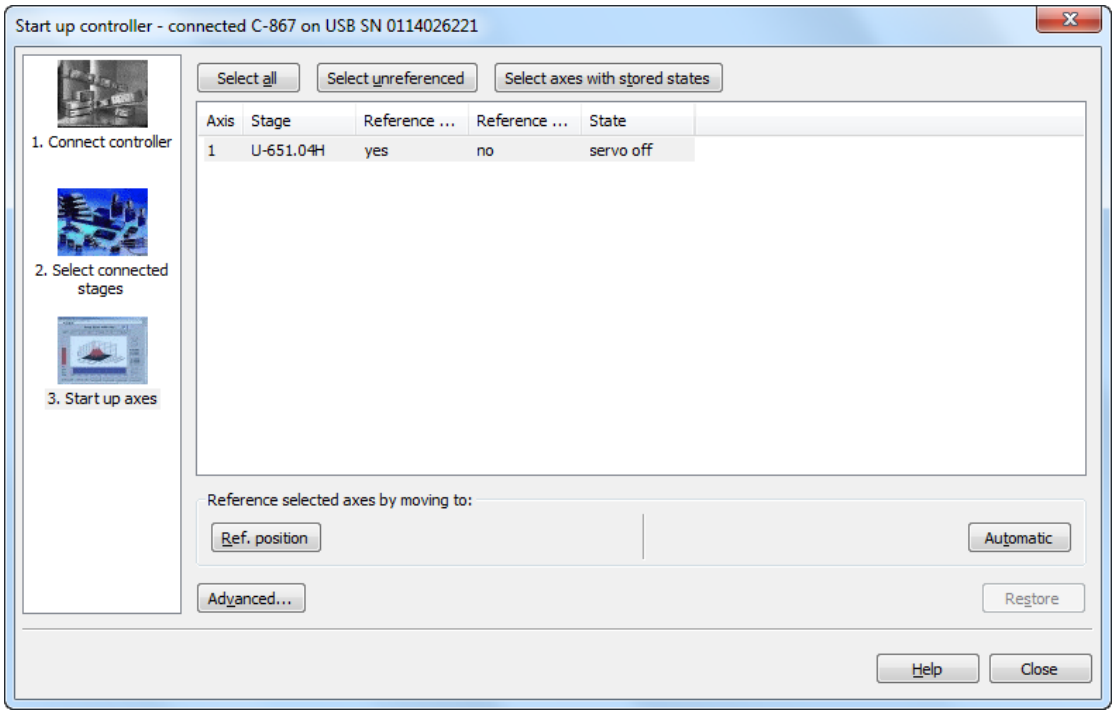
If the controller is connected via a USB port, this USB interface might appear as an additional COM port in the port selection list.

After the connection was established, PIMikroMove guides you through the further steps of the startup procedure. It depends on the controller which of the following steps are necessary:

- Controllers which support loading settings from a stage database and are not preconfigured for a certain stage type:
  Select connected stages (p. 172)
- E-518 interface module:
  Start up E-518 (p. 174)
- System whose axes have to be initialized (e.g. referenced) before normal use:
  Start up axes (p. 175)

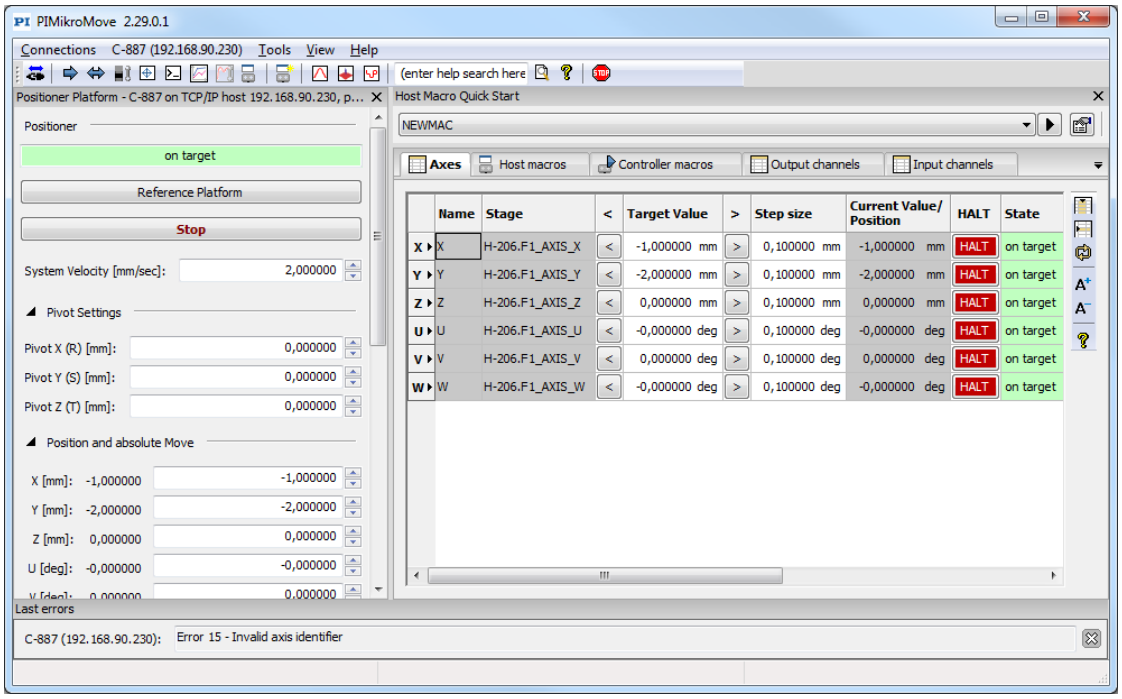
The screenshots below show the steps for loading the stage settings from a database and for axis referencing.





When the ***Start up controller*** window is closed, the PIMikroMove main window will open. Furthermore, single-axis windows may open for the available axes. You may need to resize or move them to see the main window.

2 Main Window

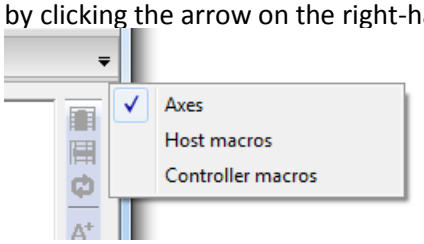


The main window consists of several elements:

- Menu bar (p. 12)
- Toolbar (p. 23)
- Pane with tab cards for:
  - Tables for axes, piezo channels, drive channels, sensor channels and PiezoWalk® channels, see "Table Tab Cards" (p. 25)
  - Host macros (p. 38)
  - Controller macros (p. 57) (if supported by one or more connected controllers)
  - Positioner 3D view (p. 31): 3D visualization of the parallel-kinematics positioner, for every parallel-kinematics system connected to PIMikroMove
  - Fast Alignment (p. 32): Definition, display, import and export of fast alignment routines
  - Trajectory Assistant (p. 35) (if supported by a controller and called from the controller's menu)
- Message line: At the bottom of the main window, the last error returned by a connected controller is displayed.

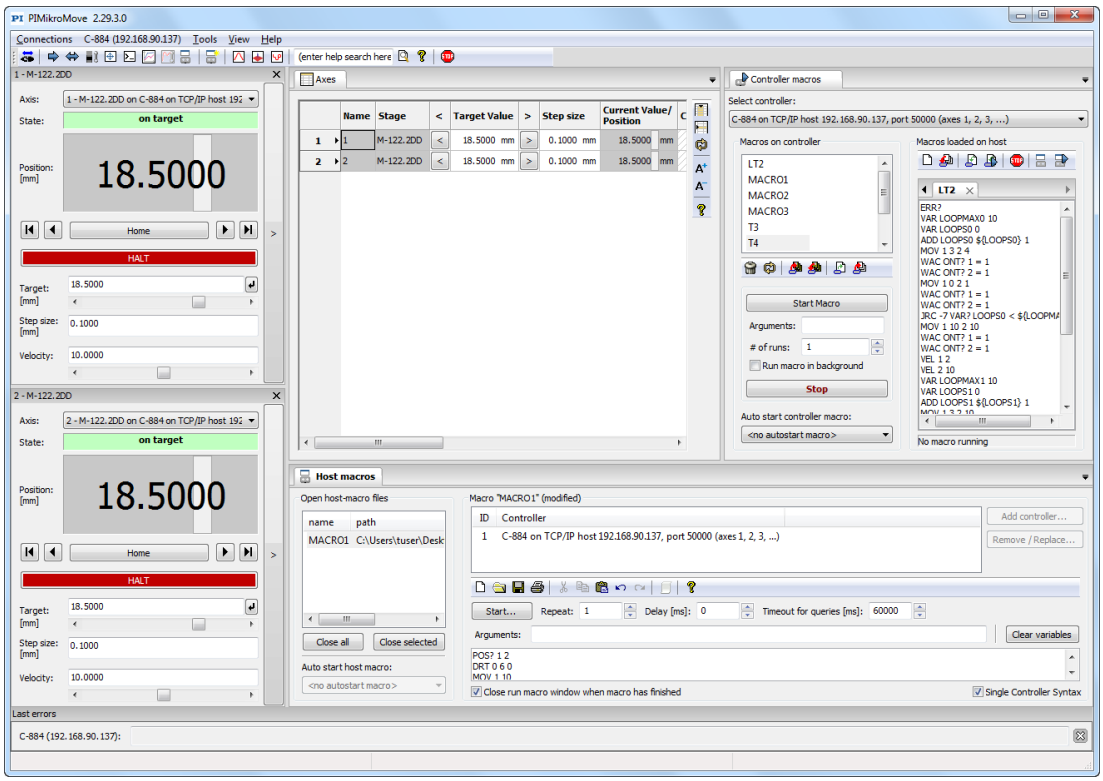
Tab cards can be selected

- by clicking on the tab
- by clicking the arrow on the right-hand side of the tab bar and selecting the tab



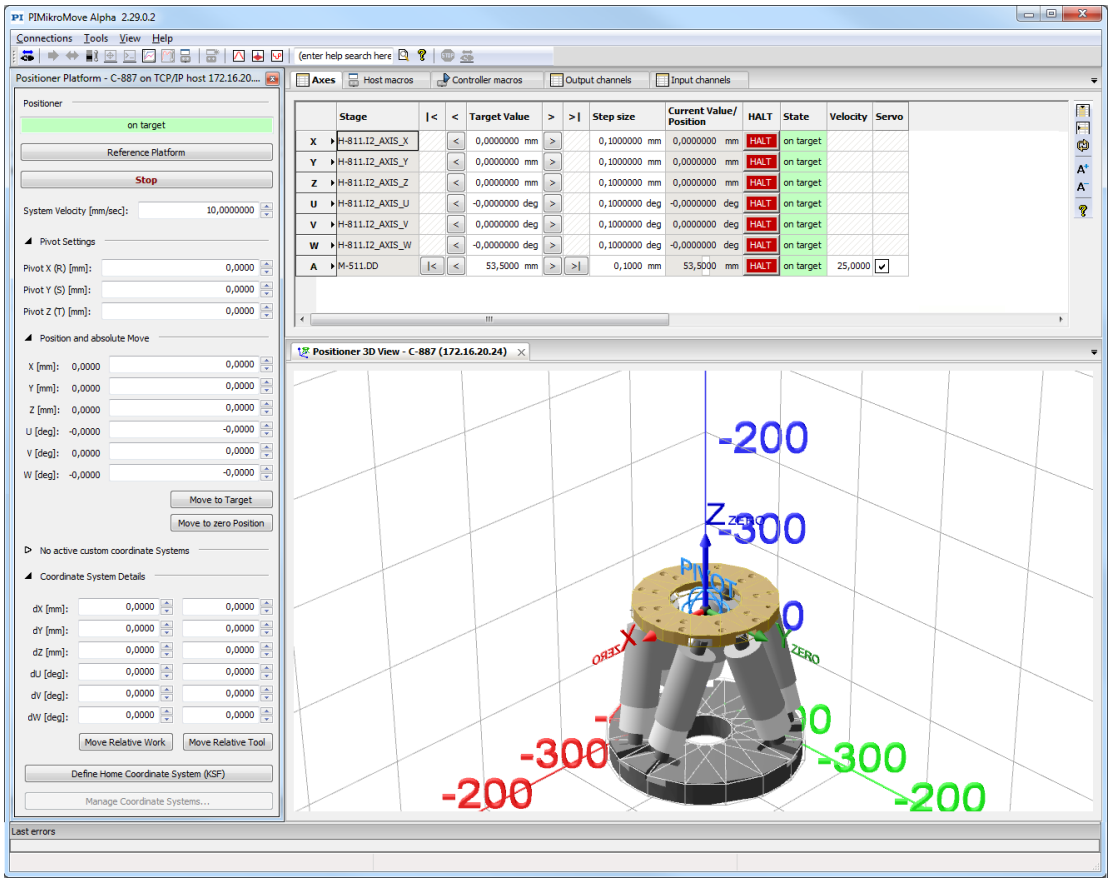
The tab cards can be flexibly arranged in the main window by dragging them with the mouse. It is also possible to display multiple tab cards in the main window by dragging them with the left mouse button pressed.

If a new controller is connected which provides sensor and piezo channels, the corresponding tab cards will be arranged automatically in the main window so that all axes and channels are visible. Exception: If the tab cards were arranged before, PIMikroMove will not change the appearance of the main window.



Two single-axis windows and the ***Axes***, ***Controller macros*** and ***Host macros*** tab cards, all docked inside the main window

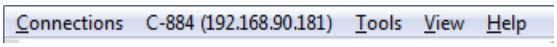
Further windows can be displayed, either as separate windows or docked to the main window. See "Additional Common Windows" (p. 61) and "Controller Specific Windows" (p. 103) for more information.



Main window showing the ***Positioner 3D View*** tab card and the ***Positioner Platform*** window (docked)

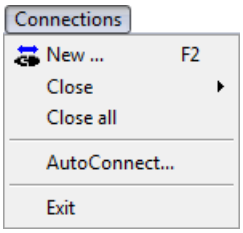
In this Chapter

2.1 Menu Bar



Note that most of the menu functions are disabled when the ***Command entry*** window (p. 64) is in locked state.

2.1.1 Connections Menu



New...

Opens the ***Start up Controller*** window (p. 169) to connect another controller or controller network.

Close

Displays a submenu with entries for each connected controller/controller network. By selecting one of the entries, the connection to that controller can be closed. Once a connection is closed, the associated controller menu is no longer displayed in the menu bar. When you close the last connection to a Daisy chain network you are prompted to close the interface completely.

Close all

Closes connections to all connected controllers. All controller menu-bar items will disappear.

AutoConnect...

Opens the ***Auto Connect Configuration*** window (p. 178) where you can change the current AutoConnect settings.

Exit

Closes all connections and exits PIMikroMove.
Depending on the program settings, you are prompted to confirm the closing of the controller connections and to specify if you want to activate the AutoConnect function for one or more connections.

2.1.2 Controller Menu

For each connected controller or network of controllers there is one controller entry in the menu bar. The text indicates the class of controller and the interface used for the connection. The items contained in the controller menus depend on the controller class—not all items are always available.

Select connected stages…

Opens the ***Start up stages/axes*** window with the step ***Select connected stages*** (p. 172). Here you can assign stage axes to the controller's axes. With most controllers, one entry is required for each motion axis.

Start up axes…

Opens the ***Start up stages/axes*** window with the step ***Start up axes*** (p. 175). Here you can reference the controller's axes.

Show data recorder...

Available if the controller has a data recorder.

Opens the ***Data Recorder*** window (p. 125) where you can analyze the recorded data and configure recording and display.

Show wave generator...

Available for controllers with wave generator.

Opens the ***PI Wave Generator Tool*** (p. 103) where you can create waveforms, configure the data recorder, administer the Dynamic Digital Linearization (DDL) feature and start the wave generator(s).

Show frequency generator...

Available for controllers with wave generator functionality.

Opens the ***PI Frequency Generator Tool*** (p. 112) where you can create waveforms to be temporarily saved in the controller. While the wave generators are permanently assigned to the controller axes, the waveforms can be assigned to the wave generators and thus to the axes as desired.

Show Positioner Platform Settings

Available for Hexapod and parallel kinematics controllers.

Opens the ***Positioner Platform*** (p. 151) window where you can set the pivot point coordinates, enter target positions and start or stop motions for the moving platform of the connected mechanics. By default, this window is displayed and docked to the left of the main window.

Show Embedded Scan Window

Available if supported by the controller and if the controller has at least one analog input channel.

Opens a submenu from which you can select a scan algorithm. This opens the corresponding ***Embedded Scan & Align Algorithms*** window (p. 156).

Show wave table editor…

Opens the ***Wave Table Editor*** window (p. 122) where you can define wave table points for later output, check them in a graphics pane and save them to the controller or to data files on the host PC.

Start wave table output…

Opens the ***Start wave table*** window (p. 123) where you can start wave table output and select the start mode (immediately or by external trigger pulses).

Stop wave table output…

Opens a window (p. 124) where you can stop the wave table output for selected axes.

Enable triggered move...

Opens the ***Enable Triggered Move*** window (p. 124) where you can specify if axis motion is to be controlled by external trigger pulses. If triggered motion is enabled, each pulse causes a relative step of a predefined size, and other control sources may not be accepted.

Configure trigger output...

Opens the ***Configure Trigger Output*** window (p. 136) where you can define trigger conditions and activate them for the digital output lines of the controller.

Show Fast Alignment Window

Available if supported by the controller.

Opens the ***Fast Alignment*** tab card (p. 32) in which you can define fast alignment routines and display the results.

Configure controller joystick(s)...

Available if the controller supports axis control by joystick.

Opens the window ***Configure Controller Joystick*** (p. 139) where you can configure and activate joystick control for the individual controller axes. Only joysticks directly connected to the controller(s) are affected here. If you have a joystick connected to the host PC see "Configure PC HID Control Window" (p. 93) for how to use it.

Calibrate controller joystick(s)...

Opens the window ***Joystick Calibration*** (p. 137) for the selected joystick device. In this window, you can select a predefined lookup table or create a custom lookup table for the individual joystick axes.

Configure controller HIDevice(s)...

Available if the controller supports axis control by HID.

Opens the window ***Configure Human Interface Devices*** (p. 140) where you can configure and activate HID control for the individual controller axes. Only HIDs directly connected to the controller(s) are affected here. If you have an HID connected to the host PC see "Configure PC HID Control Window" (p. 93) for how to use it.

Trajectory Assistant

Opens the ***Trajectory Assistant*** tab card (p. 35) in which you can define and run trajectories conveniently.

Dynamic Tuner

Available for piezo controllers.

Opens the window ***Piezo Dynamic Tuner*** (p. 77) in which you can modify and optimize the control parameters of the piezo servo controller, e.g. in case the load has been changed.

Positioner 3D View

Positioner 3D View > Show displays the ***Positioner 3D View*** (p. 31) tab card with a visualization of the positioner's current position.

Positioner 3D View > Change visualization… is available when the ***Positioner 3D View*** tab card is displayed in the main window. The function opens the ***Positioner 3D View - Visualization*** window where you can load a DAT file when PIMikroMove cannot find suitable data for the visualization of the connected positioner (e.g. when a customized Hexapod is used and you have received a special DAT file from PI).

Show service tools…

Opens the window ***Hexapod Service Tools*** (p. 160) where you can perform tests of the Hexapod struts. You should perform Hexapod strut tests only for diagnosis if failure of the Hexapod system occurs.

Show hexapod tuner…

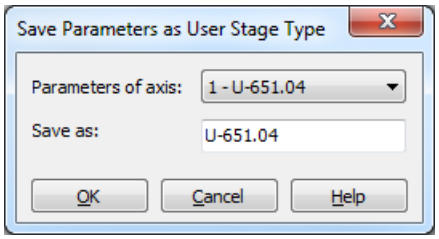
Opens the window ***Hexapod Tuner*** (p. 164) where you can adjust the settings of a Hexapod system.

Parameter Configuration

Opens the ***Device Parameter Configuration*** window (p. 144) where you can check and edit the controller parameters.

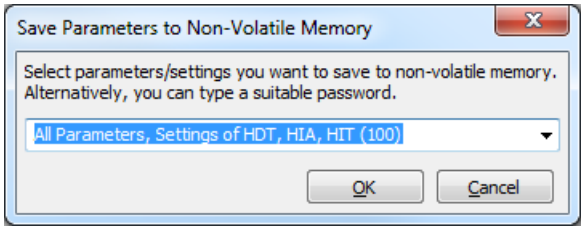
Save parameters as User Stage type...

The current settings of a stage can be stored as a new user-defined stage type in the stage database. In the dialog that is opened enter a name for the user stage entry in the stage database, and click ***OK*** to save the parameters. If an entry of the same name already exists in the database, its settings will be overwritten when the new ones are saved.



Save parameters to non-volatile memory

Opens a dialog in which you can select the parameters/settings you want to save from the drop-down list. Alternatively you can enter the password for the WPA command. Entering the password and pressing ***OK*** will save the current valid parameter values of the controller as startup values to the non-volatile memory of the controller.



Change command level (CCL)

Opens a dialog in which you can select a command level and enter the corresponding password. Changing some parameters and running some commands may require the command control level (CCL) 1. The password for command control level 1 is "advanced". Note that command levels higher than 1 are reserved for PI service personnel.

Show system parameters

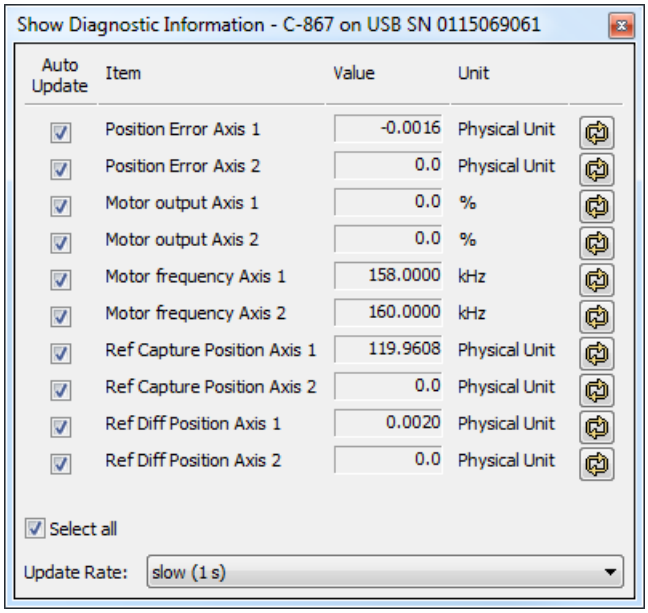
Opens a window (p. 166) in which the controller's system parameters are displayed and can be changed.

Show diagnostic information

Available if supported by the controller.

Opens a window in which diagnostic information for the controller can be displayed. The items displayed correspond to the measurands the controller supports for the `DIA?` command.

Click on an item's checkmark to periodically update the diagnostic information, or click ***Select all*** to check all items. The update rate can be set via the ***Update Rate*** drop-down list. Via the button in each line, the items' data can be updated manually.



Version info...

Displays version info of the software used for the associated controller.

Configure interface...

Available if the controller has at least one configurable communication interface.

Opens the ***Configure Interface*** window (p. 151) where you can modify the current and the default communication settings for the controller.

Log window...

Opens a controller-specific log window (p. 168). In this window, you can monitor the commands sent to the controller via the controls of PIMikroMove.

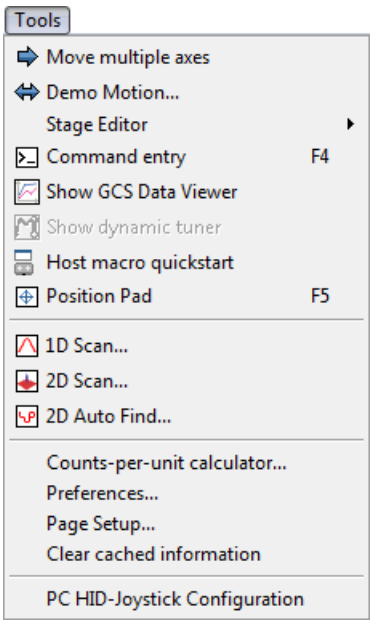
Close connection

Closes the connection to the controller or controller network. Once the connection is closed, the associated controller menu is no longer displayed in the menu bar.

2.1.3 Tools Menu

The ***Tools*** menu gives access to special windows which in principle are available for all controllers.

Note that windows which are in any way controller-specific are accessible from the corresponding controller menu, see "Controller Specific Windows" (p. 103).



Move multiple axes

Available if at least one of the connected axes is referenced and has servo on.

Opens the dialog ***Move multiple axes*** (p. 61) for initiating a synchronous move of more than one axis.

Demo Motion…

Available if at least one of the connected axes is referenced and has servo on.

Opens the window ***Demo Motion*** (p. 62) in which you can perform test moves for the connected axes.

Stage Editor

Available if the PIStageEditor or PIStages3Editor is installed on the PC.

Opens an editor in which you can inspect the parameters of PI stages contained in a stage database. If multiple stage databases are installed on the PC, a submenu is opened that contains the types of installed stage databases. Select an option to open the corresponding stage editor. See "Stage Editor (p. 63)" for details.

INFORMATION

Depending on the controller, you can also inspect parameters on the tab cards of the main window (p. 9), in the single-axis window (p. 97) or using the ***Device Parameter Configuration*** (p. 144).

Command Entry

Opens the terminal window ***Command entry*** (p. 64) for the communication with a connected controller: Commands can be entered and responses seen. The window can be locked in order to get exclusive access to the controller communication. While the window is in the locked state, no updates of other PIMikroMove windows are allowed.

Show GCS Data Viewer

Opens a file selection dialog in which you can select a GCS array file (format: .dat). The data of the selected file is then displayed in the window ***GCS Data Viewer*** (p. 67).

Show dynamic tuner

Available for piezo controllers.

Opens the window ***Piezo Dynamic Tuner*** (p. 77) in which you can modify and optimize the control parameters of the piezo servo controller, e.g. in case the load has been changed.

Host macro quickstart

Displays the window ***Host Macro Quick Start*** (p. 54) that gives you quick access to host macros. You can select and start a host macro without the need to switch to the ***Host macros*** tab card. Furthermore, you can assign a short-cut button to a host macro so that this macro can be started by simply clicking the button.

By default, the ***Host Macro Quick Start*** window is docked to the upper border of the main window.

Position Pad

Displays the ***Position Pad*** (p. 84), in which two axes can be controlled using the mouse.

The ***Position Pad*** window can be docked to the bottom border of the main window by dragging it with the left mouse button pressed.

1D Scan...

Opens the window ***Scan 1D*** (p. 86) where you can start a single-axis scan motion during which an input source is measured.

Do not confuse this PIMikroMove functionality with the embedded fast scanning offered by some controllers.

2D Scan...

Opens the window ***Scan 2D*** (p. 88) with the ***Scan*** tab card (p. 90) on top where you can start biaxial scan motion during which an input source is measured.

Do not confuse this PIMikroMove functionality with the embedded fast scanning offered by some controllers.

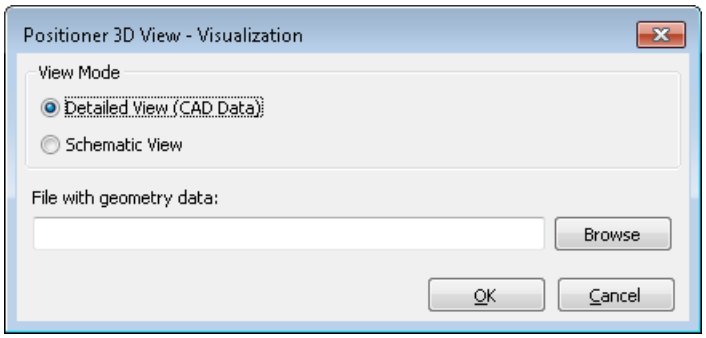
2D Auto Find...

Opens the ***Scan 2D*** window with the ***Auto Find*** tab card (p. 92) on top where you can start a scan procedure which tries to find the maximum of an intensity signal by modifying the positions of two axes. The measured intensity signal can be, for example, a selected analog input or the position of a selected axis.

Do not confuse this PIMikroMove functionality with the embedded fast scanning offered by some controllers.

Show Positioner 3D View - offline

Displays the ***Positioner 3D View*** (p. 31) tab card with an offline visualization of a positioner's position data. The function opens the ***Positioner 3D View - Visualization*** dialog where you can load a DAT file which contains data for the visualization of the connected positioner.



Counts-per-unit calculator...

Opens the window ***Counts per unit calculator*** (p. 95) in which you can calculate the values for numerator and denominator of the counts-per-physical-units factor (parameters 0xE and 0xF) for your stage. For stages with incremental sensors, the counts-per-physical-unit factor determines the unit of length to be used for all closed-loop motion commands (the controllers internally use counts for motion command processing).

Preferences...

Opens the program settings window (p. 101) in which you can configure the behavior of the program's user interface.

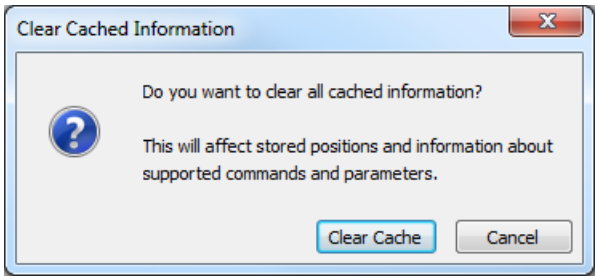
Some program settings that can be activated via dialogs cannot be deactivated in the same way, such as turning off future appearances of confirmation dialogs. These settings can always be changed in the ***Preferences***... window.

Page Setup...

Allows setting parameters which are used when printing from the ***Command entry*** window and the ***Controller macros*** tab card.

Clear cached information

This function clears the information cached by the program, e.g. stored positions.

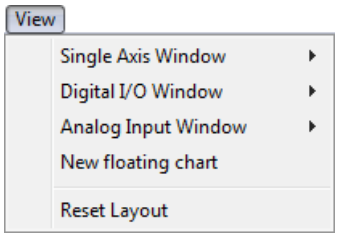


PC HID-Joystick Configuration

Displays the ***Configure PC HID Control*** (p. 93) window where you can configure, test and calibrate HIDs (e.g. joysticks, game pads) connected to the PC.

2.1.4 View Menu

Using the items of the ***View*** menu, you can open separate windows for the individual axes, digital I/O lines and analog input lines of all connected controllers, e.g. for monitoring purposes. Depending on the controller(s) connected to PIMikroMove, not all items of the ***View*** menu are always available.



INFORMATION

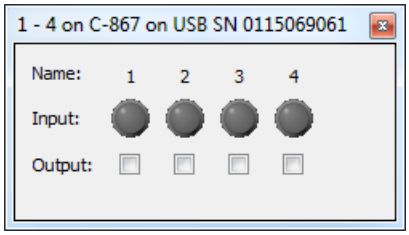
Single Axis, ***Digital I/O***, and ***Analog Input*** windows can be docked to the left- or right-hand border of the main window by dragging them with the left mouse button pressed.

Single-Axis Window

Opens a window in which data about a single axis is displayed and can be modified. The controller is interrogated at regular intervals (unless ***Command entry*** is in the locked state) to update the display. See "Single Axis Window" (p. 97) for details.

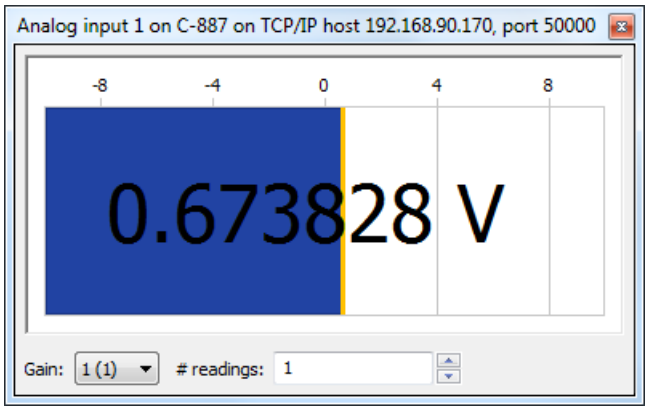
Digital I/O Window

Shows a window in which the state of the digital input lines can be monitored and that of the digital output lines changed. The controller is interrogated at regular intervals (unless ***Command entry*** is in the locked state) to update the display.



Analog Input Window

Shows a window in which the state of the analog input lines can be monitored. The controller is interrogated at regular intervals (unless ***Command entry*** is in the locked state) to update the display. Minimum and maximum values are represented by orange bars. If you move the cursor with the mouse into the window, the numerical values are displayed. To reset them, click the right mouse button in the window, and from the context menu select ***Reset minimum and maximum value***. The display range can be changed by the context menu option ***Define displayed range***.



If supported by the controller, you can also configure the analog input channel:

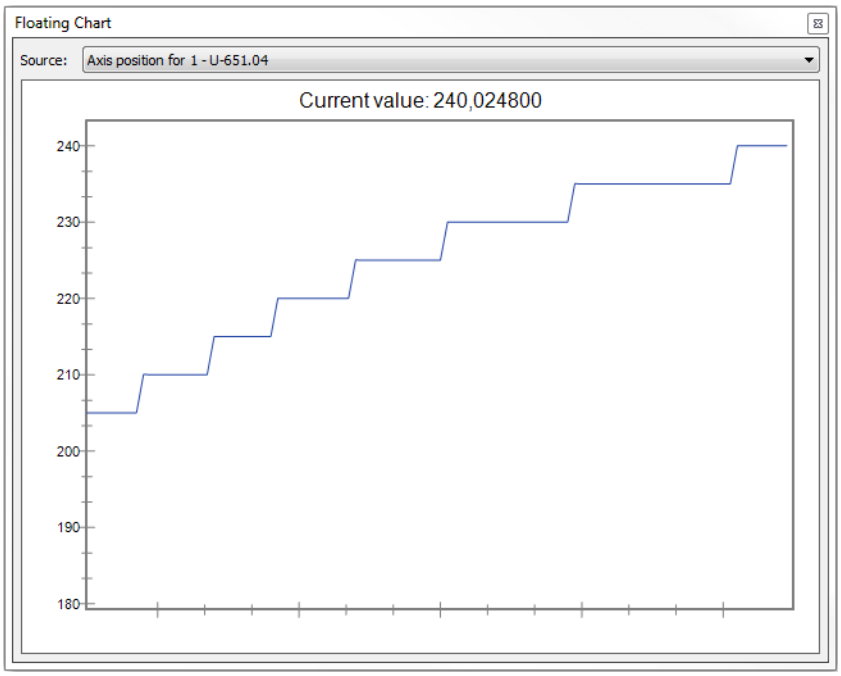
- In the ***Gain*** field, select the gain value to be used for optical input.
- In the ***# readings*** field, determine the number of readout values of the analog input that are averaged. The greater the noise of the analog input signal, the higher the number of readout values of the analog signal should be set.

New floating chart

Opens the window ***Floating Chart*** in which you can start the continuous display of data for the connected axes: Open the drop-down list ***Source:*** via the arrow button and select the data to be

displayed in the chart. The displayed data is continuously updated (unless ***Command entry*** is in the locked state) by PIMikroMove. Please note that other processes running on the PC may slow down or temporarily halt the continuous update of the displayed data.

The continuous display of data does not stop until you select a different display option or close the window. Display options are, for example, axis positions, target positions, or the response to a GCS command you can enter.



Continuous display of data related to a connected axis

Reset Layout

In case of a changed arrangement or order of the tabs in the program window, this function resets the program window to the default layout.

2.1.5 Help Menu

Help...

Opens the online help for the program.

FAQ, Tutorials...

Opens the task-oriented help for PIMikroMove.

Open PI's website

Opens the PI website (http://www.pi.ws).

Start the PI Update Finder

Available if the PIUpdateFinder is installed on your PC.

Starts the PIUpdateFinder installed on your PC. Using the PIUpdateFinder you can find and download updates for the PI software installed on your PC (e.g. PIMikroMove, GCS DLL and PI stage databases). For details, see "Installing Updates" (p. 3) and the Technical Notes which are provided with the PIUpdateFinder.

Download the PI Update Finder

Available if the PIUpdateFinder is not installed on your PC.

Opens the PI website at the download area where the latest version of the PIUpdateFinder is available for download.

Show version information...

Opens a window with version information about PIMikroMove and the installed controller software.

About PIMikroMove...

Displays PIMikroMove version information.

2.2 Toolbar

The availability of the individual toolbar icons depends on the connected controller(s) and the axis states.

New connection
Opens the ***Start up Controller*** window (p. 169) to connect another controller or controller network.

Move multiple axes
Opens the dialog ***Move multiple axes*** (p. 61) for initiating a synchronous move of more than one axis.

Start demo motion
Opens the window ***Demo Motion*** (p. 62) in which you can perform test moves for the connected axes.

Show PI Stage Editor
Opens an editor in which you can inspect the parameters of PI stages contained in a stage database. If multiple stage databases are installed on the PC, a submenu is opened that contains the types of installed stage databases. Select an option to open the corresponding stage editor. See "Stage Editor (p. 63)" for details.
Depending on the controller, you can also inspect parameters on the tab cards of the main window (p. 9), in the single-axis window (p. 97) or using the ***Device Parameter Configuration*** (p. 144).

Show position pad
Displays the ***Position Pad*** (p. 84), in which two axes can be controlled using the mouse.

Show command entry
Opens the terminal window ***Command entry*** (p. 64) for the communication with a connected controller: Commands can be entered and responses seen. The window can be locked in order to get exclusive access to the controller communication. While the window is in the locked state, no updates of other PIMikroMove windows are allowed.

Show GCS Data Viewer
Opens a file selection dialog in which you can select a GCS array file (format: .dat). The data of the selected file are then displayed in the window ***GCS Data Viewer*** (p. 67).

Show dynamic tuner
Opens the window ***Piezo Dynamic Tuner*** (p. 77) in which you can modify and optimize the control parameters of the piezo servo controller, e.g. in case the load has been changed.

Show host macro quickstart
Displays the window ***Host Macro Quick Start*** (p. 54) that gives you quick access to host macros. You can select and start a host macro without the need to switch to the ***Host macros*** tab card. Furthermore, you can assign a short-cut button to a host macro so that this macro can be started by simply clicking the button.

Create new host macro with current position as target
Opens the window ***Create New Host Macro with Current Position*** (p. 55) where you can create a new host macro containing move command(s) with the current axis position(s) as target(s).

Show 1D Scan dialog
Opens the window ***Scan 1D*** (p. 86) where you can start a single-axis scan motion during which an input source is measured.

Show 2D Scan dialog
Opens the window ***Scan 2D*** (p. 88) with the ***Scan*** tab card (p. 90) on top where you can start biaxial scan motion during which an input source is measured.

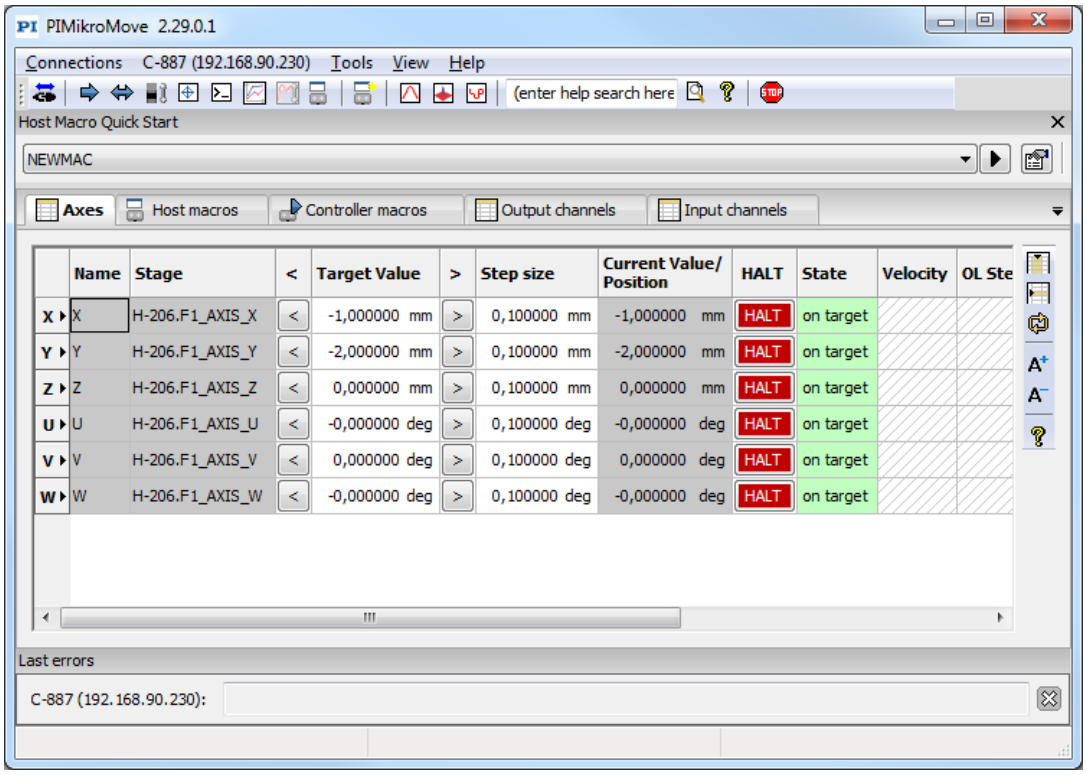
Show 2D AutoFind dialog
Opens the ***Scan 2D*** window with the ***Auto Find*** tab card (p. 92) on top where you can start a scan procedure which tries to find the maximum of an intensity signal by modifying the positions of two axes. The measured intensity signal can be, for example, a selected analog input or the position of a selected axis.

Search help
The online help for PIMikroMove is searched for a search term. Enter the search term in the field to the left, then start the search with the Enter key or via this button. The online help will be opened with the search result.

Open help
Opens the online help for the program.

Stop all
Sends an immediate stop command (ASCII #24) to all axes connected to PIMikroMove.

2.3 Table Tab Cards



If supported by the connected controller(s), separate tab cards with tables for axes, piezo channels, drive channels, sensor channels and PiezoWalk® channels are provided in the main window.

On a table tab card, you can specify the columns to be displayed and change the order of the rows (axes or channels). You can also type new values into the white fields.

INFORMATION

The parameter values in the fields are hardware-specific. Wrong values may lead to improper operation or damage of your hardware.

If the target or the open-loop value for an axis is changed, the system will move immediately. PiezoWalk® channels will move when you enter a certain number of steps in the ***Open-Loop Steps*** field and click the arrow buttons beside this field.

Controllers will keep the changed settings until the next power on-off cycle. With some controllers, it is possible to store the changed settings with the ***Save parameters as User Stage Type*** item or the ***Save parameters to non-volatile memory*** item of their controller menu (p. 12).

On the right-hand side of the table tab cards there is a small toolbar with functions to show and hide columns (p. 26) and sort rows:

Select columns to be displayed
Opens the dialog ***Select Columns*** (p. 26) in which you can define which data to display on the tab card.

Sort rows
Opens the dialog ***Sort Rows*** (p. 29) in which you can change the order of the displayed rows.

Refresh
Reloads the values for all displayed fields. Use this function to have the current active values displayed.

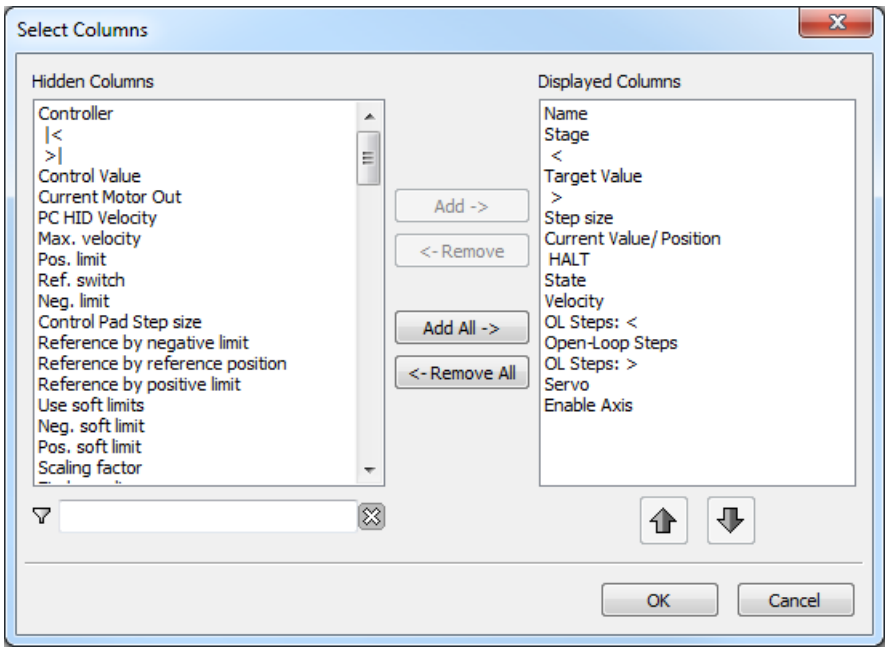
Increase font size
Increase the font size used in the table.

Decrease font size
Decrease the font size used in the table.

Show help
Opens the online help for the program.

2.3.1 Select Columns

To select the columns to be displayed on the table cards, open the dialog shown below. This can be done from the table cards either by right-clicking the header of the table or by clicking the corresponding button on the right-hand side of the table.



Here you can select which axis- or channel-related fields are to be displayed. You can also change the order of the displayed columns with the arrow buttons (top is left, bottom is right). Fields depending on motion are updated when the window is active.

The available fields depend in part on the connected controller. For a description of general fields provided by PIMikroMove see "Fields Displayed" (p. 27). For details regarding controller-specific fields (parameters) see the controller's user manual or the GCS DLL manual.

To filter the list of available parameters for certain entries, you can type a part of a parameter name into the input field below the ***Hidden Columns*** pane. The pane will then list only the parameters whose names contain the input you typed. To delete the input, press the ***Clear filter*** text button next to the input field.

2.3.2 Fields Displayed

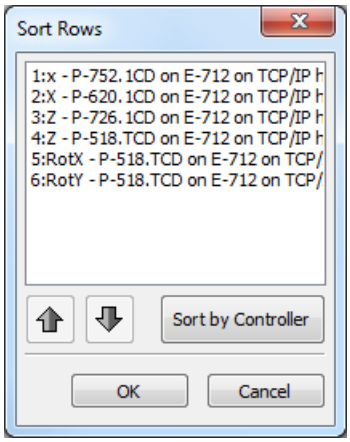
The following fields are provided by PIMikroMove. They can be displayed, e.g. on the ***Axes*** tab card, if the connected hardware supports the corresponding feature.

Header	**Displayed data**
Name	Axis / channel designator
Stage	Stage type
Controller	Controller the axis / channel is connected to
Target Value	Current target position
|<	Move to minimum position Rotary stages: Motion caused by this button is limited to the next complete rotation.
<	Perform step in negative direction
Step size	Size used for relative steps
>	Perform step in positive direction
>|	Move to maximum position Rotary stages: Motion caused by this button is limited to the next complete rotation.
Current Value / Position	Current position of the axis
Control Value	Currently valid control value of the axis
Current Motor Out	Open-loop control value for the axis. The open-loop control value corresponds to the commanded position for the axis.
Voltage	Current output voltage (for piezo channels)
HALT	Click to stop the axis (controller supports HLT command)
STOP	Click to stop all axes connected to the same controller (controller does not support HLT command)
State	State of the axis, e.g. "moving", "on target", "servo off"
Min. value	Minimum position value
Max. value	Maximum position value
Velocity	Velocity to use during moves
Max. velocity	Limit for velocity

Header	Displayed data
Servo	Servo state indicated via a checkbox: checked = servo on, unchecked = servo off
Enable Axis	Motor enable state (motor for axis switched on/off)
Open-loop target position	Open-loop target position for axes driven by PiezoWalk® drives (OMA command)
Open-loop target value	Dimensionless open-loop value for axes driven by piezo actuators Depending on the controller settings it may correspond to a position or to a voltage.
Open-Loop Analog Driving	Voltage for open-loop analog driving (done by the shear piezos)
Open-Loop Velocity	Velocity to use during open-loop nanostepping motion, in step cycles per second
OL Steps: <	Perform open-loop steps in negative direction
Open-Loop Steps	Number of open-loop steps to go, floating point number
OL Steps: >	Perform open-loop steps in positive direction
Open-Loop Remaining Steps	Steps still to perform to reach the target position in open-loop mode
Pos. limit	Indicates whether the positive limit signal is active
Ref. switch	Indicates whether current position is above or below the reference point
Neg. limit	Indicates whether the negative limit signal is active
Reference by neg. lim.	Referencing is to be done using the negative limit switch: axis moves in negative direction until negative limit switch is tripped, then moves as small amount in positive direction until the limit condition is no longer fulfilled. Afterwards, the absolute position of the axis is set.
Reference by reference	Referencing is to be done using the reference switch: axis moves toward the reference switch until reference signal changes state; if state has changed from high to low, the process is repeated. Afterwards, the absolute position of the axis is set.
Reference by pos. lim.	Referencing is to be done using the positive limit switch: axis moves in positive direction until positive limit switch is tripped, then moves as small amount in negative direction until the limit condition is no longer fulfilled. Afterwards, the absolute position of the axis is set.
Find neg. lim.	Axis moves in negative direction until negative limit switch is tripped; state of axis is not changed.
Find reference	Axis moves toward the reference switch until reference signal changes state; if state has changed from high to low, the process is repeated; state of axis is not changed.
Find pos. lim.	Axis moves in positive direction until positive limit switch is tripped; state of axis is not changed.
Drift compensation	Drift compensation enabled/disabled

Header	Displayed data
Velocity control	Velocity control enabled/disabled
Online	Online state
Overflow	Overflow state of the axis / channel
A/D Value	Current input value from A/D converter (for sensor channels)
Normalized Value	Normalized value (for sensor channels)
Step Amplitude	Step amplitude (for PiezoWalk® channels)
Relax	Relax piezos (for PiezoWalk® channels)

2.3.3 Sort Rows



To change the order of the rows displayed on a table tab card, open the ***Sort Rows*** dialog. This can be done from the tab card either by right-clicking the header of the table and select the function in from the context menu, or by clicking the corresponding button on the right-hand side of the table.

Sort by controller will sort the axes or channels first by controller and then alphabetically by name. The arrow buttons allow moving the selected axis or channel up or down in the list.

2.3.4 Axis Menu

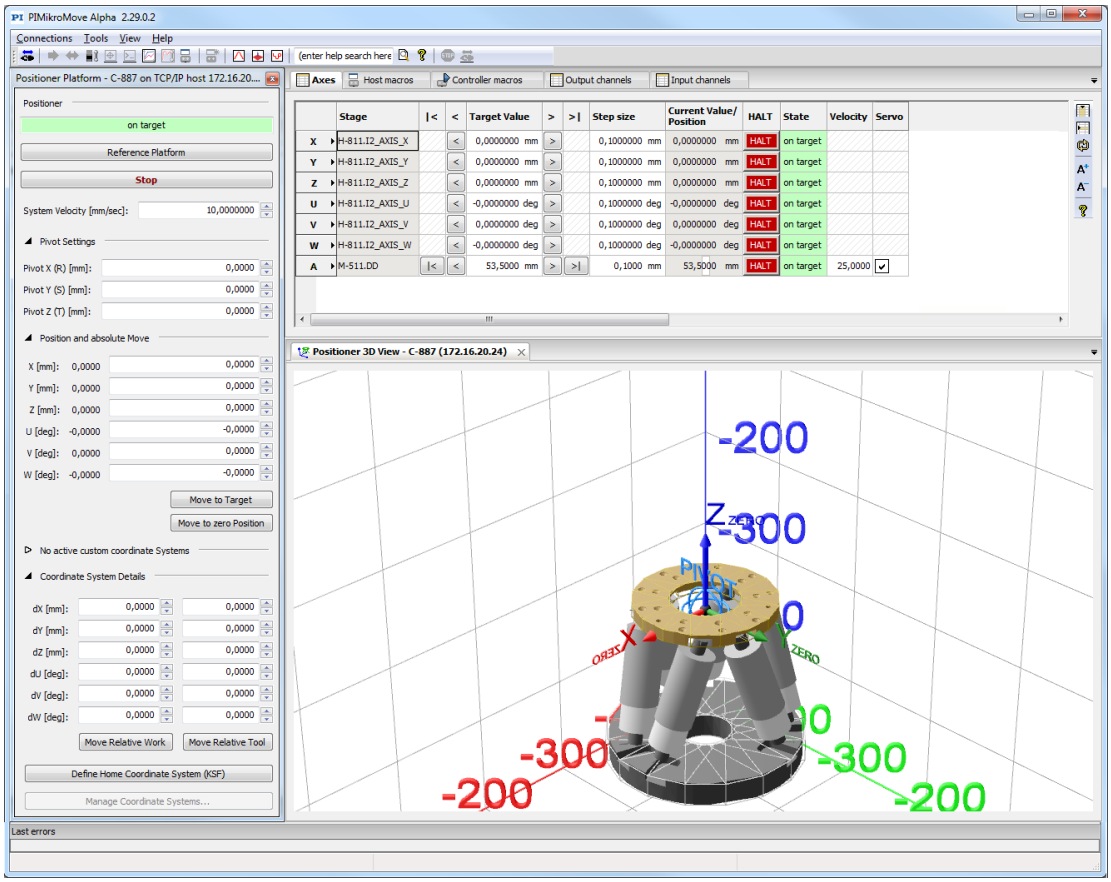
The ***Axis*** menu can be displayed from the ***Axes*** tab card by clicking the triangle in the first column of the axes grid, by right-clicking anywhere in the corresponding row or by right-clicking on any free area in the ***Single-Axis Window*** (p. 97).

The items of the ***Axis*** menu depend in part on the connected controller.

- ***Show/Hide Single Axis Window***
  Toggles the display of the corresponding single-axis window
- ***Show Expanded Single Axis Window***
  Displays the expanded view of the corresponding single-axis window

- ***Define Home Position***
  Makes the current physical position the home position of the axis. The current position is set to 0
- ***Move to Home Position***
  Moves the corresponding axis to its home position by setting the target to 0
- ***Create new host macro with current position***
  Only available if the axis has servo on (axes with incremental sensors must be referenced). Opens the window ***Create New Host Macro with Current Position*** (p. 55) where you can create a new host macro containing move command(s) with the current axis position(s) as target(s).
- ***Save current settings***
  Saves the current stage configuration and the corresponding motion parameter values of the axis to non-volatile memory of the controller and thus make them the new power-on defaults
- ***Restore settings***
  The stage configuration and the motion parameters which were last saved with ***Save current settings*** are loaded and become the currently active settings
- ***Save parameters as User Stage type...***
  The current settings of a stage can be stored as a new user-defined stage type in the stage database. In the dialog that is opened enter a name for the user stage entry in the stage database, and click ***OK*** to save the parameters. If an entry of the same name already exists in the database, its settings will be overwritten when the new ones are saved.
- ***Reload settings from stages database***
  The parameter values for the current stage type are reset to the values defined in the stage database. Note that this resets the referencing state of the axis and may switch the servo off.
- ***Initialize axis***
  Initializes the corresponding axis
- ***Clear Error***
  Shows current error code with explanation and, if possible, returns the axis or axes to a non-error state
- ***Start up axes...***
  Opens the ***Start up Controller*** window at the ***Start up axes*** (p. 175) step
- ***Change number of displayed decimal places...***
  Opens a dialog in which you can change the number of decimal places that are displayed for the values of the axis in the tab card

2.4 Positioner 3D View



For every parallel kinematics system (controller or simulation) connected to PIMikroMove, a ***Positioner 3D View*** tab card is provided in the main window.

The ***Positioner 3D View*** tab card visualizes the following:

- Current position of the parallel kinematics positioner
- Current location of the pivot point and coordinate systems
- Any objects which were defined for the environment of the parallel kinematics positioner in PIVeriMove

You can change the visualization on the ***Positioner 3D View*** tab card as follows:

- Rotate the view by dragging the mouse pointer across the tab card with the left mouse button pressed.
- Zoom in or out on the view by dragging the mouse pointer across the tab card with the right mouse button pressed.
- Load a DAT file with geometry data for the parallel kinematics positioner and select the visualization mode (detailed CAD view or schematic view) using ***Positioner 3D View > Change visualization...*** This item is available on the controller menu when the ***Positioner 3D View*** tab card is present in the main window. Loading a DAT file only affects the visualization on the ***Positioner 3D View*** tab card and not the actual motion of the parallel kinematics positioner.

To visualize motions of the parallel kinematics positioner, the display on the ***Positioner 3D View*** tab card is permanently updated. To start motion, using the ***Positioner Platform*** (p. 151) window is recommended since you can also change the pivot point coordinates and coordinate systems. By default, for every parallel kinematics system (controller or simulation) connected to PIMikroMove, a ***Positioner Platform*** window is docked in the main window.

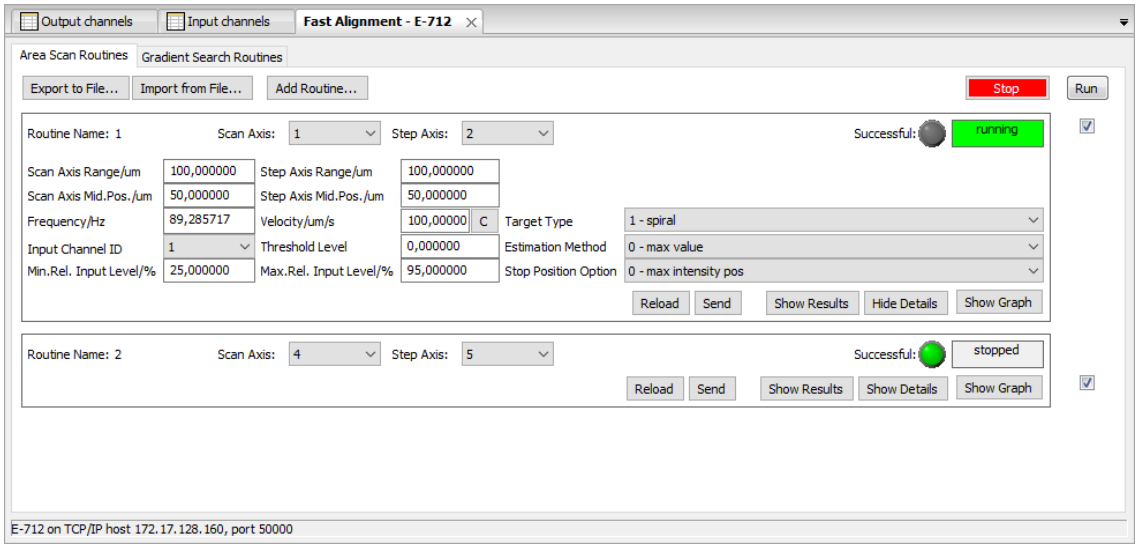
The display is also updated permanently during moves started, for example, from the ***Axes*** tab card (p. 25) in the main window, from ***Single-Axis*** windows (p. 97), from the ***Demo Motion*** window (p. 62) or from the ***Move multiple axes*** window (p. 61).

INFORMATION

The display on the ***Positioner 3D View*** tab card is not updated as long as the ***Command entry*** window (p. 64) is in the locked state.

2.5 Fast Alignment

The menu option ***<Controller name> > Show Fast Alignment Window*** opens the tab for the definition of fast alignment routines and the graphic display of their results.



There are individual tabs for different types of routines:

- ***Area Scan Routines***

  An area scan is a spiral or sinusoidal scan to find the position of the global intensity maximum of the measured signal.

- ***Gradient Search Routines***

  A gradient search is a circular scan with gradient formation to find the maximum intensity value of the measured signal.

Fast alignment routines can be exported to a text file via the button ***Export to File...*** and imported from a text file via ***Import from File...***. The button ***Add Routine...*** is for the definition of scan routines. Here, either new routines can be defined or existing routines can be cloned.

On the ***Gradient Search Routines*** tab you find an additional button, ***Define Coupling ...***. Use this button to couple gradient search routines. Coupled routines are not allowed to stop until all routines coupled to them are finished.

For the execution of a fast alignment routine, select the axes you want to use as

- ***Scan Axis***: Master axis for the scan routine.
- ***Step Axis***: Second axis for the scan routine. For single-axis routines set this value to 0 or to the value of the selected scan axis.

The current position of the selected scan axis can be set as the center position of the scan axis via the context menu of the ***Scan Axis Mid. Pos.*** field.

To display the detailed definition of a fast alignment routine click the button ***Show Details***. The button then changes to ***Hide Details*** with which you can switch off the detailed display of the routine definition.

Changed routine settings can be reset with the button ***Reload***. With the button ***Send***, you can send the currently loaded routine to the controller. If any routine settings were changed, but not yet sent to the controller, the program shows a corresponding message.

Clicking the button ***Show results*** displays a text field in which the scan results are shown, e.g., the measured values of intensity, maximum positions etc.

Click the button ***Show Graph*** to open a window in which the result of the fast alignment routine can be displayed in a 3D and 2D graph. In the ***2D*** window you can configure the data display in the graphics pane and perform operations with the data via toolbar buttons (p. 68).

To start fast alignment routines click the corresponding checkboxes and click the button ***Run***. If a selected routine contains any changes that were not yet sent to the controller, the program opens a dialog in which you can send the changes to the controller before running the routine.

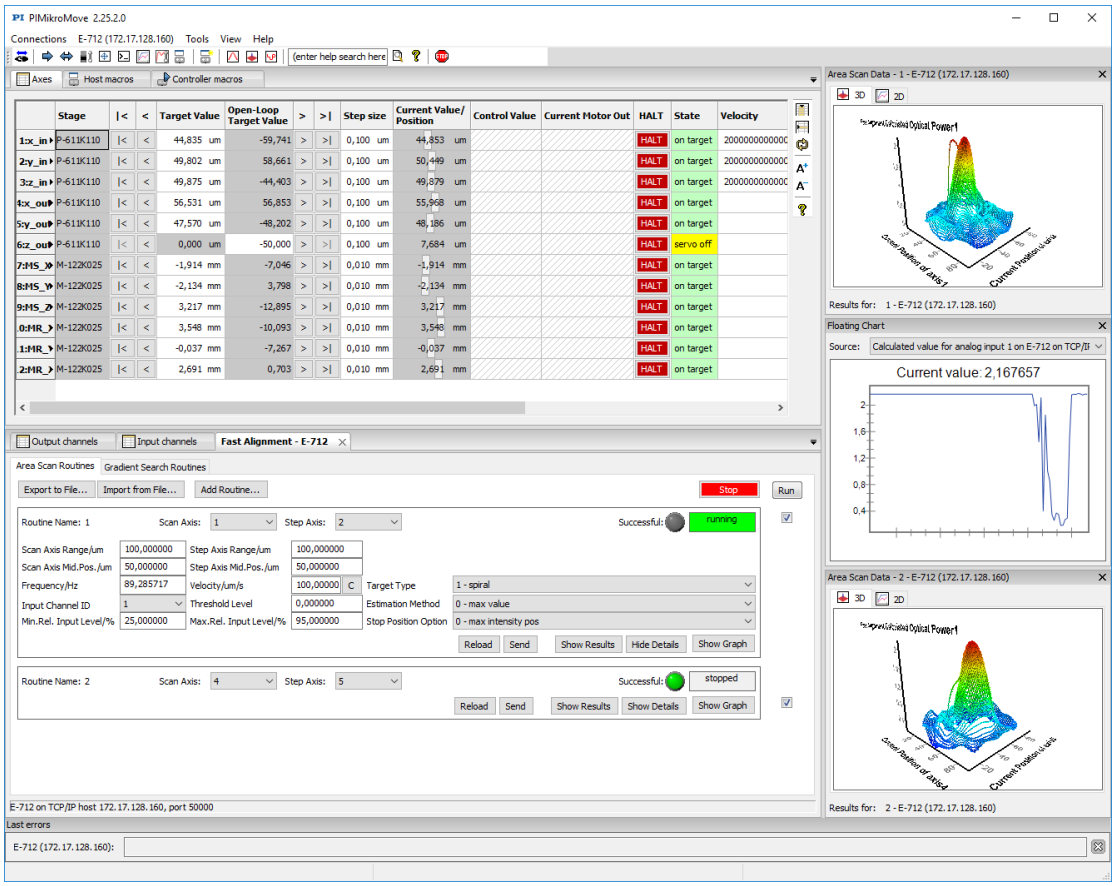
The status of the routine's execution is shown via two items:

- LED ***successful***: The LED turns green when the routine could be successfully completed.
- Status box: The box shows the current execution state of the routine, e.g., "running", "stopped".

After a successful scan, the result is shown in the corresponding window (if open).

A running routine can be stopped at any time with the button ***Stop***.

The picture shows an example of how tab cards and windows can be arranged in PIMikroMove for fast alignment operations:

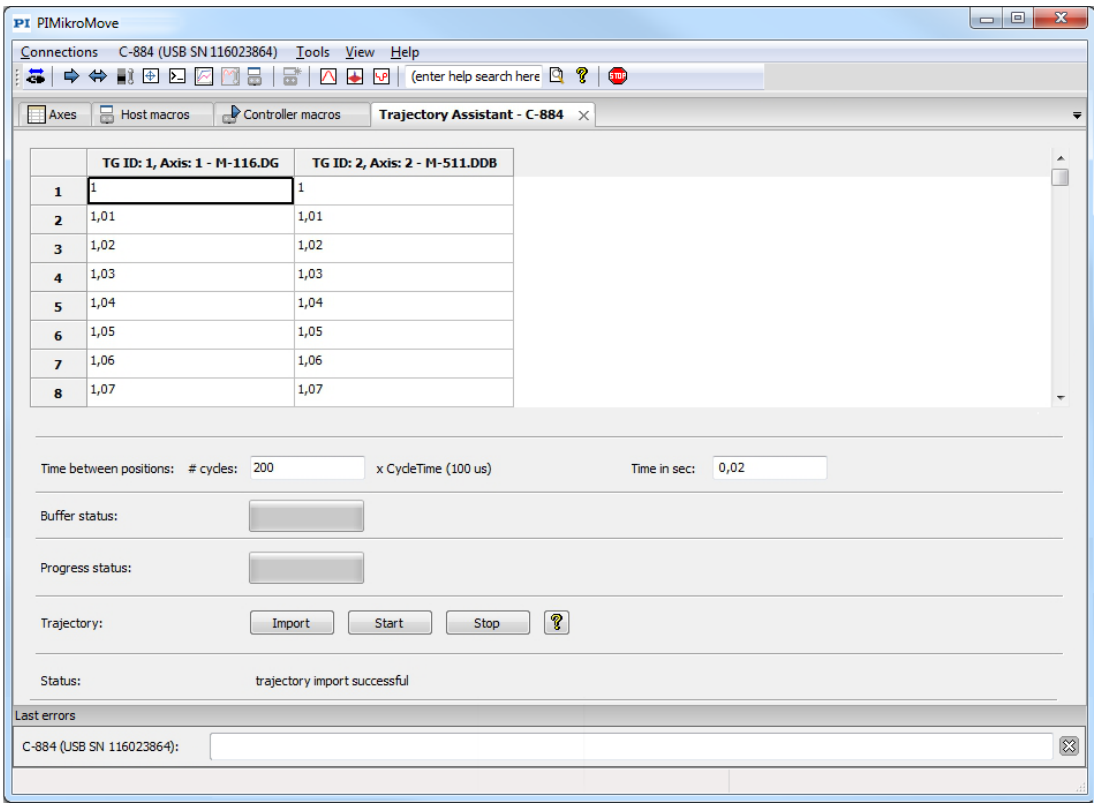


The ***Fast Alignment*** tab card is docked to the main window's bottom so both the ***Axes*** and ***Fast Alignment*** tab cards are active. The graph windows for the scan results are docked to the right, as is a floating chart window that shows the corresponding analog input values.

For further information on fast alignment routines see the user manual of your controller.

2.6 Trajectory Assistant

The menu option ***<Controller name> > Trajectory Assistant*** opens the tab for the definition and execution of trajectories.



Working with trajectories

Trajectories describe one or multi-dimensional motion paths (e.g., circles, sine curves) that are made up of externally calculated points (trajectory points) loaded into the controller. During the execution of a trajectory the trajectory points are travelled according to a specified chronological interval.

NOTICE

Execution of trajectories!

The controller does **not** calculate a dynamics profile during the execution of a trajectory. After the last trajectory point has been reached, the motion of the axis is abruptly stopped. This holds true for the proper completion of trajectories as well as for their cancellation (e. g., by a stop command or error). Acceleration / deceleration, velocity, and steadiness of the motion therefore depend on the following factors during trajectory execution:

- Values of the trajectory points
- Timing for the trajectories

The execution of an unsuitable trajectory can cause the stage to oscillate or the motion to abruptly stop. Oscillation or abrupt stopping can damage the stage and/or the load affixed to

it.

➢ Therefore, observe the following when working with trajectories:

- The path that is specified by the trajectory points must be continuously differentiable at least twice.
- During the execution of the trajectory, the maximum permissible velocity and acceleration of the axis must **not** be exceeded.
- During the execution of the trajectory, an abrupt stop must **not** damage the load on the stage.

Defining trajectories

For the definition of trajectory points there are two possibilities: You can either import trajectory points from a csv file or manually enter the values in the tab.

Trajectory points are given as floating point numbers.

A **csv file** for the import of trajectory points must meet the following requirements:

✓ The first line contains the trajectory IDs.

✓ The characters used as delimiter and decimal separator are either a) or b). Do not mix characters!

a) Delimiter: semicolon (;) - decimal separator: comma (,) - e. g.: 1,0394; 2,39439

b) Delimiter: comma (,) - decimal separator: period (.) - e. g.: 1.0394, 2.39439

The ***Trajectory Assistant - <controller name>*** tab consists of two sections:

- Table of trajectory points for the connected axes
- Functions for trajectories

Table of trajectory points

For each axis connected to the controller, a column for trajectory points is displayed.

	TG ID: 1, Axis: 1 - M-116.DG	TG ID: 2, Axis: 2 - M-511.DDB
1	1	1
2	1,01	1,01
3	1,02	1,02
4	1,03	1,03
5	1,04	1,04
6	1,05	1,05
7	1,06	1,06
8	1,07	1,07

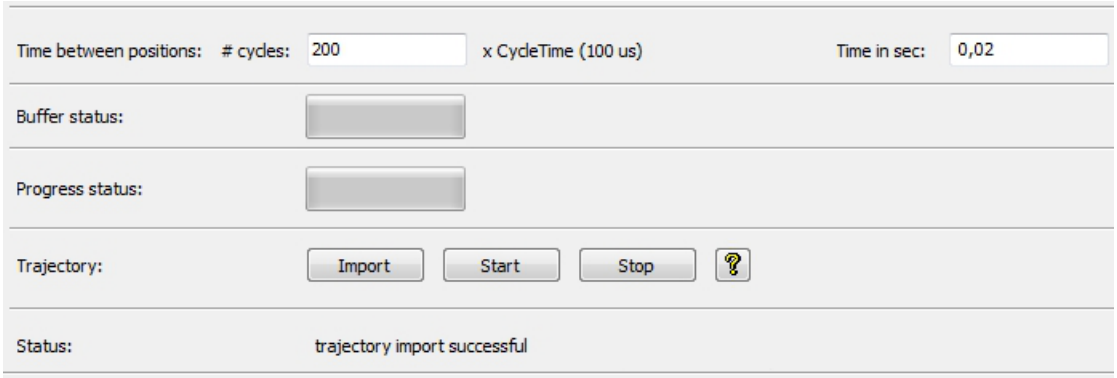
Each column header shows the following data: TG ID: #, Axis: # - Stage name

- ***TG ID: #***: # = ID of the trajectory
  The allocation of trajectories to the axes of the controller is fixed: Trajectory 1 to axis 1, trajectory 2 to axis 2, etc.

- ***Axis: #***: # = ID of the controller's motor connector to which the axis is connected
- ***Stage name***: Name of the stage or stage axis connected to axis #

Functions for trajectories

This section contains functions to set the trajectory timing, to import files with trajectory points, and to start and stop the execution of trajectories.



Time between positions
Via this option the trajectory timing is set, i.e., the chronological interval according to which the trajectory points are travelled during the execution of a trajectory. The timing is given in ***# cycles*** (number of servo cycles). The specified value is valid for all trajectories.

Time in sec
This field shows the trajectory timing in seconds. The value is automatically calculated from the specified number of servo cycles (***# cycles***) and the controller's servo cycle (***x CycleTime (<servo cycle of the controller>)***).

Buffer status
The status bar shows the progress of the continuous process of loading and unloading trajectory points to and from the trajectory buffer of the controller.

Progress status
The status bar shows the progress of the execution of the trajectories.

Trajectory
Operations can be performed with the following buttons:

- ***Import***
  A csv file containing trajectory points can be imported. When importing a csv file make sure the requirements mentioned above ("Defining Trajectories") are fulfilled.
- ***Start***
  This button starts trajectory execution. When the execution is started, all the connected axes for which trajectory points are defined move along the paths defined by the points in their respective columns.
- ***Stop***
  This button stops trajectory execution.
- 
  With the help button this help page is called.

Status
The current status message is displayed. If an error occurs it is displayed in the ***Last errors*** section.

2.7 Host Macros

Host macros allow you to store a sequence of commands in a text file for later execution by a connected controller. Basics of host macro usage are given in "Controls on Host Macro Tab Card" (p. 39) and "Controls in ***Run host macro*** Window" (p. 42).

Host macros can be useful, for example, in the following cases:

- Repeat specific actions:
  Using host macros, you can, for example, store a specific action done with PIMikroMove. To obtain the corresponding command sequence, open a log window for the controller of interest via the ***Log window… (p. 168)*** item on the controller's menu. In the log window, you can monitor the commands which are sent to the controller when you use the controls of PIMikroMove. In most cases it will be sufficient to copy the complete command sequence from the log window and save it as host macro. Example: Waveforms defined in the PI Wave Generator Tool (p. 103) for the controller's wave tables are lost when the controller is powered down or rebooted. Using a host macro you can store them permanently.
- Move to certain positions:
  Should it be necessary to recall certain positions, you can use the ***Create new host macro with current position…*** functionality of PIMikroMove. Via the appropriate toolbar button or axis menu item, you can create a new host macro which simply contains a move command with the current position as target. Furthermore, you can assign a separate button to this macro for easy access from the PIMikroMove main window. See "Create New Host Macro with Current Positions" (p. 55) and "Host Macro Quick Start" (p. 54) for details
- Develop controller macros:
  If the controller is able to store sequences of commands (controller macros), it is recommended to develop and test them as host macros firstly. Once the host macro is running successfully, you can load it to the controller as controller macro. This way you can use, for example, the ***Run host macro…*** window to monitor macro execution and variable values while developing the macro. Furthermore, the macro will be written to the controller's non-volatile memory only once (when finished) and not every time you change the macro. See "Controls in Run host macro Window" (p. 42) and "Controller Macros" (p. 57) for details.

INFORMATION

Host macros are interpreted by the host PC and PIMikroMove. This can be done for any controller even if the controller itself cannot store macros. If the controller supports macros, it may be faster to use them. (See also "What is the difference between controller macros and host macros?" (p. 188)). The timing of host macro operation is influenced by the time resolution of the operating system on the host PC (no real time operation).

There are special commands for usage in host macros that are interpreted by PIMikroMove directly: WAC (p. 43), MEX (p. 44), JRC (p. 44), BREAK (p. 45), MAC NSTART (p. 45), CMAC (p. 45), LOOP (p. 45), PRINT/MESSAGE/MESSAGE_CANCEL (p. 45), VAR (p. 47), CVAR (p. 47), VAR? (p. 47), CVAR? (p. 48), ADD (p. 48), CADD (p. 48), CPY (p. 48), CCPY (p. 48), UNLOAD (p. 49).

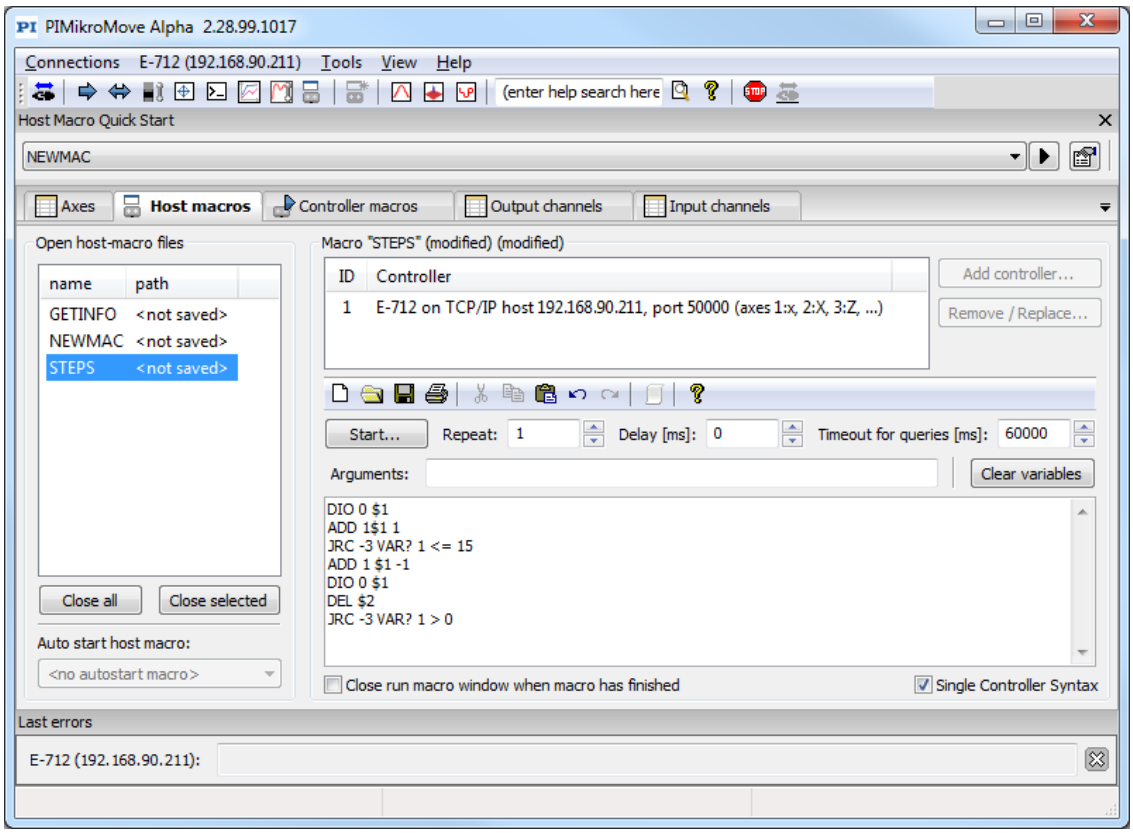
Every other command is sent as-is with no syntax check to the controller when a host macro is run. After each line sent to the controller, PIMikroMove checks the state of the controller by sending #7 (query the busy state). If the controller indicates that it is "busy", PIMikroMove will wait before sending the next line of the host macro.

Variables are provided for more flexibility in host macro programming. See "Variables" (p. 49) for details.

Macro command sequences are given for the following examples:

- Wait and Trigger (p. 50)
- Trigger at Specified Position (p. 51)
- X-Y-Scan with Different Macros (p. 51)
- X-Y-Scan with Loops (p. 52)
- Pushbutton Control (p. 53)

2.7.1 Controls on Host Macro Tab Card



The ***Host macros*** tab card gives access to all connected controllers—multiple controllers can be accessed with a single host macro.

List of available host macros

The available host macro files are listed on the left-hand side of the ***Host macros*** card. To add a macro existing on the host PC to the list, use the ***Open and show another macro from text file*** button in the icon bar of the card. The host macro selected in the list can be edited, executed or saved, and all controllers affected by this macro are shown in the top right window of the card. The macro list also shows the directory in which the host macro file is located on the host PC. If the entry is <not saved>, the host macro was not saved to the host PC yet.

Host macro execution

With the ***Start…*** button in the center of the ***Host macros*** card, you can start the execution of the currently selected host macro. Starting a macro opens the window ***Run host macro <macro name>*** (p. 42) in which you can monitor the processing of the macro's commands.

You can also specify that the macro command sequence will be executed repeatedly a certain number of times when the macro is run (***Repeat***).

A fixed execution delay can also be specified between the command lines in the ***Delay [ms]*** field: After #7 has indicated that the controller is not busy anymore, PIMikroMove will wait that amount of time between sending each line. (Note: On Windows systems, the time resolution is about 10 ms, so you will notice no difference between a delay of 5 and 10 ms!)

In the ***Timeout for queries [ms]*** field you can specify how long PIMikroMove has to wait for long-lasting macro commands to be processed.

INFORMATION

Controllers with GCS syntax version 1.0 may send a resonse after a long-lasting command has finished (e.g. after scanning procedures started with FSA for F-206 Hexapod systems or referencing procedures started with MNL, MPL and REF for C-863.10, C-663.10 and C-843 motor controllers). This response has to be "catched" by the host macro to avoid failure in the macro execution. To catch the response, use such long-lasting commands in host macros in combination with a command which checks conditions for further macro execution (e.g. MEX, JRC or WAC). You can specify the timeout value using the ***Timeout for queries [ms]*** field.

Example 1 simply "ignores" the response: JRC 1 MNL A = 1 will cause the macro execution to continue with the next line (jump one line ahead) if MNL returned 1. But JRC will also continue with the next line in any other case.

Example 2: You can "use" the response in combination with WAC, MEX and JRC. E.g. with MEX REF A = 0 the macro execution will be stopped if referencing of axis A failed.

In the ***Arguments*** field you can enter the values for the local variables used in the macro (see "Variables" (p. 49) for details). The values have to be separated by space characters. Macro execution may fail if the macro uses local variables but no entries are made in the Arguments field.

Using the ***Clear variables*** button, you can delete all variables in the host macro interpreter of PIMikroMove.

INFORMATION

The host macro selected in the macro list can only be executed if all controllers concerned by that host macro are connected to PIMikroMove via the interface(s) used during host macro creation. Controllers which are affected by the currently selected host macro but not connected or connected via an interface other than the one used during macro creation are highlighted in red in the controller list.

If the controller affected by the selected host macro is highlighted in red in the controller list, you can replace it by a currently connected controller. To do this, click on the highlighted controller and use the ***Remove / Replace…*** button and select the new controller from the list that opens (you can also click on the controller entry with the right mouse button to obtain the list).

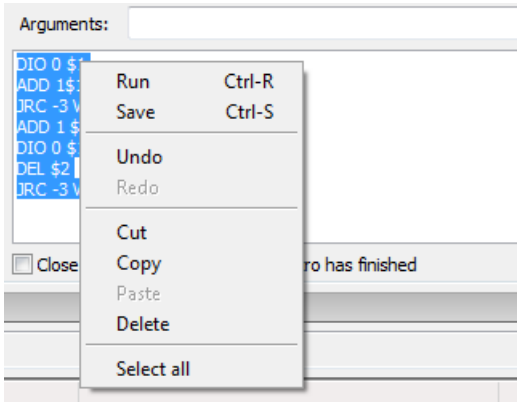
Autostart host macro

Below the macro list, you can select one of the macros to be executed automatically at the next AutoConnect start of PIMikroMove. Selection is only possible if the AutoConnect feature is activated (see "AutoConnect" (p. 178) for details). Note that the selected macro will only be executed if AutoConnect was successfully finished for all controllers affected by the macro.

Macro editor

The bottom right-hand window in the ***Host macros*** card has a macro editor, consisting of a text field and an icon bar with buttons for creating, opening and saving macro command sequences. Note that PIMikroMove provides a saving dialog if there are unsaved host macros when you quit the program.

In the editor, clicking the right mouse button opens a context menu that contains functions for text editing:



The following keyboard shortcuts are available:

- Ctrl+S saves the current macro
- Ctrl+R starts the current macro (without saving)

In the text field, you can either edit the currently selected macro or write a new host macro. The individual lines of the macro consist of a controller identifier (integer number from 0 to 255) and the command. If a line starts with ";", "'" or "//" it is ignored and so it can be used to comment macros.

With the controller identifier it is possible to specify the controller which should receive the command. Two special identifiers exist: 0 is the host PC and can only be used for commands that PIMikroMove executes (like DEL and MAC START/NSTART), and 255 stands for a broadcast to all connected controllers, i.e. the command is sent to each controller specified for this macro.

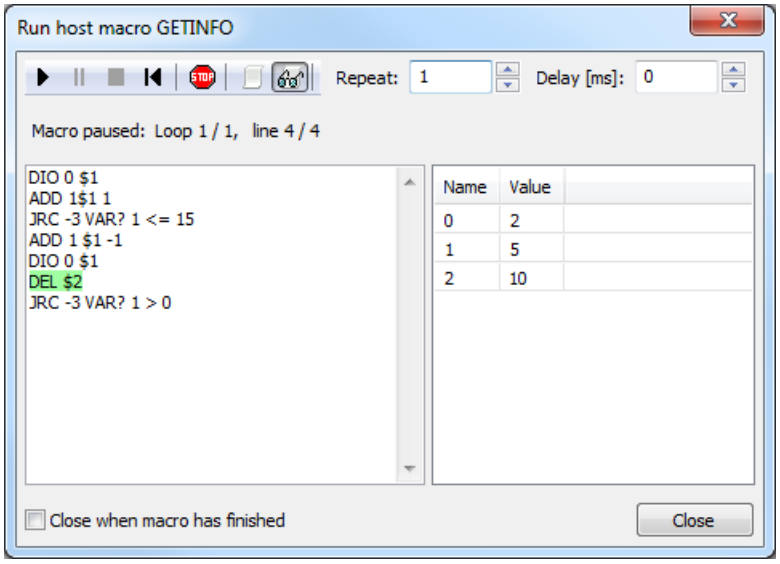
If only one controller is defined for a macro the ***Single Controller Syntax*** can be used, and macros can be written without explicitly specifying the controller ID for each line (see the checkbox on the bottom right of the card).

Version compatibility

In previous versions of ***PIMikroMove***, all commands in a given host macro had to address the same controller or controller network. If a macro stored with these versions is loaded and more than one controller is connected, the user is prompted to select a controller for this macro. For such old macros ***Single Controller Syntax*** is activated. Once loaded, additional controllers can be added.

Older versions of ***PIMikroMove*** might not be able to load macros saved with the current version.

2.7.2 Controls in Run host macro Window



When a host macro is run (***Start…*** button in the center of the ***Host macros*** tab card was pressed), the ***Run host macro*** window opens. There you can monitor macro execution and stop or restart it. This can be helpful, for example, during debugging.

Below the toolbar, the currently running macro is listed. It is either the macro started by the ***Start…*** button on the ***Host macros*** tab card, or another macro called by that macro. See the

title bar of the ***Run host macro*** window for the name of the running macro. In the list, the macro execution pointer is visualized by a small green bar.

The toolbar buttons in the window have the following functions:

- starts macro execution
- pauses macros execution, same function as BREAK command
- stops macro execution in PIMikroMove, i.e. the macro command sequence is no longer sent to the controller
- resets macro execution pointer and values of local variables when macro execution was paused with or BREAK. The macro will be restarted when is used afterwards.
- stops all, i.e. not only macro execution in PIMikroMove but also all other tasks on the controller (e.g. motion, scan algorithms, …)
- opens separate window with PRINT statements. During macro execution, PIMikroMove can output arbitrary text and the response to GCS query commands using the PRINT <”text”> <GCS query command. See "PRINT / MESSAGE / MESSAGE_CANCEL" (p. 45) for details.
- shows/hides a variable list where you can monitor the variable values. The list comprises all variables which are available for the currently running macro: the macro’s local variables and all global variables present in the host macro interpreter of PIMikroMove . You can delete the variables via the ***Clear variables*** button on the Host macros tab card.
- ***Repeat*** field: You can specify that the macro command sequence will be executed repeatedly a certain number of times when the macro is run.
- ***Delay [ms]*** field: A fixed execution delay can be specified between the command lines: PIMikroMove will then wait that amount of time between sending each line. (Note: On Windows systems, the time resolution is about 10 ms, so you will notice no difference between a delay of 5 and 10 ms!)

2.7.3 WAC <command?> <operator> <value>

WAit until a given **C**ondition of the following type is fulfilled: The condition involves comparing a specified value with the result of a command that returns one value on one line.

<command>	is the command in its full syntax. The command must be one that provides a response consisting of one single value on one line.
<operator>	is the comparison operator; supported are "=" "<=" "<" ">" ">=" "!=" **Important: There must be a blank space before and after the operator!**
<value>	is the value to be used on the right-hand side of the comparison.

For example: with WAC ONT? 1 = 1 you can wait for axis 1 to be on target. WAC DIO? A = 1 waits for the digital input A to be high. WAC POS? B > 4 waits for axis B to reach a position greater than 4.

2.7.4 MEX <command> <operator> <value>

Stop macro execution when a condition of the following type is fulfilled. The condition involves comparing a specified value with the result of a command that returns one value on one line.

When the macro interpreter accesses this command for execution the condition is checked. If it is true the macro is stopped, otherwise the macro execution continues with the next line. Later fulfillment of the condition will not in itself trigger any action.

<command>	is the command in its full syntax. The command must be one that provides a response consisting of one single value on one line.
<operator>	is the comparison operator; supported are "=" "<=" "<" ">" ">=" "!=" **Important: There must be a blank space before and after the operator!**
<value>	is the value to be used on the right-hand side of the comparison

For example: MEX DIO? A = 0 will exit the macro if the digital input A is 0.

All arguments of the MEX command can be omitted. In this case, macro execution is stopped by MEX without evaluating a condition.

2.7.5 JRC <jump> <command?> <operator> <value>

Jump relatively depending on a given condition of the following type: one given value is compared with a queried value according to a specified rule.

<jump>	is the size of the relative jump. -1 means the macro execution pointer goes to the previous line, 0 means the command is executed again, which is the same behavior as WAC. 1 goes to the next line, making the command unnecessary, and 2 jumps over the next command. Only jumps within the current macro are allowed.
<command?>	is one query command in its usual syntax. The response has to be a single value and not more.
<operator>	is the comparison operator; supported are "=" "<=" "<" ">" ">=" "!=" **Important: There must be a blank space before and after the operator!**
<value>	is the value to be used on the right-hand side of the comparison

Example: Using the following macro, you can stop motion of axis "1" using a stop-button connected to a digital input. The result of the POS? 1 query is being copied to the variable TARGET. Then this variable is used as second argument for the MOV command. Therefore, the stage stays where it just was. To clean up, TARGET is then defined empty with the VAR command which deletes the variable.

Write macro "stop":

```
MOV 1 20
JRC 2 DIO? 1 = 1
JRC -1 ONT? 1 = 0
CPY TARGET POS? 1
MOV 1 ${TARGET}
VAR TARGET
```

2.7.6 BREAK

Pauses macro execution, same as if is pressed in the ***Run host*** macro window. Macro execution can be continued with .

BREAK is helpful when debugging macros.

2.7.7 MAC [N]START <name>

With MAC START you can call another host macro. <name> specifies the filename of the text file with the macro content. With MAC NSTART you can specify how many times the called macro should be run. Thus you can implement loops by placing the commands of the loop inside a separate macro file and calling it with MAC NSTART (see sample below).

2.7.8 CMAC <options>

If a controller provides macro functionality, CMAC <options> gives complete access to it by calling the "MAC" command. CMAC is to be used with the syntax and options of the MAC command as described in the controller User manual. For example, "CMAC START DEMO" starts the macro DEMO stored on the controller by sending the "MAC START DEMO" command to the controller. With "CMAC BEG <macroname>" and "CMAC END" controller macros can be created at run time.

2.7.9 LOOP <nr> / ENDLOOP

With LOOP and ENDLOOP you can mark a section in the macro as loop code. The number of repetitions must be specified with the argument to LOOP. Loops can be nested without restrictions but the LOOP and ENDLOOP statements must be in the same macro.

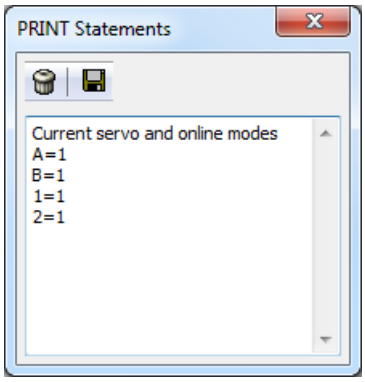
2.7.10 PRINT / MESSAGE / MESSAGE CANCEL

During macro execution, PIMikroMove can output arbitrary text and the response to GCS query commands. Availability of the output and further macro processing depend on the output command used which can be PRINT, MESSAGE or MESSAGE_CANCEL. The output can be set by the optional <"text"> and <GCS query command> arguments of the output commands. All characters between the quotation marks in the <"text"> argument will be output as is. The <GCS query command> argument gives one GCS query command in its usual syntax. This query command will be sent to the controller, and every response from the controller will be output.

Output command details:

- PRINT <"text"> <GCS query command>

  All output is written to the ***PRINT Statements window*** (see figure below). This window can be opened using the ***Show window with PRINT statements*** icon in the toolbar of the ***Host macros*** tab card. Macro execution is not affected by the PRINT command.



In the example shown in the figure above, the output is caused by the following PRINT commands (which are part of an executed host macro):

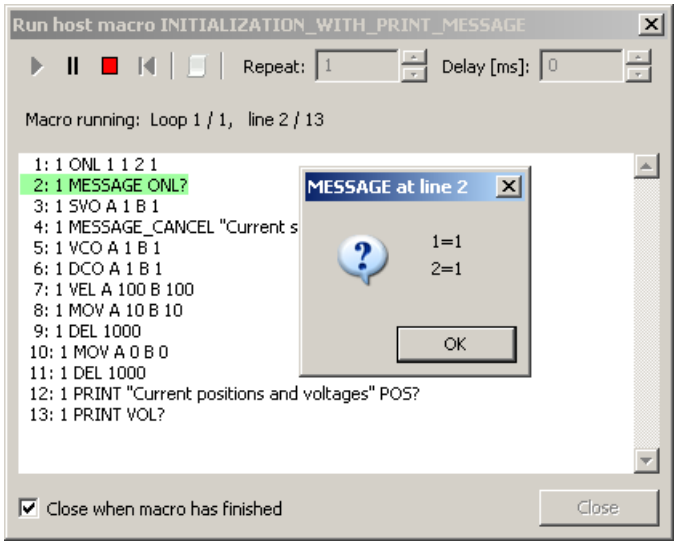
PRINT "Current servo and online modes" SVO?

PRINT ONL?

You can save the content of the ***PRINT Statements*** window to a text file on the host PC.

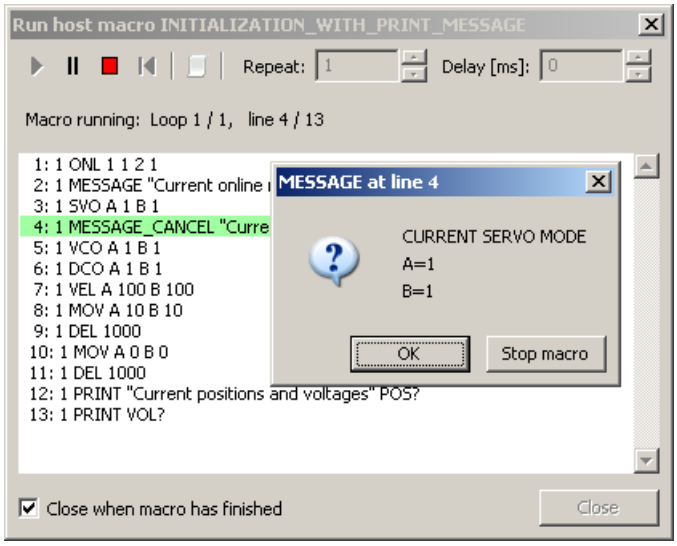
- MESSAGE <"text"> <GCS query command>

  All output is written to a message box which pops up when the macro executes the MESSAGE command. Macro execution is then paused until you press ***OK*** in the message box.



- MESSAGE_CANCEL <"text"> <GCS query command>

  All output is written to a message box which pops up when the macro executes the MESSAGE_CANCEL command. Macro execution is then paused until you press ***OK*** in the message box or it can be aborted by the ***Stop macro*** button.



2.7.11 VAR <variable> <string>

Set a variable stroed in PIMikroMove to a certain value. See "Variables" (p. 49) for conventions regarding variable names and values.

<variable>	is the name of the variable whose value is to be set
<string>	is the value which the variable is to be set to. If omitted, the variable is deleted. The value can be given directly or via the value of a variable.

Example: It is possible to set the value of one variable (e.g. SOURCE) to those of another variable (e.g. TARGET):

```
VAR TARGET ${SOURCE}
```

2.7.12 CVAR <options>

CVAR <options> gives access to variables stored on the controller by calling the "VAR" command. CVAR is to be used with the syntax and options of the VAR command as described in the controller's user manual.

2.7.13 VAR? <variable>

Get the value stored in a variable. If VAR? is combined with CPY, JRC, MEX or WAC, the response to VAR? has to be a single value.

<variable>	is the name of the variable to be queried. See "Variables" (p. 49) for name conventions.

2.7.14 CVAR? <options>

CVAR? <options> gives reading access to variables stored on the controller by calling the "VAR?" command. CVAR? is to be used with the syntax and options of the VAR? command as described in the controller's user manual.

2.7.15 ADD <variable> <FLOAT1> <FLOAT2>

Add two values and store the result in a variable. For the summands, floating point numbers are expected. They can be given directly or via the value of a variable. See "Variables" (p. 47) for conventions regarding variable names and values.

<variable>	is the name of the variable to which the result is to be saved
<FLOAT1>	is the first summand
<FLOAT2>	is the second summand

It is also possible to substract using the ADD command. Example:

```
ADD c $b -$a
```

will substract the value of variable a from those of variable b. The result will be written to variable c. Note that the value of variable a must not have a negative sign.

2.7.16 CADD <options>

CADD <options> adds two values and stores the result in a variable stored on the controller by calling the "ADD" command. CADD is to be used with the syntax and options of the ADD command as described in the controller's user manual.

2.7.17 CPY <variable> <command?>

Copy a command response into a variable.

<variable>	is the name of the variable to which the command response is to be copied. See "Variables" (p. 49) for conventions regarding variable names.
<command?>	is one query command in its usual syntax. The response has to be a single value and not more.

Example: It is possible to copy the value of one variable (e.g. SOURCE) to another variable (e.g. TARGET):

```
CPY TARGET VAR? SOURCE
```

2.7.18 CCPY <options>

CCPY <options> copies a command response into a variable stored on the controller by calling the "CPY" command. CCPY is to be used with the syntax and options of the CPY command as described in the controller's user manual.

2.7.19 TIM [<FLOAT>]

Set the timer to the given value. The timer targeted by this command is an internal timer in PIMikroMove.

When no value is given, the timer is reset to zero.

<FLOAT> is the value in milliseconds the timer is to be set to.
Minimum allowed value: 0

The timer is incremented each servo cycle and can be used for time measurement. The incrementation starts at zero each time the controller is switched on or rebooted.

2.7.20 CTIM <options>

If a controller provides timer functionality, CTIM <options> gives access to it by calling the "TIM" command. CTIM is to be used with the syntax and options of the TIM command as described in the controller's user manual.

2.7.21 TIM?

Get the current value of the timer. The timer queried by this command is an internal timer in PIMikroMove.

PIMikroMove returns the current value of the timer in milliseconds; format: FLOAT

The timer is incremented each servo cycle and can be used for time measurement. The incrementation starts at zero each time the controller is switched on or rebooted.

2.7.22 UNLOAD

Terminates the currently running macro and unloads it from the program's buffer.

If the same macro is started again, it will be loaded anew from the PC.

2.7.23 Variables

With host macros, variables are provided for more flexibility in programming. While global variables can be used for multiple macros, local variables are only valid for a given macro. There is no limit for the number of local and global variables.

The values of local variables must be given in the ***Arguments*** field of the ***Host macros*** tab card before starting the macro. The values have to be separated by space characters. Macro execution may fail if the macro uses local variables but no entries are made in the ***Arguments*** field.

Using the ***Clear*** variables button on the ***Host macros*** tab card, you can delete all variables in the host macro interpreter of PIMikroMove.

During macro execution, variable values can be monitored in the ***Run host macro window*** (p. 42).

Inside of host macros, you can use the ADD, CPY, VAR and VAR? commands to deal with variables.

INFORMATION

The local variable 0 is read-only. Its value gives the number of arguments set for the host macro.

Conventions for variable names:

- Variable names must not contain special characters, especially no "$".
- The maximum number of characters is 8.
- Names of global variables can consist of characters A to Z and 0 to 9. They must start with an alphabetic character.
- Names of local variables must not contain alphabetic characters. Possible characters are 0 to 9.
- The variable name can also be given via the value of another variable.

If the value of a variable is to be used, the notation must be as follows:

- The variable name must be written with preceding "$".
- Variable names consisting of multiple characters must be put in curly brackets.
- If the variable name consists of a single character, the curly brackets can be omitted.

Note that if variable names consisting of multiple characters are not enclosed in curly brackets, the first character after the "$" is interpreted as the variable name.

2.7.24 Example: Wait and Trigger

Single controller example of how to send a trigger signal after the motion of an axis has finished. Put the logic of waiting and triggering in a macro named WAITTRIG and call it after each MOV or MVR command.

Macro waittrig.txt:

```
WAC ONT? A = 1
DIO 1 1
DEL 100
DIO 1 0
```

You can use this in other macros:

```
MOV A 10
MAC START WAITTRIG
MOV A 0
MAC START WAITTRIG
```

2.7.25 Example: Trigger at Specified Position

Single controller example of how to send a trigger signal at a specified position during the motion of an axis.

```
DIO 1 0
MOV 1 10
WAC POS? 1 > 5
DIO 1 1
WAC ONT? 1 = 1
DIO 1 0
MOV 1 0
WAC ONT? 1 = 1
```

This macro will do the following:

Line 1: set digital output 1 to 0

Line 2: start motion of axis 1, target 10

Line 3: wait until the position of 1 is greater than 5

Line 4: set digital output 1 to 1

Line 5: wait until axis 1 is on target

Line 6: set digital output 1 to 0

Line 7: move back to 0

Line 8: wait until axis is on target

2.7.26 Example: X-Y-Scan with Different Macros

Example of loops implemented with MAC NSTART: All three macros together will perform a 10x10 mm scan. At each target point during the scan one of the digital outputs is set high for 100 ms to trigger an external device.

Macro scanxy.txt:

This macro will move two axes, A of controller 1 and C of controller 2, to their home positions and wait for both axes to stop. Then the macro scanrow.txt is called 10 times.

```
1 MOV A 0
2 MOV C 0
255 WAC #5 = 0
0 MAC NSTART SCANROW 10
```

Macro scanrow.txt:

This macro will move axis A of controller 1 by 1 mm and wait for it to stop. Then the macro scanstep.txt is called 10 times. At the end, axis C of controller 2 is moved back to its home position.

```
1 MVR A 1
1 WAC ONT? A = 1
0 MAC NSTART SCANSTEP 10
2 MOV C 0
2 WAC ONT? C = 1
```

Macro scanstep.txt:

This macro will move axis C of controller 2 by 1 mm, wait for it to stop and set the digital output 1 of controller 1 to 1 for 100 milliseconds.

```
2 MVR C 1
2 WAC ONT? C = 1
1 DIO 1 1
0 DEL 100
1 DIO 1 0
```

2.7.27 Example: X-Y-Scan with Loops

This macro will move the two axes A of controller 1 and C of controller 2 to their home positions and start a 10x10 mm scan with both axes.

Macro scanxyloop.txt:

```
1 MOV A 0
2 MOV C 0
255 WAC #5 = 0
0 LOOP 10
1   MVR A 1
1   WAC ONT? A = 1
0   LOOP 10
2      MVR C 1
2      WAC ONT? C = 1
1      DIO 1 1
0      DEL 100
1      DIO 1 0
0   ENDLOOP
2   MOV C 0
```

```
2   WAC ONT? C = 1
0 ENDLOOP
```

2.7.28 Example: Pushbutton Control

If you have connected four push buttons to 4 digital inputs you can implement pushbutton control of five different macros using the "MEX" command.

The following single controller example illustrates a small program: you can move axis A 1 mm forward or backward with button 1 and 2 and you can switch between two velocities with buttons 3 and 4.

Macro KEYPBLOOP.txt

This macro will call the macros for the individual buttons and then call itself.

```
MAC START KEYPB1
MAC START KEYPB2
MAC START KEYPB3
MAC START KEYPB4
MAC START KEYPBLOOP
```

Macro KEYPB1.txt

This macro will exit if digital input 1 (keyPB 1) is not active, otherwise a relative move of 1 mm is made for axis A. The short delay prevents this macro being called too rapidly when the button is pressed.

```
MEX DIO? 1 = 0
MVR A 1
DEL 100
```

Macro KEYPB2.txt

This macro will exit if digital input 2 (keyPB 2) is not active, otherwise a relative move of -1 mm is made for axis A. The short delay prevents this macro being called to rapidly when the button is pressed.

```
MEX DIO? 2 = 0
MVR A -1
DEL 100
```

Macro KEYPB3.txt

This macro will exit if digital input 3 (keyPB 3) is not active, otherwise the velocity is set to 0.5. The short delay prevents this macro being called too rapidly when the button is pressed.

```
MEX DIO? 3 = 0
```

```
VEL A 0.5
DEL 100
```

Macro KEYPB4.txt

This macro will exit if digital input 4 (keyPB 4) is not active, otherwise the velocity is set to 1.5. The short delay prevents this macro being called too rapidly when the button is pressed.

```
MEX DIO? 4 = 0
VEL A 1.5
DEL 100
```

When you combine commands for multiple controllers in those macros, it is also possible to connect a pushbutton control device to one controller but use it to command another controller. Example: Connect a pushbutton box to a Mercury class controller and use it to control a C-843 motor controller card in the PC.

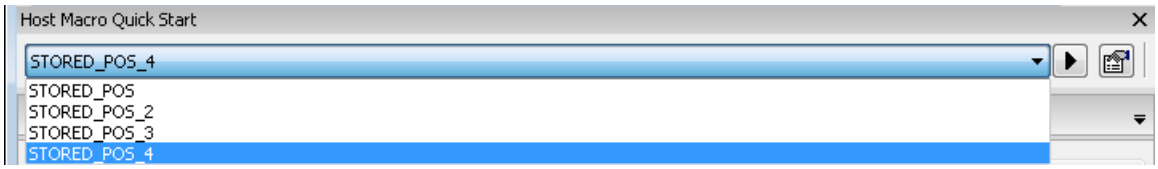
2.7.29 Host Macro Quick Start

The ***Host Macro Quick Start*** window is accessible from the PIMikroMove main window via a toolbar button and via the ***Tools*** menu.

The ***Host Macro Quick Start*** functionality offers quick access to host macros:

- You can select and start a host macro without the need to switch to the ***Host macros*** tab card
- You can assign a customized short-cut button to a host macro so that this macro can be started by simply clicking the button

For successful macro execution, the affected axes (controllers) must be connected via the same interface(s) as used during macro creation (see "Controls on Host Macro Tab Card" (p. 39) for more information).



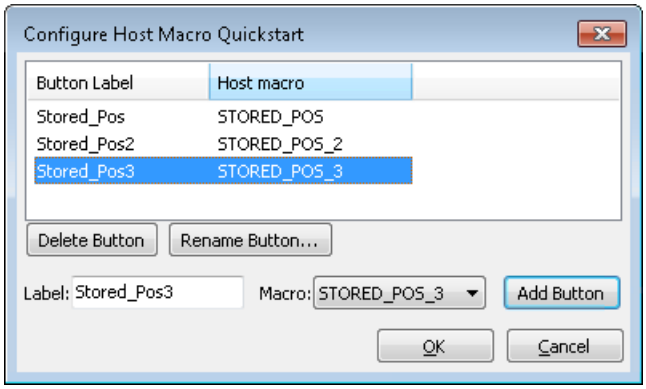
INFORMATION

The ***Host Macro Quick Start*** window can be docked to the borders of the main window by dragging it with the left mouse button pressed (see "Main Window" (p. 9)).

The macro selection field lists the same macros as the ***Host macros*** tab card. If you want a host macro to be included in the list, load it on the ***Host macros*** tab card using the ***Open and show another macro from text file*** button in the icon bar. Macros created as described in "Create

New Host Macro with Current Positions" (p. 55) are automatically added to the list. Execution of the selected macro can be started by clicking the icon.

The short-cut buttons can be configured in a separate window which can be opened via the icon.



The ***Configure Host Macro Quickstart*** window lists the existing short-cut buttons with their labels and assigned macros. You can delete the selected button (the assigned macro is not deleted) or rename it. The button label can differ from the name of the assigned macro.

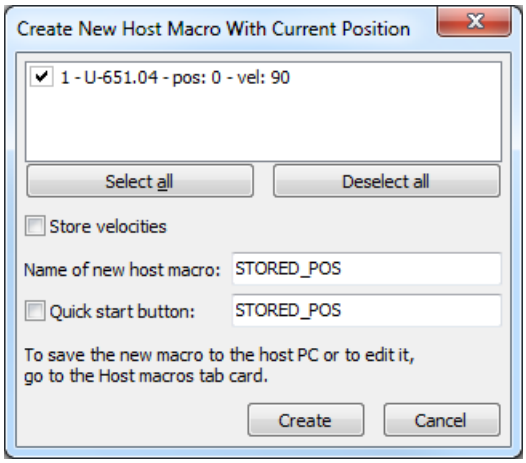
To create a new short-cut button, enter its label in the ***Label*** field, select the macro to be assigned in the ***Macro*** field, and click ***Add Button***. New buttons are also added when creating new host macros as described in "Create New Host Macro with Current Positions" (p. 55).

INFORMATION

It is not possible to change the macro assignment of an existing button. If you want to assign another macro to a button, delete this button and create it again with the new macro selected.

2.7.30 Create New Host Macro with Current Positions

The ***Create new host macro with current position*** window is accessible via a main-window toolbar button and the individual ***Axis*** menus. The functionality offered by this window is useful whenever it is necessary to recall certain positions for selected axes.



INFORMATION

This window is only available when at least one of the connected axes has servo on. Axes equipped with incremental sensors must be in the referenced state to appear in this window.

The topmost pane of the window shows the available axes with their current positions. These are the position values that will be used in the host macro to be created (position changes after window opening are ignored). Using the checkboxes, you can select the axes to be included in the host macro.

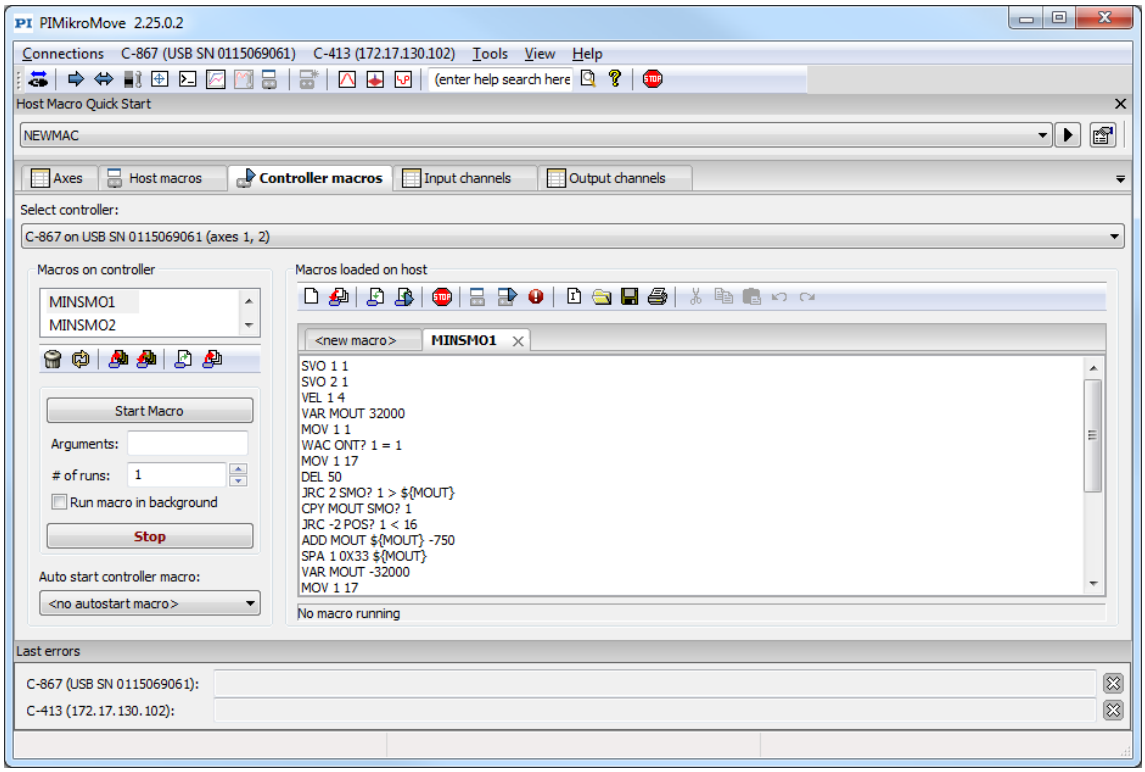
If supported by the controller(s), the current velocity settings of the selected axes can be included in the host macro by checking the ***Store velocities*** option.

Enter the name for the host macro to be created in the corresponding field.

For easy access, a separate short-cut button will be assigned to the host macro if the ***Quick Start Button*** box is checked. In the field to the right of the checkbox, you can enter the button label which may differ from the host macro name. You can start the host macro at a later date by simply clicking the button as long as the axes (controllers) affected by the host macro are connected via the interface(s) used for the current connection. See "Host Macro Quick Start" (p. 54) for more information.

Clicking the ***Create*** button creates the new host macro with the necessary motion commands and - if selected - velocity commands. The macro is not saved to the host PC yet. To save it to the host PC or to edit it like any other host macro, go to the ***Host macros*** tab card. The new host macro is listed there with <not saved> as path information. See "Controls on Host Macro Tab Card" (p. 39) for details.

2.8 Controller Macros



Controller macros are sequences of commands which are stored directly on the controller. They can be executed independently without a host PC. The ***Controller*** macros tab card gives access to the complete macro functionality of all connected controllers which provide the macro feature. You can select the ***Controller macros*** card by clicking on its tab or using the arrow button in the tab bar.

In the ***Select controller*** field, select the controller whose macros are to be displayed.

Macros on controller

The list shows the macros stored on the controller. The icons below the macro list have the following functions:

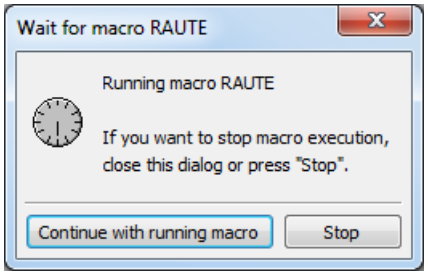
- displays the names of all macros stored on the controller
- deletes the selected macro(s) on the controller
- saves the selected macro(s) as text file(s) to the host PC
- sends macro(s) stored on the host PC to the controller
- loads the selected macro from the controller
- loads all selected macros from the controller

Start Macro
You can start a macro with this button.

Arguments
Provided that the controller supports variables, enter the values of the local variables used by the macro to be started in the ***Arguments*** field. The values have to be separated by space characters. Macro execution may fail if the macro uses local variables but no entries are made in this field. See the user manual of the controller for a detailed description of variables.

# of runs
To have the macro command sequence executed not only once, but a number of times, select or enter a number in the ***# of runs*** field.

Run macro in background
If this option is available (depends on controller) and checked, you can continue working with PIMikroMove and perform other tasks (e.g. start a host macro) while the macro is running on the controller. If the option is not checked, you can choose background processing of the macro via the ***Continue with running macro*** button in the ***Wait for macro*** dialog:



Stop
With the ***Stop*** button on the ***Controller macros*** tab card you can immediately stop the currently running macro.

Auto start controller macro
From the drop-down list, you can select one of the macros stored on the controller to be the startup macro. This macro will be run automatically every time the controller is powered on or rebooted.

Macros loaded on host

The right pane of the tab card contains an edit field for macros. Toolbar buttons are available for:

- Creating macro command sequences
- Saving macros to / loading them from the controller

  You can save / load either a single macro or multiple macros.

  Note that with controllers which require a separate saving procedure, the ***Send macro to controller*** and ***Send all open macros to controller*** buttons will save macros to volatile memory only, and the ***Save parameters to non-volatile memory*** button must be used afterwards to save them permanently.
- Load and run controller macros as host macros. That way, controller macros become also available on the Host macros tab card (p. 38) where they can be run by controllers which do not provide the macro feature.
- Opening macros from / saving them to text files on the host PC.

INFORMATION

Depending on the controller, the icon is present in the toolbar of the right tab card pane. Using this icon you can report the first error which occurred during macro execution. The error report contains the name of the macro, the line in the macro where the error occurred, the error code and the command that caused the error.

The following keyboard shortcuts are available:

- Ctrl+S saves the current macro
- Ctrl+D sends the current macro to controller
- Ctrl+R starts the current macro; if the macro content has changed, the macro is sent to controller before it is started

Most of the controllers that can be connected to PIMikroMove automatically save all changes regarding macros to non-volatile memory. But some controllers require a separate, password-protected procedure to save changes permanently (e.g. if a controller macro was created, has been changed, deleted or (re-)defined as startup macro). If this is the case for one of the connected controllers, the ***Save parameters to non-volatile memory*** button is present in the bottom left corner of the ***Controller macros*** tab card (not shown in the figure above).

INFORMATION

For the password required when using ***Save parameters to non-volatile memory***, see the WPA command description in the controller's user manual.

Depending on the controller, ***Save parameters to non-volatile memory*** may save further settings to the controller's non-volatile memory. See the controller's user manual for details.

To save macro content you typed in the text field, first use the ***Send macro to controller*** button and then the ***Save parameters to non-volatile memory*** button.

The message that is displayed in the line below the edit field for macros indicates whether a macro is currently running.

For further details regarding the macro functionality see the controller's user manual. If the controller does not support the macro feature, you can use the host macro functionality of PIMikroMove instead. See "Host Macros" (p. 38) and "What is the difference between "controller macros" and "host macros"?" (p. 188) for details.

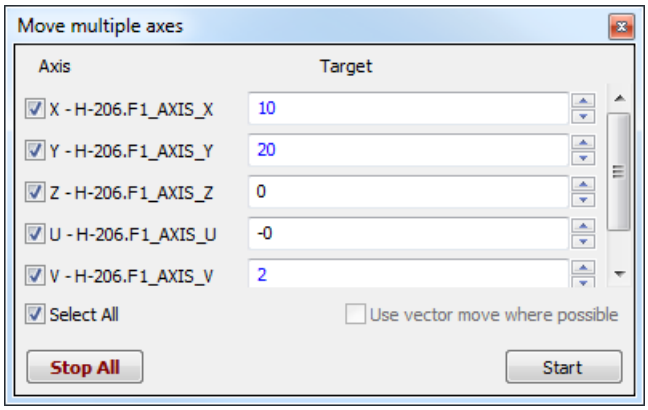
3 Additional Common Windows

The windows described below are available for almost all controllers (exceptions noted below). They are accessible via the ***Tools*** menu and the ***View*** menu in the main window. Some can also be opened with a key combination or toolbar button.

Windows which are in any way controller-specific are accessible from the corresponding controller menu, see "Controller Specific Windows" (p. 103).

In this Chapter

3.1 Move Multiple Axes



INFORMATION

This window is only available when at least for one of the connected axes the servo is switched on. Axes equipped with incremental sensors must be in the referenced state to appear in this window.

The ***Move multiple axes*** window allows starting and stopping a move of several axes by clicking a single button. You can configure which axes will be moved and set their targets. The velocities to be used for the axes can be set, for example, on the ***Axes tab*** card in the main window.

The start of the moves will not be synchronized if they are connected to different controllers. Axes connected to the same controller will be synchronized to the extent the controller supports synchronous moves.

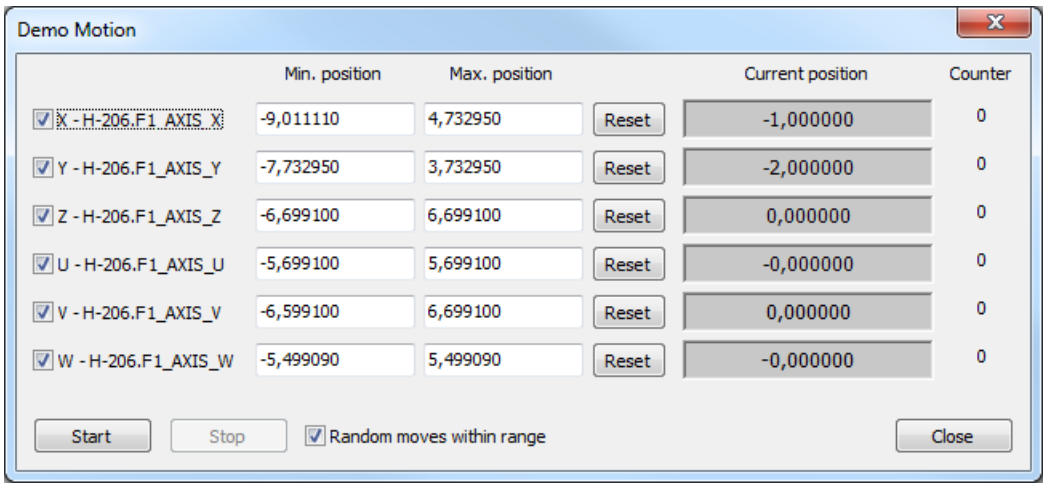
With controllers supporting simultaneous "vectorial" motion of coupled axes (MVE command), you can check ***Use vector move where possible***. The axes to be moved must then be connected to the same controller. The combined motion of the coupled axes will describe a linear path. This is achieved by appropriate calculation of accelerations, velocities and decelerations. The current settings for velocity, acceleration and deceleration define the maximum possible values, and the slowest axis determines the resulting velocities.

3.2 Demo Motion

Opens the ***Demo Motion*** window.

INFORMATION

This window is only available when at least one of the connected axes has servo on. Axes equipped with incremental sensors must be in the referenced state to appear in this window.



The motion range can be set in the ***Min. position*** and ***Max. position*** fields. ***Reset*** will reset the position limits to their default values.

INFORMATION

With Hexapod mechanics, the default position limits may be too large for your model type. Adapt the limits to the travel range values given for that model in the Hexapod User manual.

INFORMATION

Hexapod axes are not moved to zero position at startup. Instead, the current position of an axis is used as the "center" for demo motion.

The white bar in the ***Current position*** field for each axis serves as a graphic representation of the axis' position. The ***Counter*** values give the number of target changes during motion.

You can switch between two different motion modes:

- Random moves (within the given range)
- Full-range moves in alternating directions, i.e. back and forth between the specified minimum and maximum positions.

3.3 Stage Editor

INFORMATION

The stage editor function is only available if the PIStages3Editor or PIStageEditor is installed on the PC.

PI's stage editor is a tool that permits examining positioner databases which are located on the host PC. The content of these database files is used by PIMikroMove for the initial configuration of positioning systems.

You can open the stage editor from the main window as follows:

- Via the ***Tools > Stage Editor*** menu sequence
- Via the ***Show PI Stage Editor*** icon in the toolbar

If multiple positioner databases are available on the host PC for the connected controllers, you have to select one when opening the stage editor.

INFORMATION

Depending on the controller, you can also inspect parameters on the tab cards of the main window (p. 9), in the ***Single-Axis*** window (p. 97) or using the ***Device Parameter Configuration*** window (p. 144).

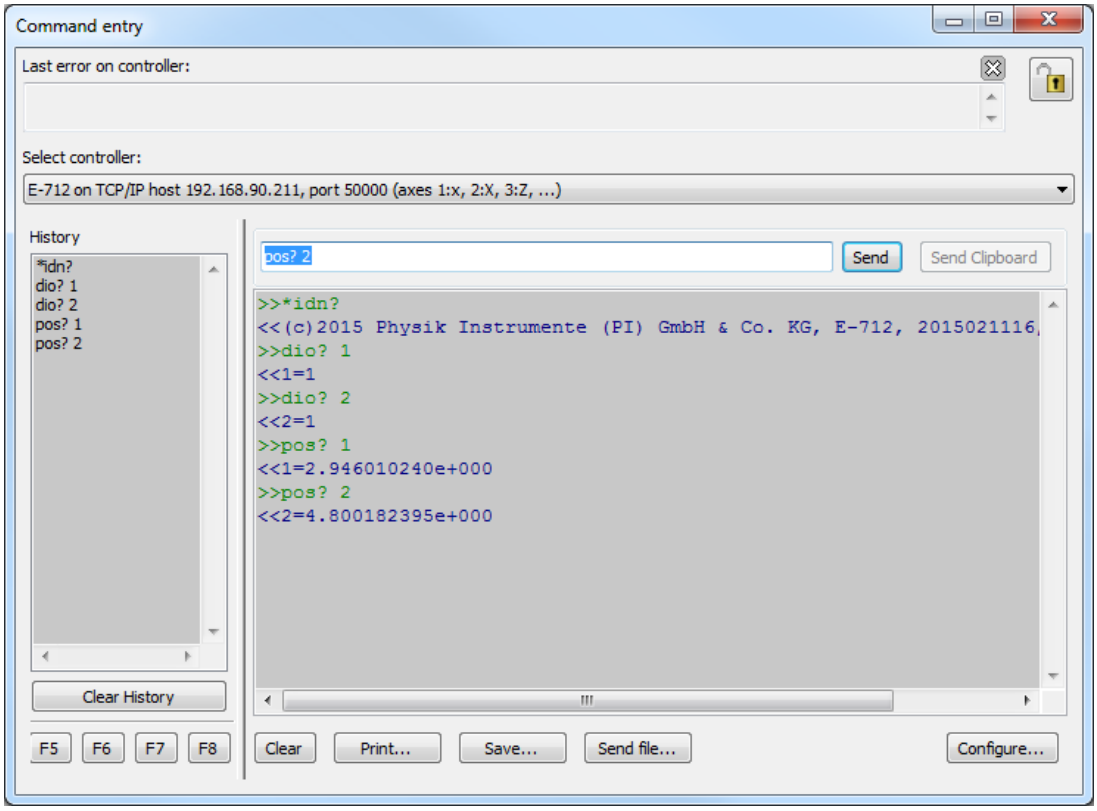
To simply store the current, active parameters under a new name, use the ***Save parameters as User Stage Type...*** function of the controller menu (p. 12) or the ***Axis*** menu (p. 29).

You can use the ***PIStages3Editor*** or ***PIStageEditor*** to view and edit parameters for user stage entries.

When saving entries as user stages, do not use names that already exist in the PIStages database. If there is a stage of the same name defined as a standard stage and at the same time as a user stage, the parameter settings of the standard stage are used when assigning that stage to an axis.

3.4 Command Entry

In the window ***Command entry*** (p. 64) commands can be entered and sent to the controller, and all the controller's responses are shown.



The window can be locked in order to get exclusive access to the controller communication. While the window is in the locked state, no updates of other PIMikroMove windows are allowed. (See "Why is everything disabled when Command entry is locked?" (p. 185)). Click on the lock icon to lock the ***Command entry*** window; to unlock the window, click the icon again.

In the window you can send GCS commands to a controller or controller network without any background processing done by PIMikroMove. For detailed descriptions of the commands, see the controller's user manual, GCS DLL manual, or the GCS Commands software manual. In addition, the response to the HLP? command consists of a listing of all commands and their arguments. It is also possible to send single bytes. To do so, enter a "#" followed by the decimal

value of the ASCII character. For example "#24" will send the byte with value 24 (^X), which is interpreted by most controllers to stop all axes and macros. If you need to send "#" as a character itself, type "##" or "#35".

In the ***Select controller*** field, select the controller or controller network which is to be commanded.

The left pane of the window shows the history of the commands typed in the ***Send*** field. When the cursor is placed in the ***Send*** text field, the contents of that field will be replaced by commands in the history list if you press the ⇑ or ⇓ arrow keys; pressing any other key returns you to the ***Send*** field where you can edit the command before sending. Four frequently used commands or command sequences are accessible with the F5, F6, F7 and F8 keys or buttons. To edit a sequence, hold down the control key while pressing the key or clicking the button.

The contents of the clipboard can be sent as-is with ***Send Clipboard***.

The display pane below the ***Send*** field by default lists all sent and received messages. If you right-click in the display, a menu opens which allows to select the whole display content and copy it to the clipboard. Furthermore, the ***Translate Error*** menu function is available when you have marked a number in the display. ***Translate Error*** is useful to obtain the detailed description which exists for every response to an ERR? command.

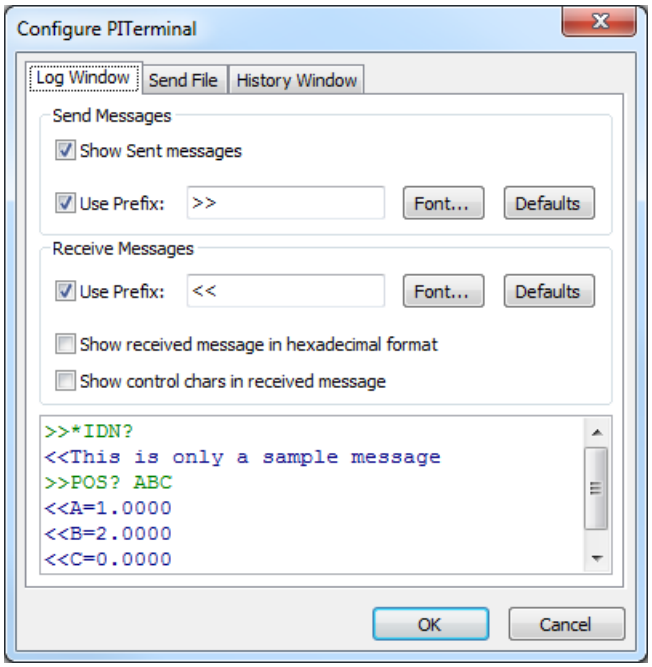
The display content can be deleted with the ***Clear*** button, sent to a printer with the ***Print*** button and saved to a file on the host PC using ***Save…***

With ***Send file…*** it is possible to load a command sequence from a text file on the host PC and send it to the controller.

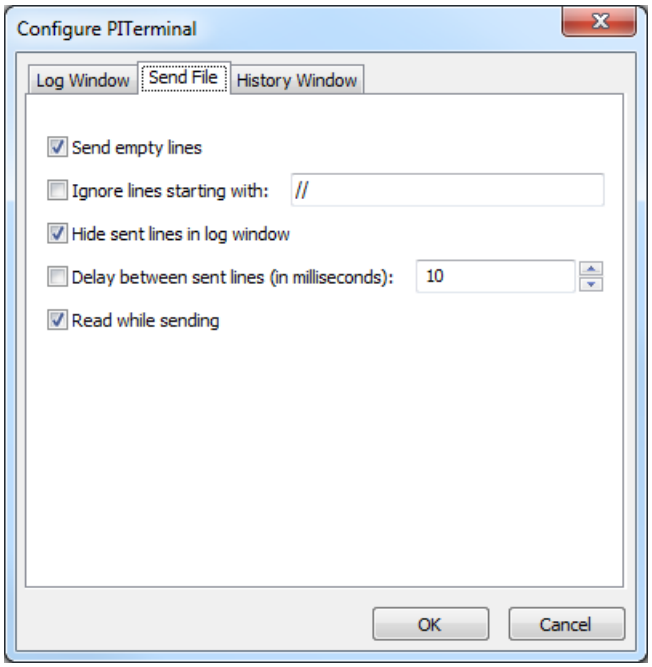
INFORMATION

Before you use ***Send file …***, make sure that the command sequence will not cause damage to your system. The commands are sent as-is and executed immediately.
The commands sent are not shown in the window. Send ERR? to find out if anything went wrong.

You can modify display options and the behavior when sending commands using a dialog which is opened with the ***Configure...*** button:

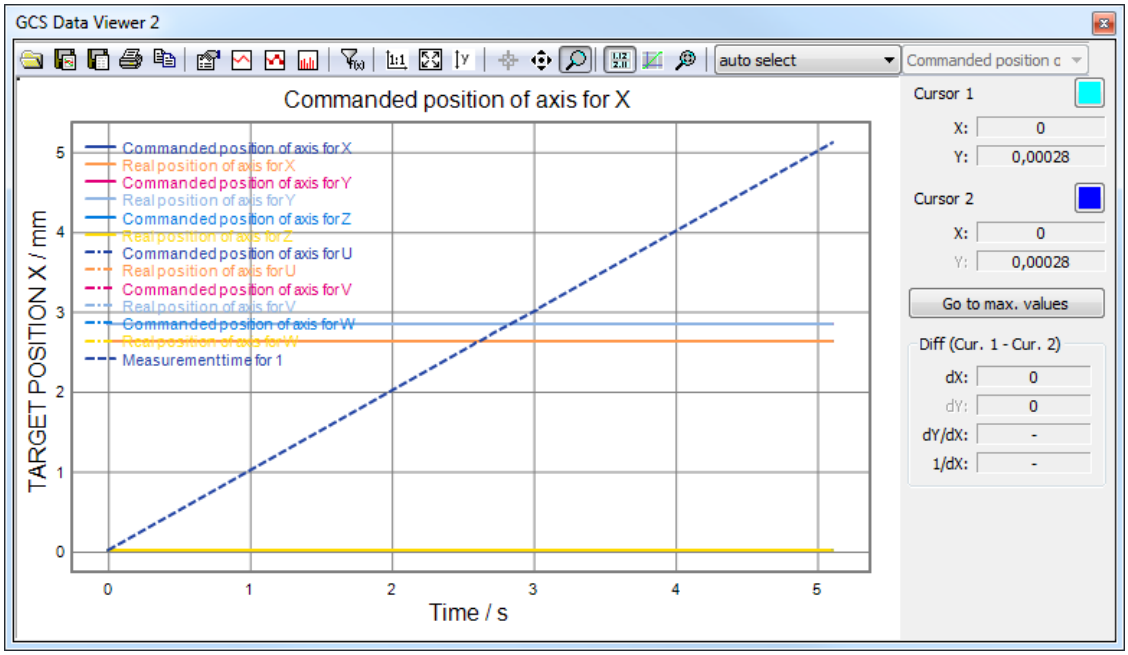


For example, it is possible to switch off the display of the strings sent, so only received messages are shown. You can also change the font and color of sent and received messages and activate / deactivate prefixes, show the received messages in hexadecimal format or show control characters in received messages.



3.5 GCS Data Viewer

In the ***GCS Data Viewer*** you can view recorded data and perform operations with it. To open the window no controller has to be connected in PIMikroMove.



When you open the ***GCS Data Viewer***, e.g. via the ***Tools*** menu or the toolbar of the main window, a file selection dialog is opened in which you can select a GCS array file (format: .dat). The data of the selected file is then displayed in the window's graphics pane.

You can load data from multiple GCS array files: Click the button and, in the file selection dialog, select another GCS array file. This opens a dialog in which you are asked whether you want to keep the existing data. Click ***Yes*** to add the new curves to the ones already loaded.

Configure the data display in the graphics pane (p. 68) and perform operations with the data via the window's toolbar buttons.

GCS array format

A GCS array file consists of a header and a data part. If a GCS array file contains several data series, each data series can be in a separate data part with its own header. The first line of each header is always in square brackets and contains the name of the data series.

Newer GCS array files that were created through an export from PIMikroMove, in addition contain information on display properties of the chart (e.g. title, axis labels) as well as of the individual data series (e.g. color, line style). When a GCS array file is loaded into the ***GCS Data Viewer***, it is displayed according to the display properties it contains. If no display properties are contained in the file, the display of the data uses the graphic pane's default settings, and the title of the chart is the path and name of the loaded file.

You can find a detailed description of the GCS array format in the SM146E manual.

3.5.1 Graphics Pane

You can configure the data display in the graphics pane and perform operations with the data via the window's toolbar. A short description is displayed when the mouse cursor hovers over a toolbar button.

Functions of the toolbar

Import data
You can load data to the display by importing them from .dat files on the host PC (GCS Array format, see GCS Array manual, SM146E).

Save image of graph
You can save the displayed graph to the host PC as an image file. Specify the size of the image in the dialog ***Select Image Size***.

Export data as data file
You can save the display content to the host PC by exporting the data as comma-separated values (.csv format) or in GCS Array format (see GCS Array manual, SM146E).

Print image of graph
You can send the displayed graph to the printer. For this, a print preview window is opened.

Copy graph image to clipboard
You can copy the graph image to the clipboard from where it can be pasted to another application.

Configure display properties
Opens the dialog ***Display Properties*** (p. 72) to configure the display of graphs.

Display lines
Sets the appearance of all data in the display to "lines". To change the appearance of single curves, configure them in the ***Display Properties*** dialog.

Display lines with rectangles
Sets the appearance of all data in the display to "lines with rectangles". To change the appearance of single curves, configure them in the ***Display Properties*** dialog.

Display sticks
Sets the appearance of all data in the display to "sticks". To change the appearance of single curves, configure them in the ***Display Properties*** dialog.

Show / Hide data toolbar
Toggles the display of the window's data toolbar (see below).

Use same scaling for X and Y
Uses the same scale for the X and Y axes of the graphics display. Clicking the button again resets the scaling to the original state.

Autoscale
In case of a changed scaling, this function resets the graphics pane to its original state.

Specify Y range
Opens the dialog ***Select range for Y axis*** in which you can specify the start and end values of the Y axis in the graphics display, or set the range of the Y axis to ***Automatic***.

Use the mouse to move the cursor in the graph
This icon is only available when the cursors are displayed via the function ***Display cursors***. If you activate this icon, you can move one of two cursors along a graph by dragging it with the left mouse button pressed. Choose the active cursor and the graph connected to the active cursor in the toolbar. The two cursors can be connected to the same graph. While you move a cursor, you can see the individual coordinates in the cursor information panes of the ***Graph Control Toolbox*** (see below).

Use the mouse to pan the graph
If you activate this function, you can scroll the content of the graphics pane by placing the mouse pointer in the graphics pane and dragging it with the left mouse button pressed. To reset the display to its original state use the ***Auto range*** button.

Use the mouse to zoom the graph
If you activate this function, you can magnify a selected area of the graphics pane. To do this, drag the mouse diagonally across the desired area with the left mouse button pressed. To reset the display to its original state use the ***Auto range*** button or right-click anywhere inside the display.

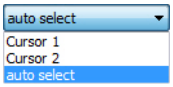
Display / Hide the graph control toolbox
The ***Graph Control Toolbox*** is shown or hidden. If activated, the toolbox is displayed to the right of the graphics pane (see below).

Display / Hide cursors
The cursors in the graphics pane are shown or hidden. If the cursors are hidden, the cursor-related icons in the toolbar are deactivated (i.e. you cannot track the cursors with the mouse and it is not possible to select the active cursor and the graph connected to the active cursor).

Auto range
In case of a zoomed or shifted display, this function resets the graphics pane to its original state, so that all the data is completely shown.

Text fields to the right of the icon bar:

Select the active cursor
This combo box is only available when cursor tracking is activated in the toolbar. You can select the cursor which is to be moved by the mouse.



Select the graph connected to the active cursor
This combo box is only available when cursor tracking is activated in the toolbar. You can specify along which graph the active cursor is to move.

Graph Control Toolbox

If active, the graph control toolbox is displayed to the right of the graphics pane. For ***Cursor 1*** and ***Cursor 2***, it contains information fields (***X***, ***Y***) for the display of cursor coordinates. Clicking the button ***Go to max. values*** sets the cursors to the maximum values of the X and Y axes. In the ***Diff (Cur. 1 - Cur. 2)*** fields, the absolute values of the difference between the cursors are displayed.

Data Toolbar

FFT

With the ***FFT*** (fast Fourier transformation) button, you can switch between two display modes in the graphics pane: the standard display and a frequency diagram of the read data. The current display properties for the FFT diagram can differ from those for the standard display. Use the ***Display Properties*** dialog to select the preferred settings.

Algebra

Opens the dialog ***Math Operation*** (see below) in which you can define an arithmetic operation or a scaling factor which will be applied by PIMikroMove to the selected data.

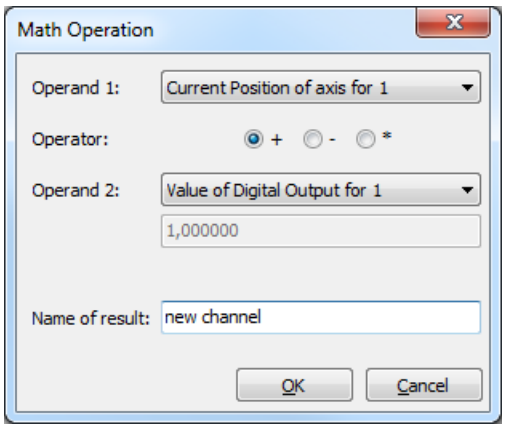
Derivative

Opens the dialog ***Add Derivatives*** (see below) in which you can select data for which derivatives are to be calculated by PIMikroMove.

Statistics

Displays an area in the window which contains statistical data for the data shown in the graph (see below).

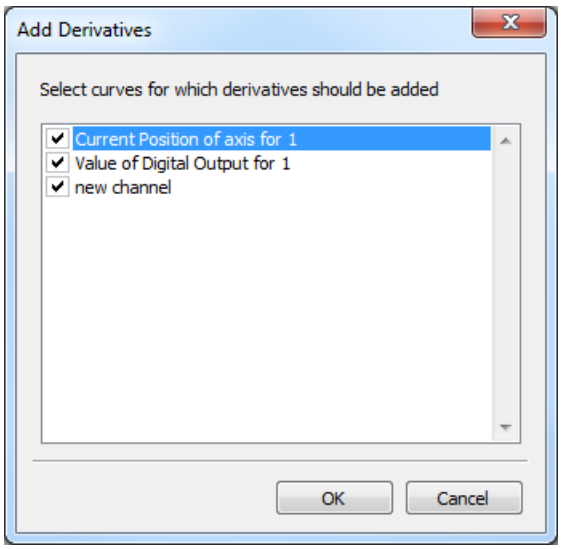
Math Operation



In the dialog ***Math Operation*** you can define an arithmetic operation or a scaling factor which will be applied by PIMikroMove to the selected data. The results are only present on the host PC but not in the data recorder tables of the controller. Arithmetic operations or the scaling factor can be applied to all recorded data and to all data resulting from former calculations (also to derivatives).

The appearance of the calculated data in the display can be configured via the ***Display Properties*** dialog.

Add Derivatives

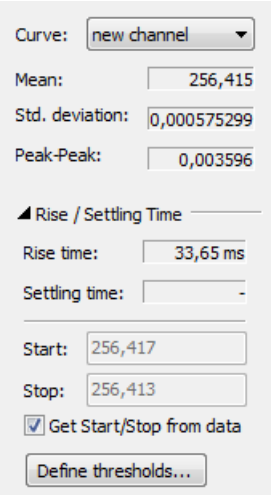


In the dialog ***Add Derivatives*** you can select data for which derivatives are to be calculated by PIMikroMove. These derivatives are only present on the host PC but not in the data recorder tables of the controller. Derivatives can be calculated for all recorded data and for all data resulting from former calculations (derivatives or results of arithmetic operations).

The appearance of the calculated data in the display can be configured via the ***Display Properties*** dialog.

Statistics

For the curve selected via the field ***Curve***, statistical data is shown.



Mean shows the arithmetic mean of the curve's values, ***Std. deviation*** its standard deviation, and ***Peak-Peak*** the distance between the maximum positive and the maximum negative amplitudes of the curve.

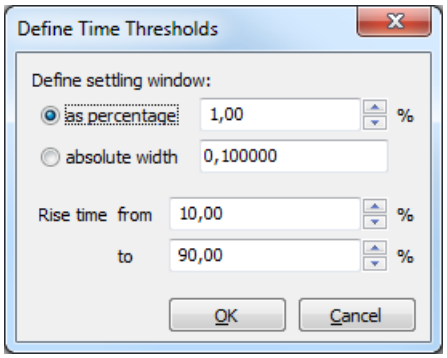
To see the data on rise and settling times, the ***Rise / Settling Time*** part of the window can be expanded:

Rise time shows the time required for increasing the amplitude from 10 % to 90 % of its maximum. 10 % and 90 % are the default values for the rise time. They can be changed via the button ***Define thresholds...***.

Settling time shows the time required for reaching the settling window around the target position, which is 1 % of the maximum amplitude. 1 % is the default value for the settling window. It can be changed via the button ***Define thresholds...***.

To view the rise and settling times for different starting and/or end points, uncheck the option ***Get Start/Stop from data*** and enter the desired values in the ***Start*** and ***Stop*** fields.

The button ***Define thresholds...*** opens the dialog ***Define Time Thresholds*** in which you can define thresholds for the settling window and the rise time:



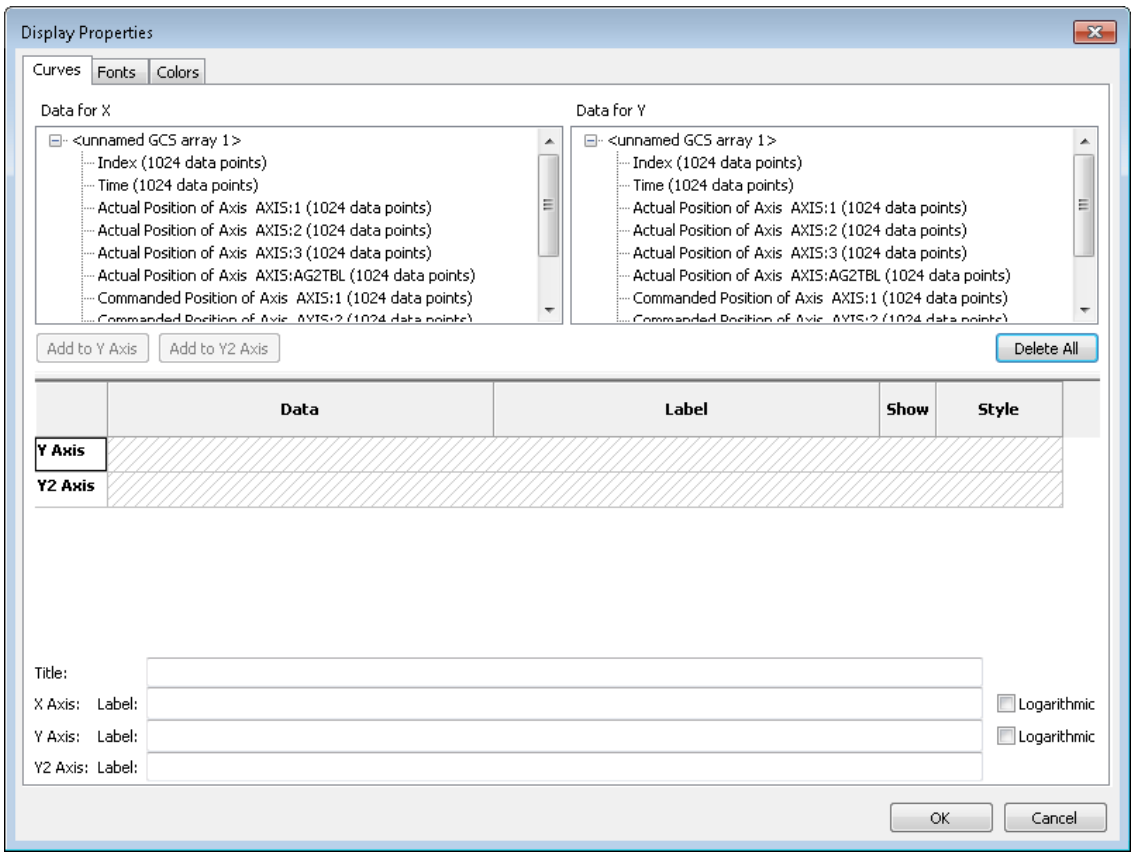
The settling window is defined either ***as percentage*** of the distance covered by the motion, or as ***absolute width*** in axis units.

The rise time is defined by two percentage values which set the starting (***from***) and end (***to***) points for measuring.

Display Properties

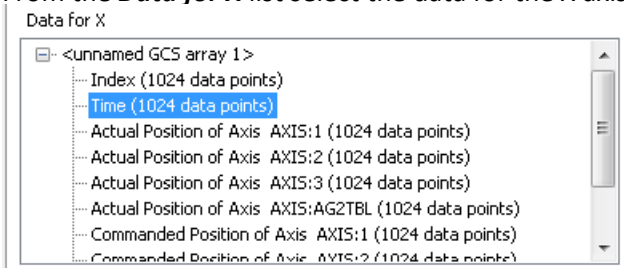
In the ***Display Properties*** dialog, you can define the data curves to be displayed in the graph and adjust the graph's settings.

Set the data curves to be displayed on the tab ***Curves***.

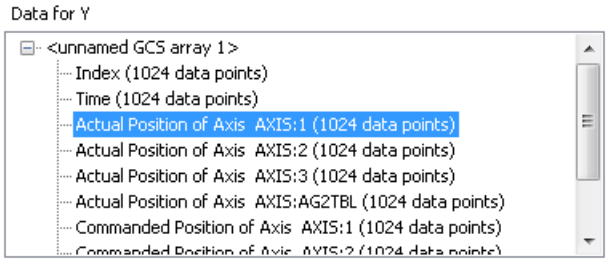


For each curve to be displayed in the graph, proceed as follows:

1. Select the data that is to make up a curve:
   - From the ***Data for X*** list select the data for the X axis of the graph.



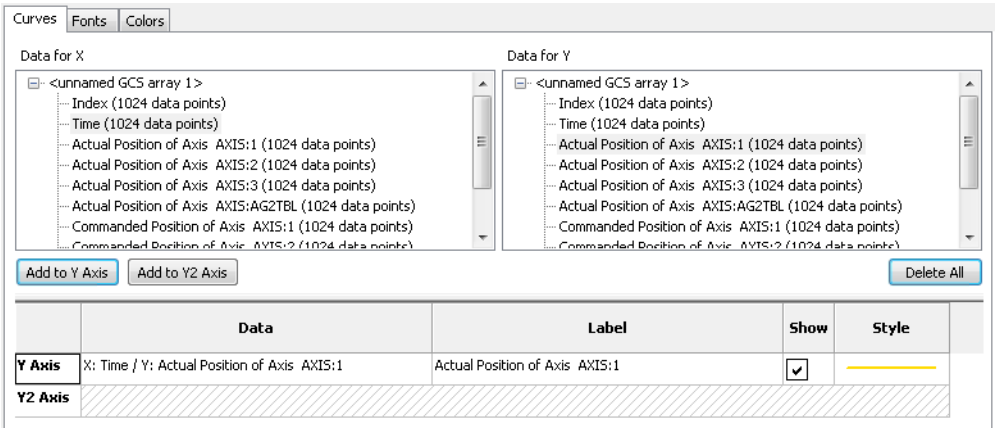
   - From the ***Data for Y*** list select the data for the Y axis of the graph.



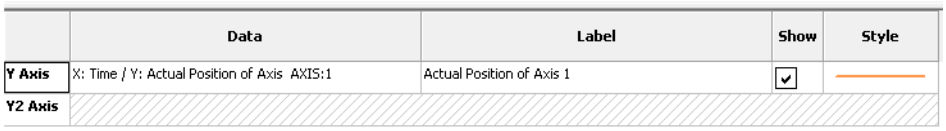
   Only when data for *both* axes of the graph are defined, the buttons ***Add to Y Axis*** and ***Add to Y2 Axis*** become active.

2. Click the button ***Add to Y Axis*** or ***Add to Y2 Axis*** to add the curve to the graph.

   It is recommended to assign any data whose range is greatly different from that of other data to the graph's second Y-axis (Y2 axis).



3. Adjust the display settings for the curve:
   - If you want to change the text that is shown as the curve's legend in the graph, you can directly edit the text in the table field ***Label***.
   - If you want to change the display style of the curve, click in the field ***Style***. This opens a dialog in which you can change the style of the curve.



Now proceed with the definition of all the curves to be displayed in the graph.

The defined curves are shown in the table with this data:

- The leftmost column contains an entry for each Y axis of the graph: ***Y Axis*** = the left Y axis of the graph (standard), ***Y2 Axis*** = the right Y axis of the graph. Below each entry, the assigned curves are listed.
- ***Data*** shows data for X and for Y that makes up the curve.
- ***Label*** shows the legend entry used for the curve in the graph.
- By checking/unchecking the ***Show*** checkbox, you can tell PIMikroMove to display/hide the curve in the graph.
- ***Style*** shows how the curve is displayed in the graph.

Right-clicking a table entry opens its **context menu** which provides the following functions:

- ***Delete***
  Deletes the selected curve definition.

  To delete all curve definitions click the button ***Delete All***.
- ***Move to Y2 axis*** / ***Move to Y axis***
  Assigns the selected curve definition to the Y2 / Y axis of the graph.

  A curve's Y axis assignment can also be changed via drag & drop: Click in the leftmost table column of the curve you want to move, and, with the left mouse button pressed, drag the entry to the desired axis.

The display options for the graph are set on the tabs ***Curves***, ***Fonts***, and ***Colors***:

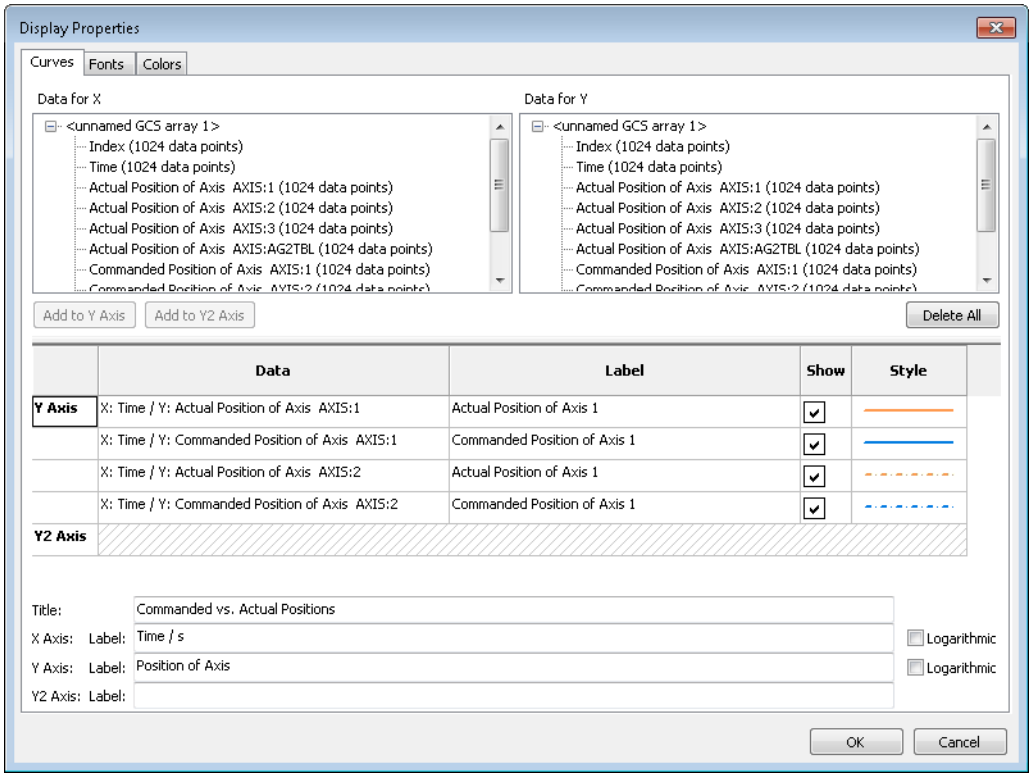
- Description texts:

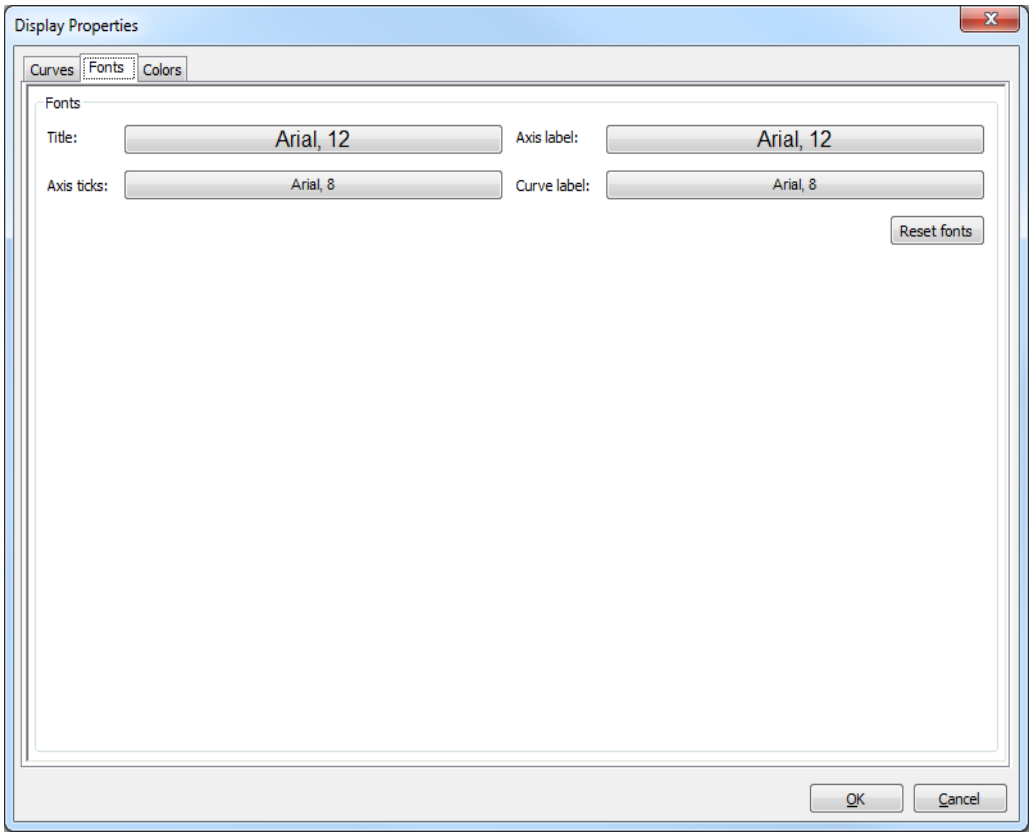
  In the fields below the table on the tab ***Curves*** specify the texts to be shown as the graph title (***Title***) and as the descriptions along the graph's axes (***<Axis>: Label***).
- Fonts:

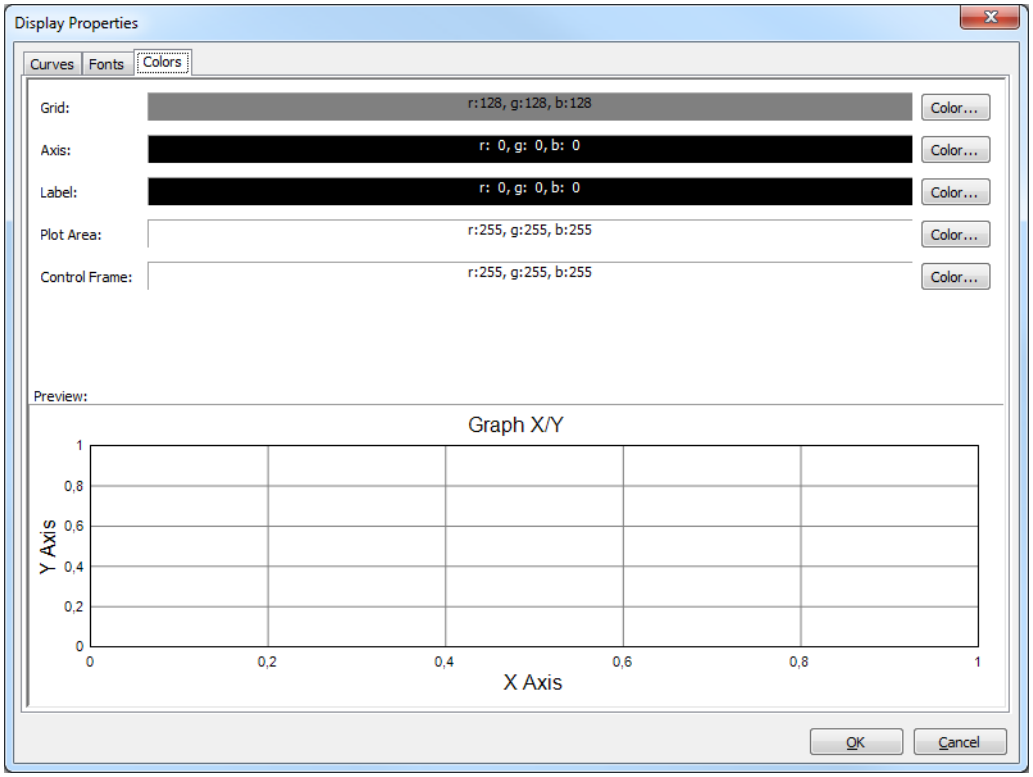
  On the tab ***Fonts*** (see below) you can define the fonts to be used for the graph's text elements.
- Colors:

  On the tab ***Colors*** (see below) you can define the colors to be used for the graphic elements of the the graph. Current settings are displayed in a preview window.

Once you have set all the display properties of the graph, click the button ***OK*** (in any of the window's tabs). This closes the window ***Display Properties*** and applies the defined settings to the graph.

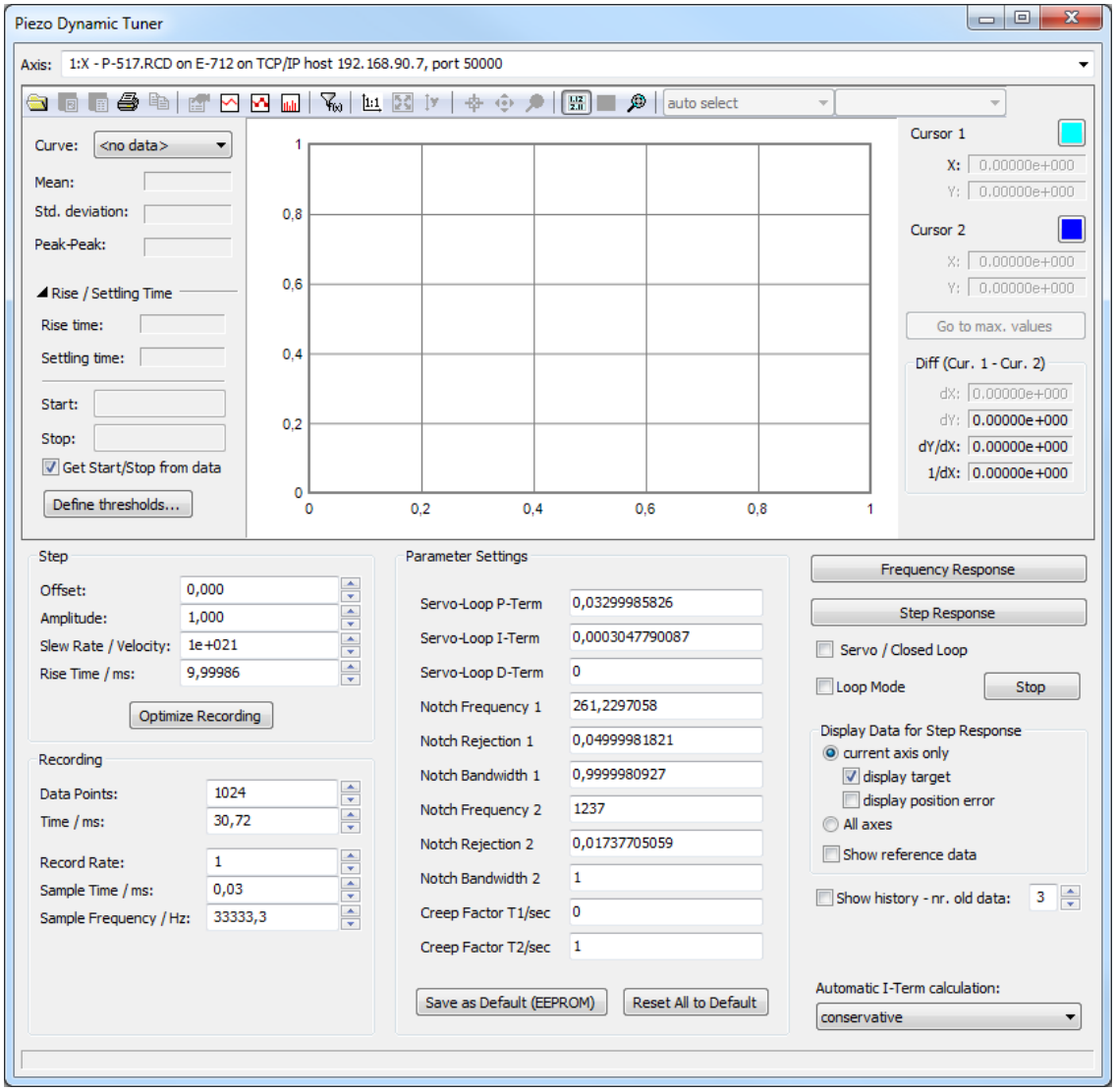

Display Properties
Curves
Fonts
Colors
Data for X
Data for Y
<unnamed GCS array 1>
Index (1024 data points)
Time (1024 data points)
Actual Position of Axis AXIS:1 (1024 data points)
Actual Position of Axis AXIS:2 (1024 data points)
Actual Position of Axis AXIS:3 (1024 data points)
Actual Position of Axis AXIS:AG2TBL (1024 data points)
Commanded Position of Axis AXIS:1 (1024 data points)
Add to Y Axis
Add to Y2 Axis
Delete All
Data
Label
Show
Style
Y Axis
X: Time / Y: Actual Position of Axis AXIS:1
Actual Position of Axis 1
X: Time / Y: Commanded Position of Axis AXIS:1
Commanded Position of Axis 1
X: Time / Y: Actual Position of Axis AXIS:2
Actual Position of Axis 1
X: Time / Y: Commanded Position of Axis AXIS:2
Commanded Position of Axis 1
Y2 Axis
Title:
Commanded vs. Actual Positions
X Axis: Label:
Time / s
Y Axis: Label:
Position of Axis
Y2 Axis: Label:
Logarithmic
Logarithmic
OK
Cancel


Display Properties
Curves
Fonts
Colors
Fonts
Title:
Arial, 12
Axis label:
Arial, 12
Axis ticks:
Arial, 8
Curve label:
Arial, 8
Reset fonts
OK
Cancel


Display Properties
Curves
Fonts
Colors
Grid:
r:128, g:128, b:128
Color...
Axis:
r: 0, g: 0, b: 0
Color...
Label:
r: 0, g: 0, b: 0
Color...
Plot Area:
r:255, g:255, b:255
Color...
Control Frame:
r:255, g:255, b:255
Color...
Preview:
Graph X/Y
Y Axis
X Axis
0
0,2
0,4
0,6
0,8
1
OK
Cancel

3.6 Piezo Dynamic Tuner

The ***Piezo Dynamic Tuner*** window is opened via the ***Dynamic Tuner*** function in the controller menu or the ***Show dynamic tuner*** function in the ***Tools*** menu. When the window is opened, PIMikroMove reads the current settings from the controller and displays them.



In the ***Piezo Dynamic Tuner*** window you can modify and optimize the control parameters of the piezo servo controller, e.g. in case the load has been changed: Notch filter settings, servo control parameters.

- If you change a parameter value by entering a new value, the value is displayed in a blue font until you press Enter on your keyboard. Pressing Enter sends the value to the controller and changes the font color from blue to black. If the values of a parameter are different in the volatile and non-volatile memory of the controller, the parameter field is highlighted.
- When the ***Notch Frequency 1*** value is set in the ***Parameter Settings*** panel of the ***Piezo Dynamic Tuner*** window, the ***Servo-Loop I-Term*** value can be set to be adjusted

accordingly. The adjustment depends on the option selected from the ***Automatic I-Term calculation*** drop-down menu.

- The settings for slew rate (***Slew Rate / Velocity*** field) and record table rate (***Record Rate*** field) can be changed in the ***Piezo Dynamic Tuner*** window. Entering new values in these fields changes the values of the corresponding parameters in the volatile memory of the controller: ***Servo Loop Slew-Rate*** (ID 0x07000200) or ***Open Loop Slew-Rate*** (ID 0x07000201) parameter, depending on the current operating mode (open-loop or closed-loop operation); ***Data Recorder Table Rate*** (ID 0x16000000). The values are not saved or reset when you use the ***Save as Default (EEPROM)*** and ***Reset All to Default*** buttons in the ***Parameter Settings*** panel of the ***Piezo Dynamic Tuner*** window.

To configure the data display in the graphics pane and perform operations with the data use the window's toolbar buttons (p. 68).

3.6.1 Adjusting the Notch Filter(s)

The corrections by a notch filter only take place in closed-loop operation by default, but can also be enabled for open-loop operation. The appropriate frequency component is reduced in the control value to compensate for undesired resonances in the mechanical system. Adjusting the notch filter frequency can be useful, particularly in the case of very high loads.

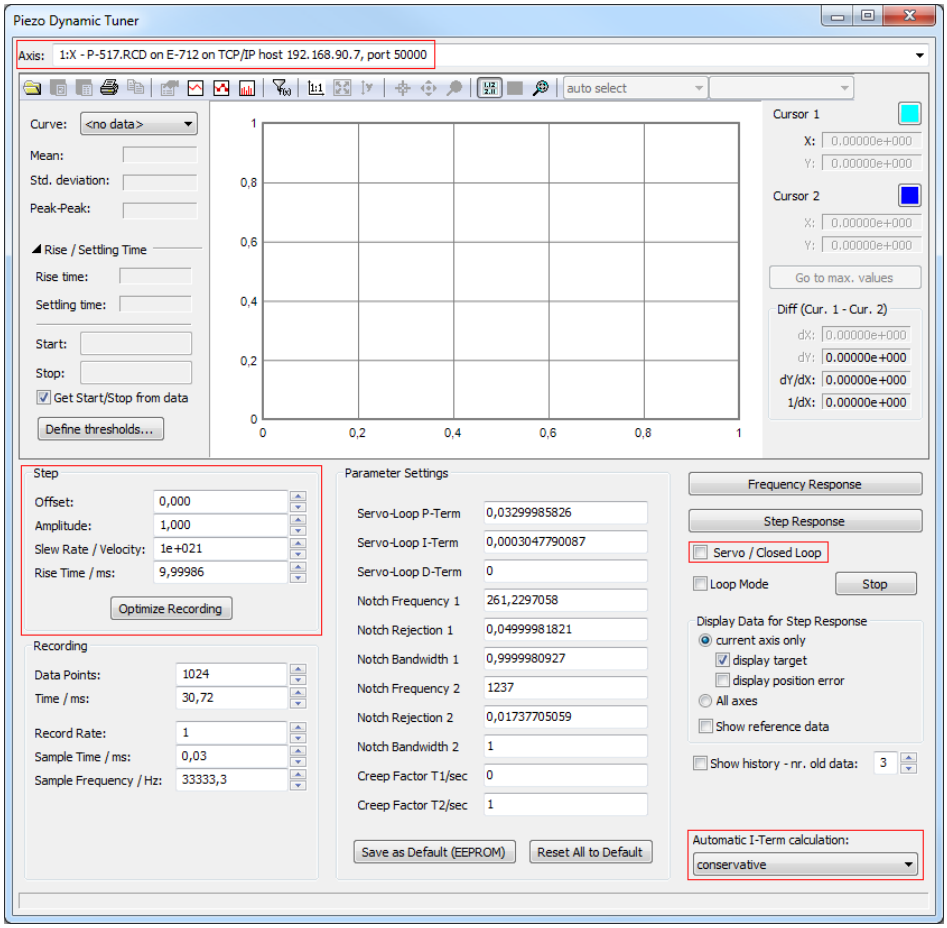
INFORMATION

The notch rejection value, which scales the damping done by the notch filter, should always be 0.05. A notch rejection value of 1 deactivates the notch filter.

Before you measure the resonant frequencies as described below, make sure that the notch filters are not enabled in open-loop operation. To do this, check the value of the ***Enable Notch In Open Loop*** parameter (ID 0x08000500) for all axes (0 = disable notch filter in open-loop operation; 1 = enable notch filter in open-loop operation). You can do this in the servo parameter groups of the ***Device Parameter Configuration*** window (p. 144).

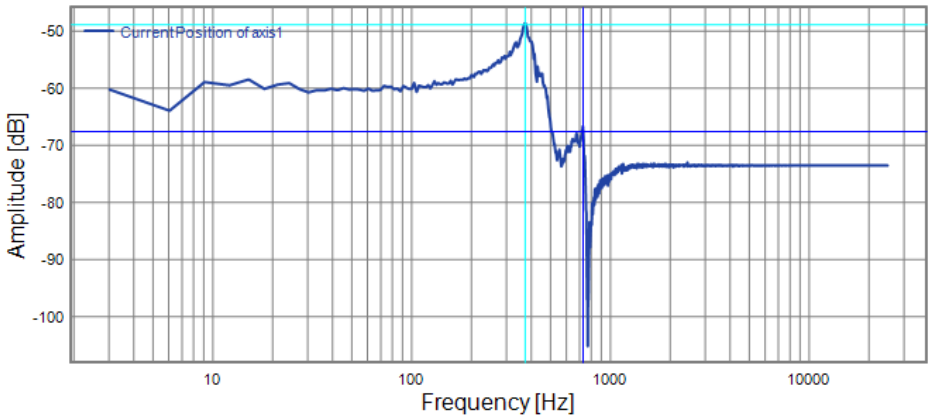
To measure the resonant frequency and adjust the notch filter(s), a frequency response (axis response to an impulse) is recorded in open-loop operation:

1. Configure the frequency response:
   a) Make sure the correct axis is selected (***Axis*** drop-down list).
   b) In the fields of the ***Step*** panel, specify suitable values for the start value (***Offset***) and amplitude (***Amplitude***) of the impulse. The start value should be 0, and the amplitude should be about 10 % of the axis travel range.
   c) Make sure the axis is in open-loop operation (***Servo / Closed Loop*** box is not checked).
   d) Via the drop-down menu ***Automatic I-Term calculation*** select if and how the Servo-Loop I-Term is to be adjusted automatically when Notch Frequency 1 is changed.



2. Perform the frequency response measurement by clicking the ***Frequency Response*** button.
3. Identify the resonant frequency in the Piezo Dynamic Tuner window:
   a) Click the button to display cursors, then click to enable cursor motion.
   b) Identify the resonance peak(s) in the FFT display. To do so, place a cursor on the peak and read out the cursor value which is displayed on the right hand side of the graph. If there is more than one resonance peak, peak 1 is always the one with the lowest frequency.
   In the figure below, cursor 1 is at the first resonance peak (372.31445 Hz), and cursor 2 is at the second (next higher) resonance peak (729.37012 Hz).
4. If necessary, adjust the notch filter settings in the ***Parameter Settings*** panel to the measured resonant frequencies (adjustment is necessary if the values significantly differ).
   a) Enter the frequency value of the first resonance peak in the Notch Frequency 1 field (in Hz). You can either right-click the field with the mouse and select the value from a menu, or type the value in the field. Note that depending on the selection for Automatic I-Term calculation, the Servo-Loop I-Term value is changed as well when you change the Notch Frequency 1 value.
   b) If you have measured a second resonance peak, enter the frequency value of the second resonance peak in the Notch Frequency 2 field (in Hz). If the second notch

filter is deactivated (rejection value = 1), change the rejection value to 0.05 in the Notch Rejection 2 field.



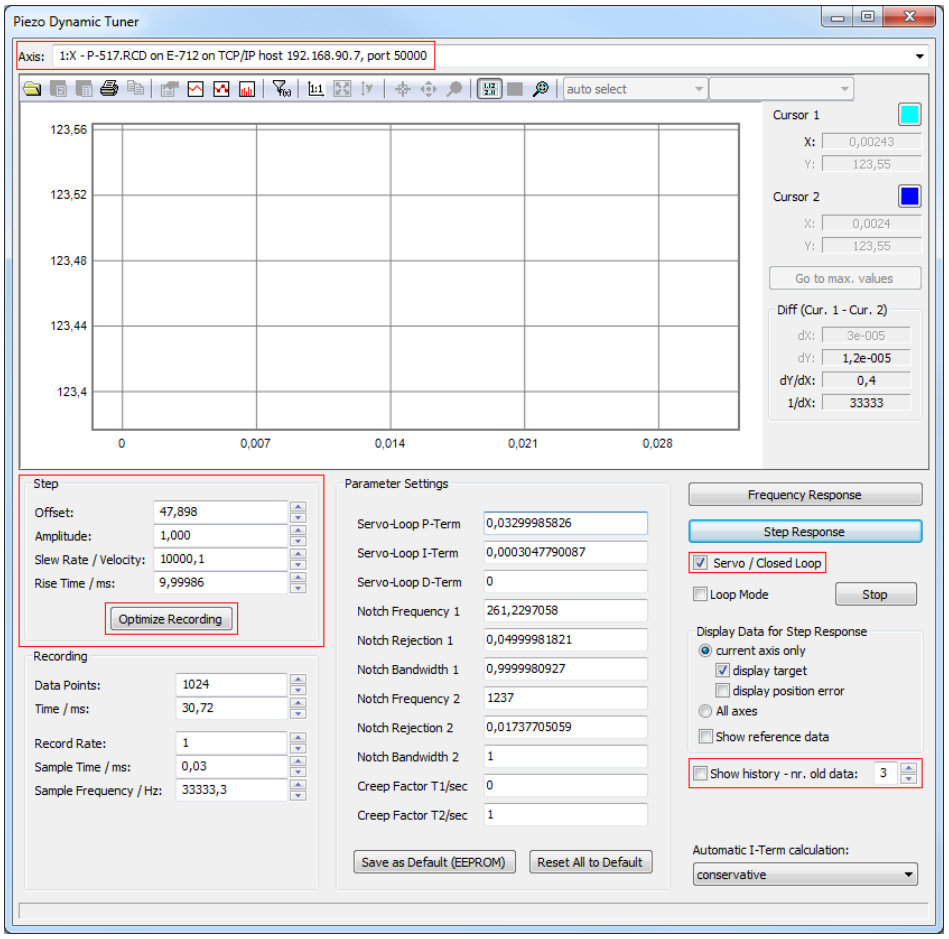
5. Save or discard the new settings in the ***Parameter Settings*** panel:
   - If you want to keep the new settings, save them to the non-volatile memory of the controller by clicking the ***Save as Default (EEPROM)*** button.
   - If you want to discard the new settings and reset the parameter values to their defaults (i.e., to their values from the non-volatile memory), click the ***Reset All to Default*** button.

3.6.2 Checking and Optimizing Servo Control Parameters

Adjusting the servo control parameters optimizes the dynamic properties of the system (overshoot and settling time). The optimum settings depend on your application and your requirements.

The optimization of the servo control parameters is typically done empirically: The response of the axes to a step, the step response, is analyzed under various values in closed-loop operation:

1. Configure the step response:
   a) Make sure the correct axis is selected (***Axis*** drop-down list).
   b) Make sure the axis is in closed-loop operation (***Servo / Closed Loop*** box is checked).
   c) In the fields of the ***Step*** panel, specify suitable values for the start value (***Offset***) and amplitude (***Amplitude***) of the step. The start value should be 0, and the amplitude should be about 10 % of the axis travel range.
   d) By clicking the ***Optimize Recording*** button in the ***Step*** panel, optimize the number of data recorder points that will be read from the controller when the step response has been performed.
   e) If you want to compare the results of multiple step resonse measurements, check the ***Show history*** box and select the number of old recordings to be displayed.

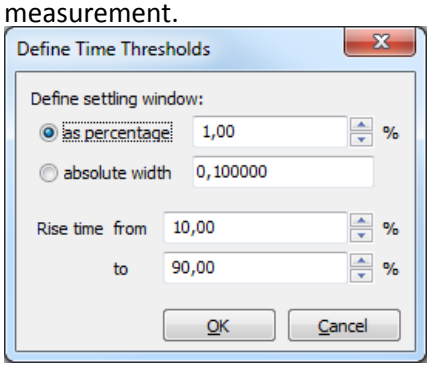


2. Perform and analyze the step response measurement:

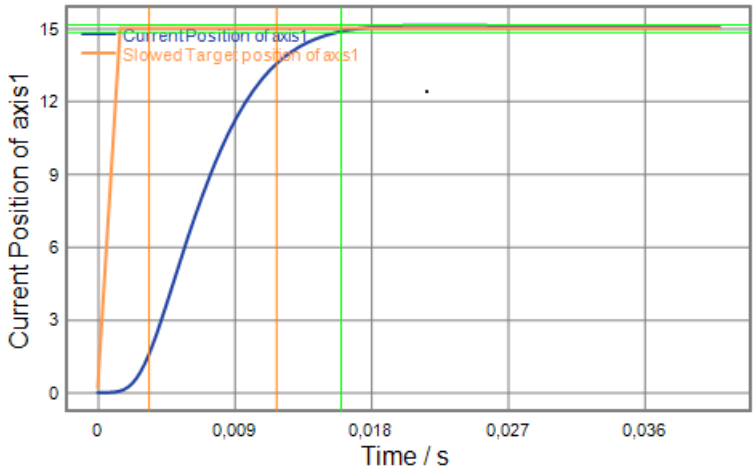
   a) Optional: Check the ***Loop Mode*** box to move the axis in a permanent loop. The loop mode is useful if you want to do the adjustment of the servo control parameters during motion. (The loop motion can be stopped at any time by clicking the ***Stop*** button.)

   b) Start the step response by clicking the ***Step Response*** button.

   c) Optional: Click the ***Define thresholds…*** button to open the ***Define Time Thresholds*** window. In the ***Define Time Thresholds*** window, you can adjust the thresholds which are used by the ***Piezo Dynamic Tuner*** window to calculate and display the rise and settling times of the axis, based on the recorded step response measurement.



d) Check the step response result and compare it with the examples shown in the examples below. The result of the step response is satisfactory when there is minimum overshoot, and the settling time is not too long, as in the figure:

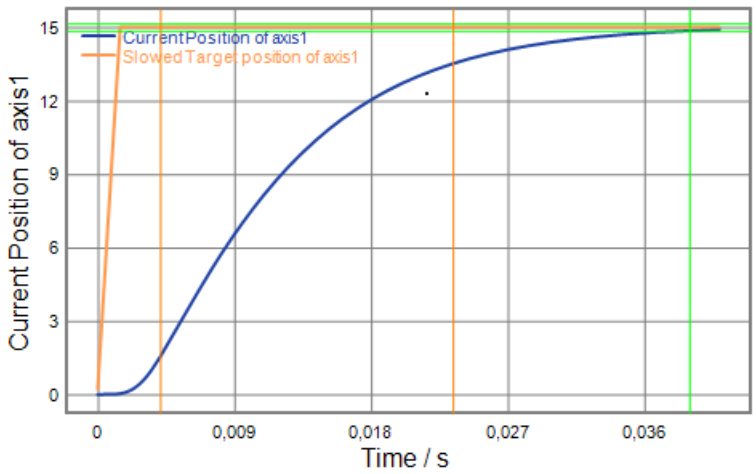


As long as the result of the step response is not satisfactory, adjust the servo loop parameters and start a new step response.

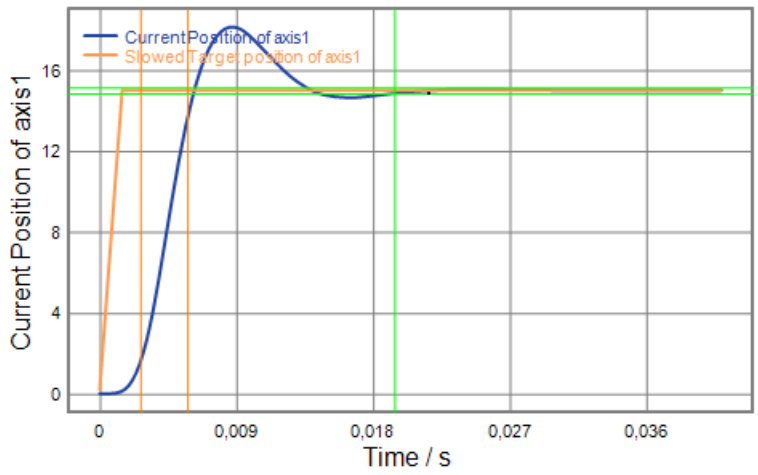
3. Save or discard the new settings in the ***Parameter Settings*** panel:
   - If you want to keep the new settings, save them to the non-volatile memory of the controller by clicking the ***Save as Default (EEPROM)*** button.
   - If you want to discard the new settings and reset the parameter values to their defaults (i.e., to their values from the non-volatile memory), click the ***Reset All to Default*** button.

Examples for step response results:

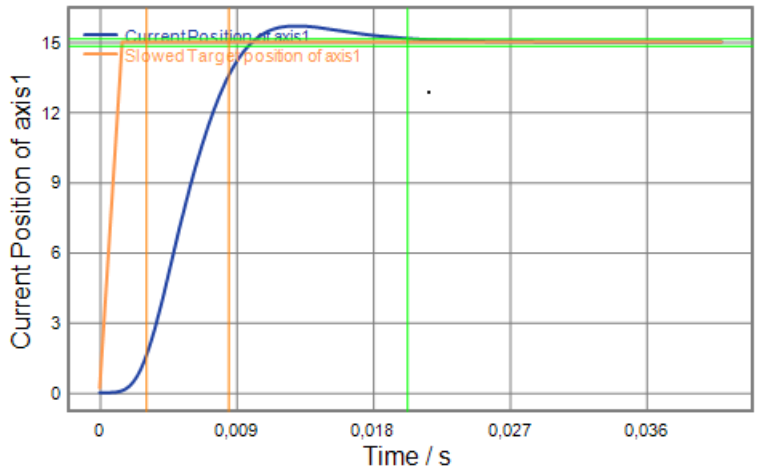
The rise rate of the step response is very low in the figure below. This means that the P term is too low and has to be increased.



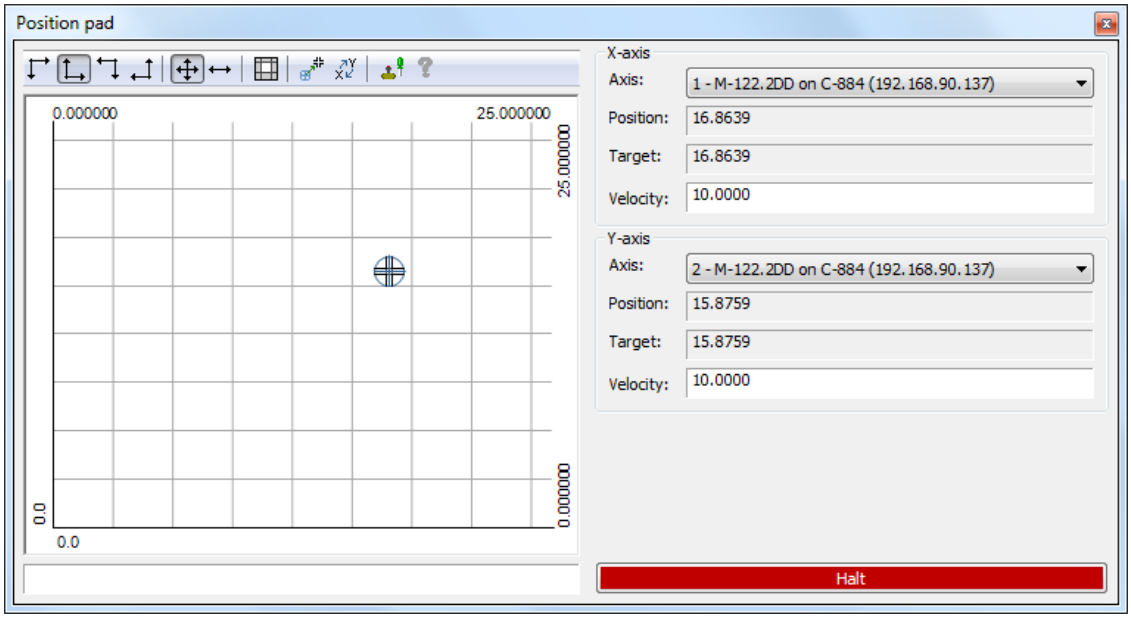
The figure below shows a step response with a high overshoot which means that the P term is too high and has to be decreased.



The figure below shows a step response with a small overshoot which means that the P term is still too high and has to be decreased.



3.7 Position Pad



INFORMATION

This window is only available when the following conditions are fulfilled for at least one of the connected axes:

- Absolute positions can be commanded (servo must be on, and axes equipped with incremental sensors must be in the referenced state).
- The travel range is known.

With the position pad you can control the position of two axes with the mouse. Positions are indicated on the pad by a cross: The black cross marks the target position, the blue cross marks the current position. You can click or drag the target position with the mouse.

INFORMATION

An HID which is directly connected to the controller(s) cannot be used in the ***Position Pad*** window.

The left-hand side of the ***Position Pad*** window shows the toolbar and the position grid. On the right-hand side you can choose which two axes to control and see their position, target and velocity settings. To change the maximum velocity, use the corresponding single-axis windows or the Axes tab card (p. 25).

INFORMATION

A ***Position Pad*** window can be docked to the bottom border of the main window by dragging it with the left mouse button pressed (see "Main Window" (p. 9)).

3.7.1 Position Pad Toolbar

The first four buttons are mutually exclusive and specify the orientation of the coordinate system.

2D and ***1D*** allow switching between one or two active axes. With 1D it is possible to control a single axis with the joystick x-axis without any effect on any other axis.

Limit range can limit the displayed range of the Position Pad. The targets for velocity control are also set to these limited values.

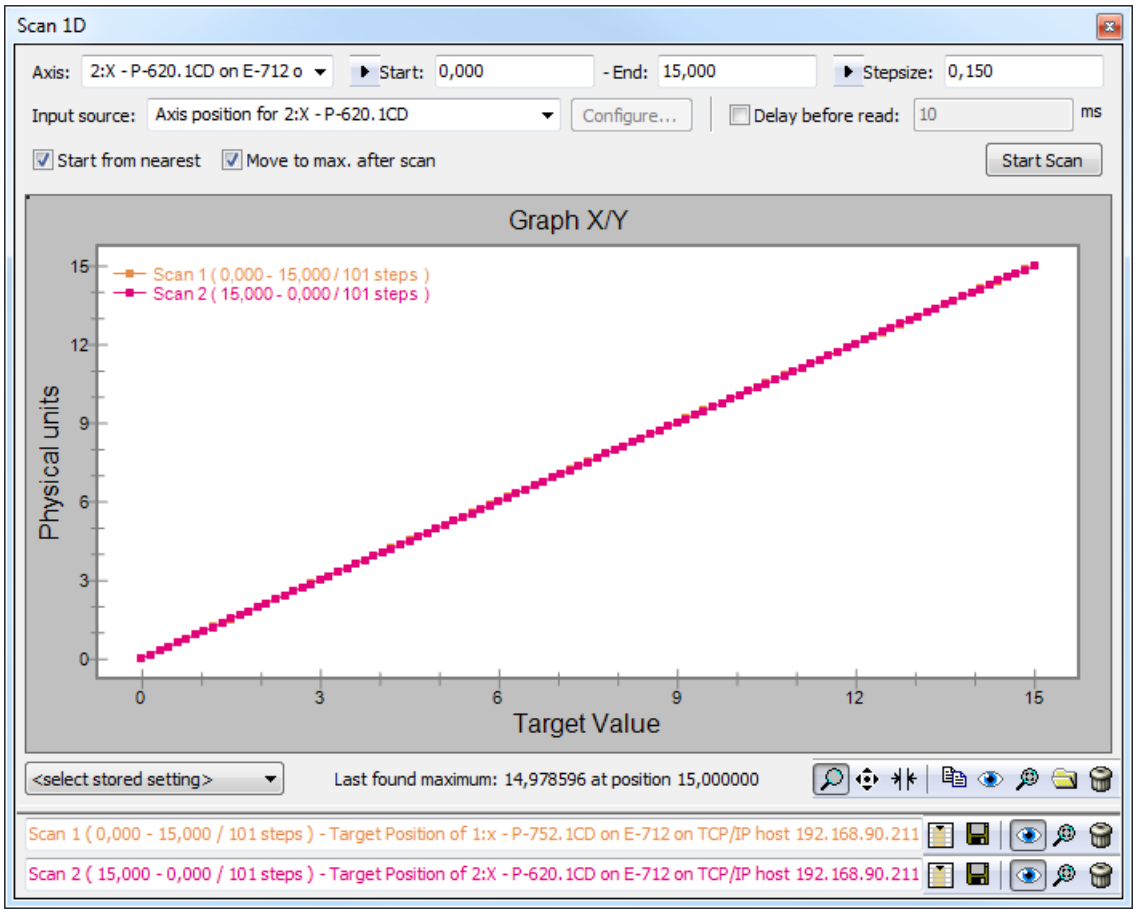
When ***Pseudo-vector moves*** is activated, the velocities are set so that both axes arrive at the target at approximately the same.

With ***Swap X and Y axes*** you can swap the display position of the axes.

Configure Joystick Buttons *is only available when the operating system has recognized an HID.*
This function opens the ***Configure PC HID Control*** (p. 93) dialog where you can configure the actions that will be performed when one of the joystick buttons is pressed.

Open help opens the online help for the program.

3.8 Scan 1D



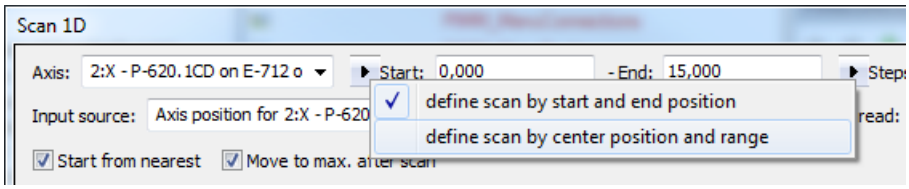
With the ***Scan 1D*** window you can measure an input source while moving an axis. Moved axis and input source need not be controlled by the same controller. So it will be necessary for the axis to stop after each step, since synchronization can not be assured otherwise.

The measured data is visualized and can be saved to a file on the host PC.

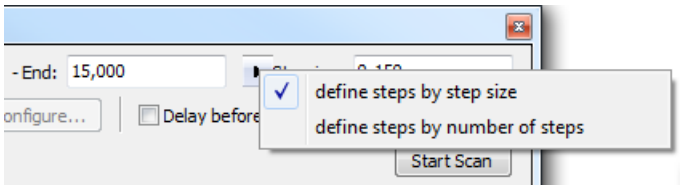
In the ***Axis*** field you can select the axis to be moved, and in the ***Input source*** field the input source to be measured. Possible input sources are:

- Analog inputs
- Current positions of the connected axes
- Target positions of the connected axes
- Input channels
- Responses to query commands sent by PIMikroMove
  To select a command response for the input source, use the ***Value of single command sent to controller…*** entry in the list of possible input sources. Then specify the command to be sent via the ***Configure…*** button next to the ***Input source*** field.

To define the scanning range, you can give its start and end positions or its center position and a range value: Click on the arrow button for the range settings to open a context menu and select the desired option.



The steps to be performed between the single measurement points can be defined by the step size or by the number of steps: Click on the arrow button for the steps to open a context menu and select the desired option.



Furthermore, you have the following configuration options for motion and measurement:

- To reduce the time required to move to the first position, you check ***Start from nearest***. Note that if you repeat the process, the axis may then move back and forth.
- After scanning, you can move the axis back to the position where the input source took its maximum value if you check ***Move to max. after scan***. If the scanning range is defined via its center position and a range value, you can also move the axis back to the center position if you select ***Move to … center after scan***.
- If ***Delay before read*** is checked, the software will wait the specified time (in milliseconds) after the axis has completed one step and before a new value of the input source is measured.

You can save the current configuration settings or reload and delete stored settings via the selection field below the graphics pane.

To start scan motion and measurement, click the ***Start Scan*** button.

The central pane of the window shows the measured scan data in a 2D graph. Below this graphics pane, the individual scans are listed and can be administered separately. The list contains all scans which were performed successfully since you had opened the ***Scan 1D*** window. Every new scan is also added to the graphics pane and can be hidden or shown again via its list entry.

Below the graphics pane, there is a common graph toolbar which applies to all scan data:

In addition to the common graph toolbar, every line of the scan list has its own toolbar for administering this scan data separately:

Zoom
If you activate this function, you can magnify a selected area of the graphics pane. To do this, drag the mouse diagonally across the desired area with the left mouse button pressed. To reset the display to its original state use the ***Zoom to fit*** button or right-click anywhere inside the display.

Pan
If you activate this function, you can scroll the content of the graphics pane by placing the mouse pointer in the graphics pane and dragging it with the left mouse button pressed. To reset the display to its original state use the ***Zoom to fit*** button.

Select range
Click this button to select a new scanning range with the mouse.

Copy graph image to clipboard
You can copy the graph image to the clipboard from where it can be pasted to another application.

Show all
Shows all scan data.

Zoom to fit
In case of a zoomed or shifted display, this function resets the graphics pane to its original state, so that all the data is completely shown.

Load scan data
Loads scan data which was saved before in GCS array format (see the GCS Array Manual, SM146E, for details).

Delete all
Deletes all scan data.

In addition to the common graph toolbar, every line of the scan list has its own toolbar for administering this scan data separately:

View data
Shows a window displaying the array data for this scan, i.e., a table with positions and measured values.

Save data
Exports data to disk in CSV or GCS array format (see the GCS Array Manual, SM146E, for details).

Show data
Shows or hides the scan data from the 2D graphics display.

Zoom to fit
Zooms the display so that the data is completely shown.

Delete this data
Deletes only this data.

3.9 Scan 2D

With the ***Scan 2D*** window, you can measure an input source while moving two axes. Moved axes and input source need not be controlled by the same controller. So it will be necessary for the axes to stop after each step, since synchronization cannot be assured otherwise.

Possible input sources are:

- Analog inputs

- Current positions of the connected axes
- Target positions of the connected axes
- Input channels
- Responses to query commands sent by PIMikroMove

Two tab cards in the ***Scan 2D*** window offer the following scan functionality:

- ***Scan*** tab card: Data is measured during motion, visualized and can be saved to a file on the host PC. See "Scan Tab Card" (p. 90) for details.
- ***Auto Find*** tab card: Data is measured during motion, and the Auto Find procedure tries to find the maximum of the measured input source signal. See "Auto Find Tab Card" (p. 92) for details.

Scan configuration

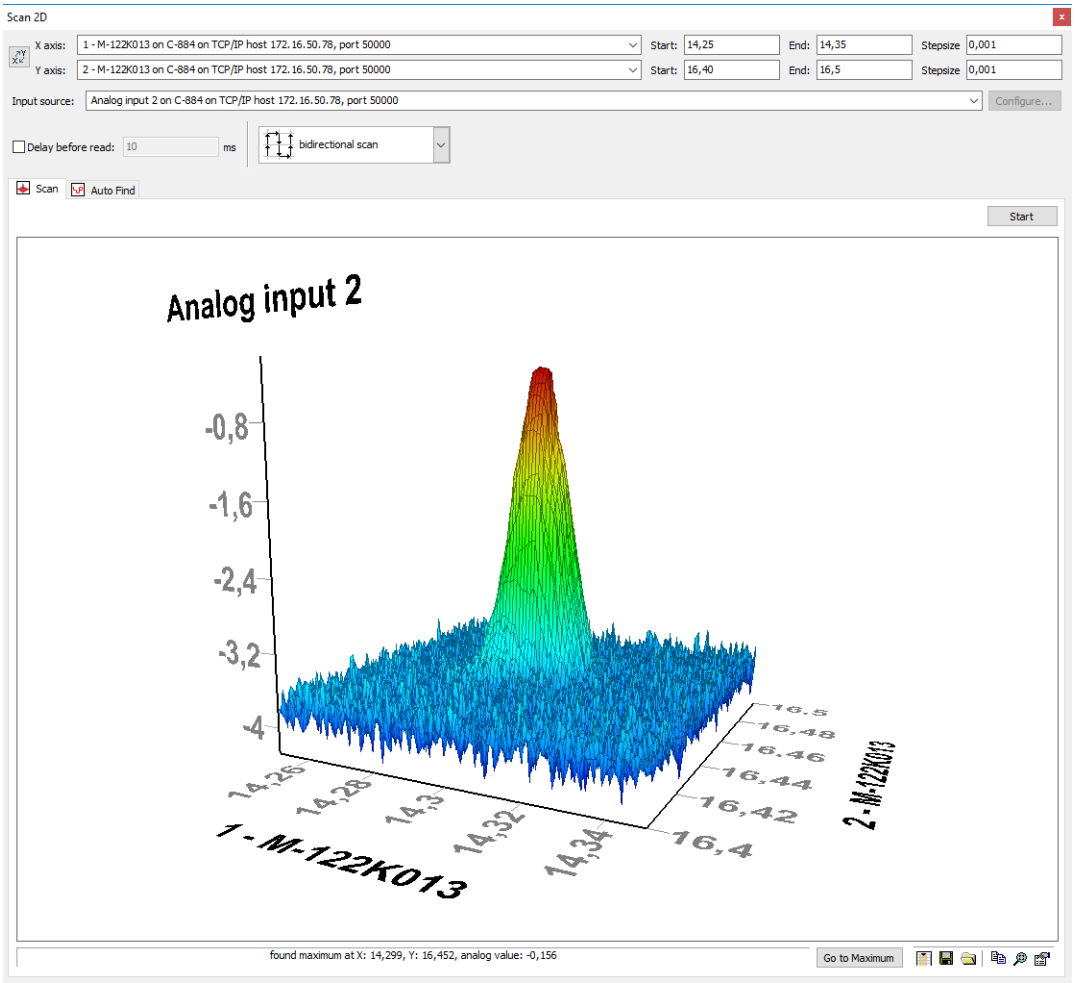
How to configure the scan motion to be started:

1. In the ***X axis*** and ***Y axis*** fields select the axes to be moved.
2. The range to be scanned is given by the position values in the ***Start*** and ***End*** fields. The size of the steps made between the single measurement points is given by the ***Stepsize*** field. Type suitable values in these fields.
3. Select the input source to be measured in the ***Input source*** field. To select a command response for the input source, use the ***Value of single command sent to controller…*** entry. Then specify the command to be sent via the ***Configure…*** button next to the ***Input source*** field.
4. Optionally: Check ***Delay before read***, if you want the software to wait the specified time (in milliseconds) after the two axes have completed their steps and before a new value of the input source is measured
5. Via the drop-down list for the direction of the scan, select whether to perform a bidirectional or unidirectional scan.

When the scan motion is configured, select the ***Scan*** tab card or the ***Auto Find*** tab card and start the scan motion as described below in the corresponding section.

3.9.1 Scan Tab Card

If you start the scan from the ***Scan*** tab card, data is measured during motion, visualized and can be saved to a file on the host PC.



Make sure the scan motion configuration (p. 88) is correct (axes, input source, range and step values). To start scan motion and measurement, click the ***Start*** button.

The central pane of the ***Scan*** tab card shows the measured scan data in a 3D graph. Every new scan overwrites the content of the graphics pane.

You have the following options for mouse control in the graphics pane:

- Drag with the left mouse button pressed: rotate view
- Drag with the left mouse button + SHIFT pressed: shift view
- Drag with the right mouse button pressed: zoom

Below the graphics pane, there is a toolbar with which you can administer the scan data:

Use the ***Go to Maximum*** button to move the axis to the position where the input source took its maximum value.

View data
Shows a window displaying the array data for this scan, i.e. a table with positions and measured values

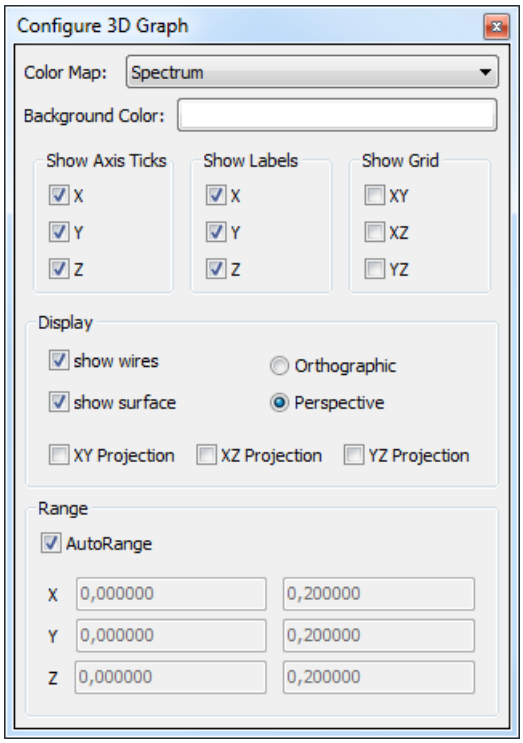
Save data
Exports data in CSV or GCS ARRAY format (see the GCS Array Manual, SM146E, for details)

Load data
Loads scan data which was saved before in GCS ARRAY format

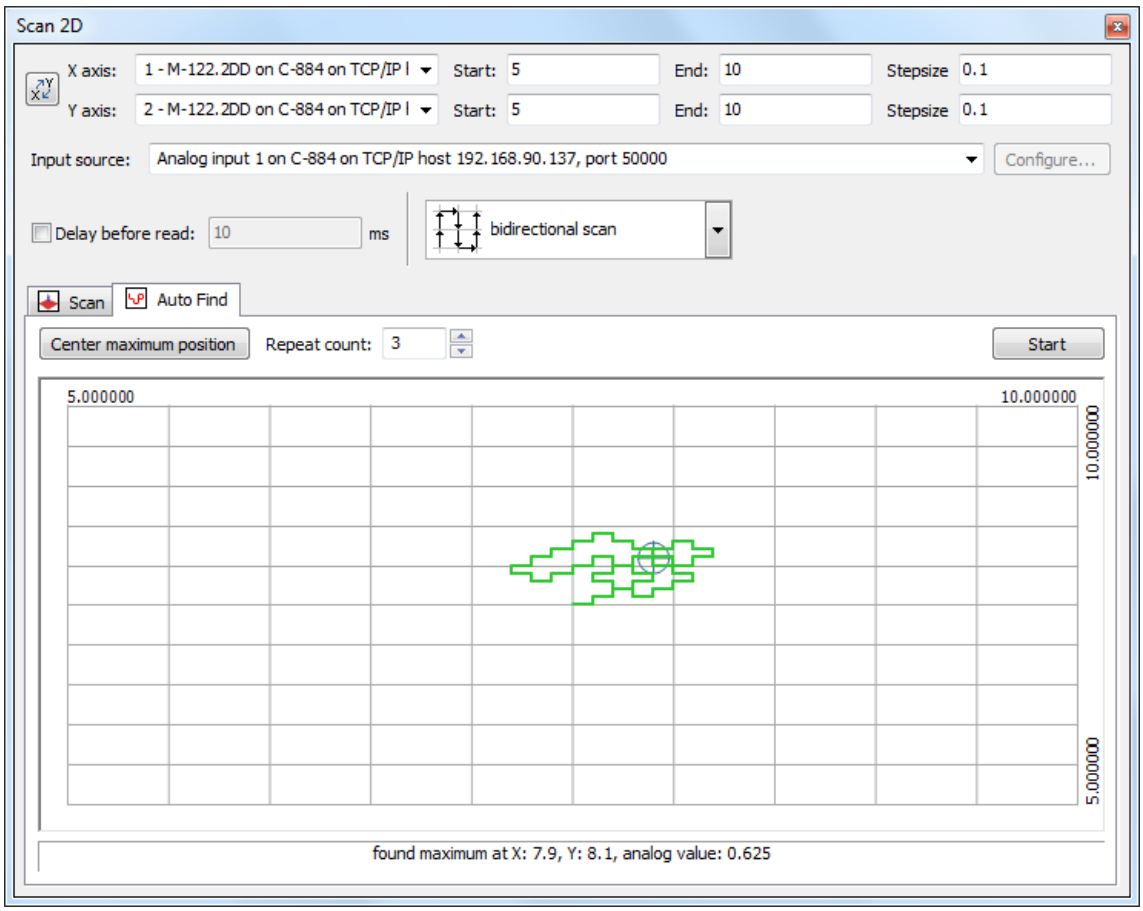
Copy graph image to clipboard
Copies a bitmap of the display in the graphics pane to the clipboard

Zoom to fit
Zooms so that the data is completely shown

Configure display properties
Opens the ***Configure 3D Graph*** dialog, where several settings for the display can be changed:



3.9.2 Auto Find Tab Card



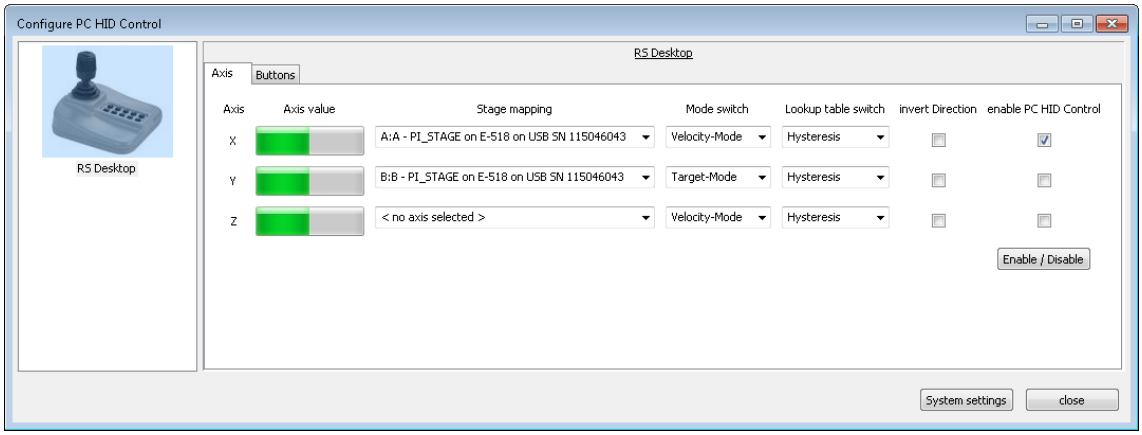
If you start the scan from the ***Auto Find*** tab card, the Auto Find procedure tries to find the maximum of an intensity signal by modifying the position of two axes. The mechanics is moved one step and the measured intensity is compared with the last value. If the intensity decreases the next move will be in the opposite direction. If the mechanics reaches the same position a specified number of times, the Auto Find procedure will stop.

Make sure that the scan motion configuration (p. 88) is correct (axes, input source, range and step values). The Auto Find procedure will start in the middle of the defined range. In the ***Repeat count*** field, set the number of times the mechanics has to reach the same position before Auto Find stops. To start the Auto Find procedure, click the ***Start*** button.

The central pane of the ***Auto Find*** tab card shows the scan motion path in a 2D graph. Every new Auto Find procedure overwrites the content of this graphics pane.

3.10 Configure PC HID Control

In the ***Configure PC HID Control*** window, you can assign axes of HIDs connected to the PC to axes of connected controllers. Up to two PC HIDs, e.g. game controllers or joysticks, can be used.

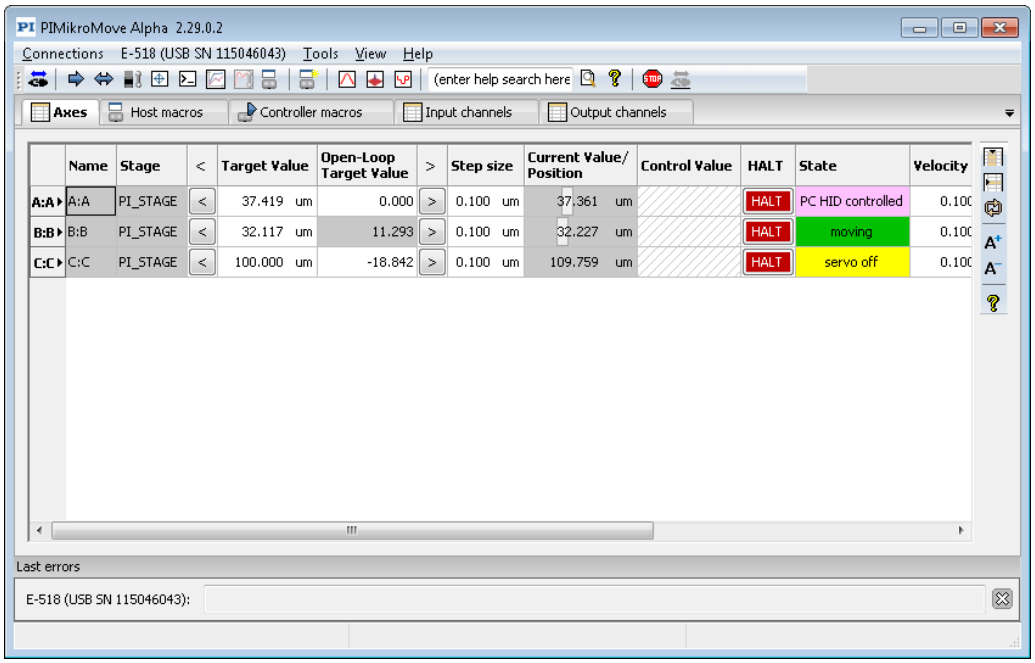


The connected PC HIDs are listed in the left pane of the window. Click on an entry to select it for configuration: The axes of the selected PC HID are shown on the tab ***Axis***, its buttons on the tab ***Buttons***.

To configure the PC HID for the control of motion variables of a connected controller proceed as follows:

- Configure the PC HID axes on the ***Axis*** tab. For each PC HID axis you want to use:
  - Select a controller axis from the ***Stage mapping*** drop-down list to assign it to the PC HID axis.
  - Select the control mode (controller-dependent) in the field ***Mode switch***.
  - Select the type of lookup table in the field ***Lookup table switch***.
  - If the direction of motion is to be inverted during HID control, check the ***invert Direction*** checkbox.
  - Activate PC HID control for the axis by checking the ***enable PC HID Control*** checkbox. To activate/deactivate all PC HID axes, click the ***Enable / Disable*** button.

A controller axis for which PC HID control is active is displayed in the ***Axes*** tab card with the state "PC HID controlled":

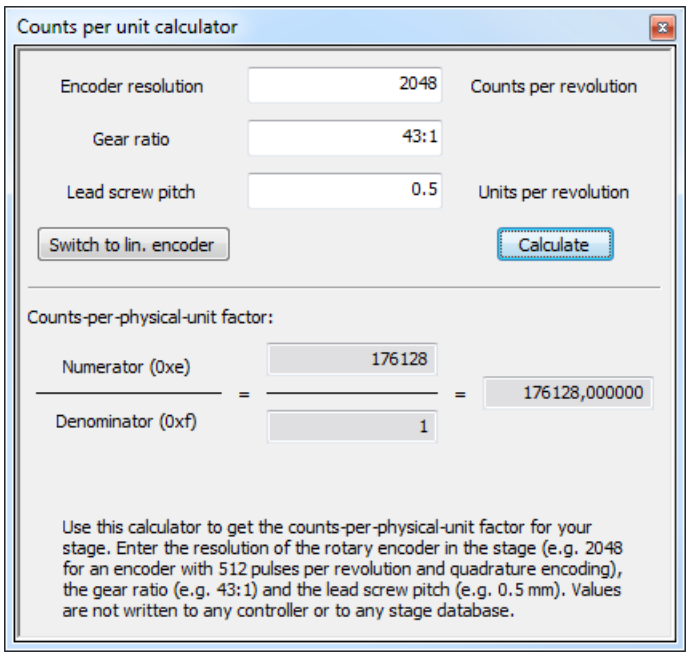


- Configure the PC HID buttons on the ***Buttons*** tab. For each PC HID button you want to use:
  - Select an option from the ***Function mapping*** drop-down list to assign it the PC HID button:
    - ***halt*** will stop both axes and disable any motion as long as the button is pressed
    - ***do nothing*** ignores that the button is pressed
    - ***nnn% velocity while button down*** sets the velocity to the specified percent of the currently set velocity, but restores the old value when the button is released - only in velocity control mode!
    - ***1D*** disables the Y-axis, only the X-axis will be active
    - ***2D*** enable both, X- and Y-axis
    - ***store position*** stores current position
    - ***move to stored position*** moves to the last stored position
    - ***button preset X-Y position—long: store, short: goto*** a long press stores the current positions for X and Y (if 2D is enabled). The positions are confirmed by a short beep and the cursor will be drawn with a thicker outline for a short time. A short press will move to these stored positions.
    - ***button preset X position—long: store, short: goto*** same as above, only X position is stored
    - ***button preset Y position—long: store, short: goto*** same as above, only Y position is stored

To test and calibrate the PC HID proceed as follows:

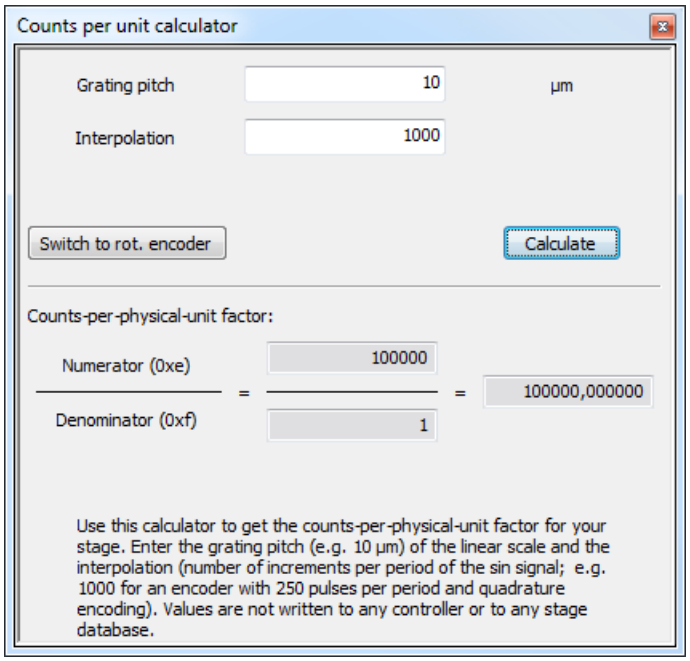
- Move its axes and for each axis check if the corresponding ***Axis value*** is displayed correctly on the ***Axis*** tab.
- Press its buttons and for each button check if the corresponding ***Button state*** is displayed correctly on the ***Buttons*** tab.
- If it is necessary to calibrate the PC HID, click the ***System settings*** button to open the operating system's HID control panel. Follow the calibration instructions there.

3.11 Counts per unit calculator



Counts per unit calculator for rotary encoders

This calculator window may be helpful when you start working with a stage that is equipped with incremental encoder and not part of any stage database yet. Using this window, you can calculate the values for the numerator and the denominator of the counts-per-physical-units factor (parameters 0xE and 0xF) for your stage. For stages with incremental sensors, the counts-per-physical-unit factor determines the unit of length to be used for all closed-loop motion commands (the controllers internally use counts for motion command processing).



Counts per unit calculator for linear encoders

INFORMATION

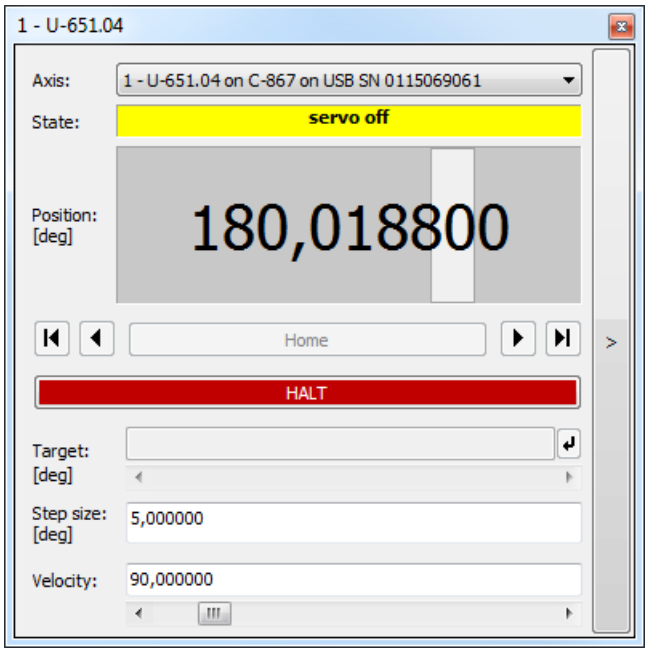
The values from the calculator window are not sent to the controller or to any stage database. If you want to use these values, make a note of them. Then enter them in the corresponding columns on the ***Axes*** tab card. To create an new stage database entry for your stage, you can use the ***Save parameters as User Stage type...*** function in the controller menu (p. 12).

When you change the numerator and denominator of the counts-per-physical-unit factor, all other parameters whose unit is based on the unit of length must be adapted too, e.g. closed-loop velocity and parameters regarding the travel range. Some controllers adapt the appropriate parameters automatically.

3.12 Single-Axis Window

A ***single-axis*** window provides controls for a single axis.

To open a ***single-axis*** window, use the ***Single Axis Window*** function in the ***View*** menu or the corresponding function of the ***Axis*** context menu on the ***Axes*** tab card (p. 25).



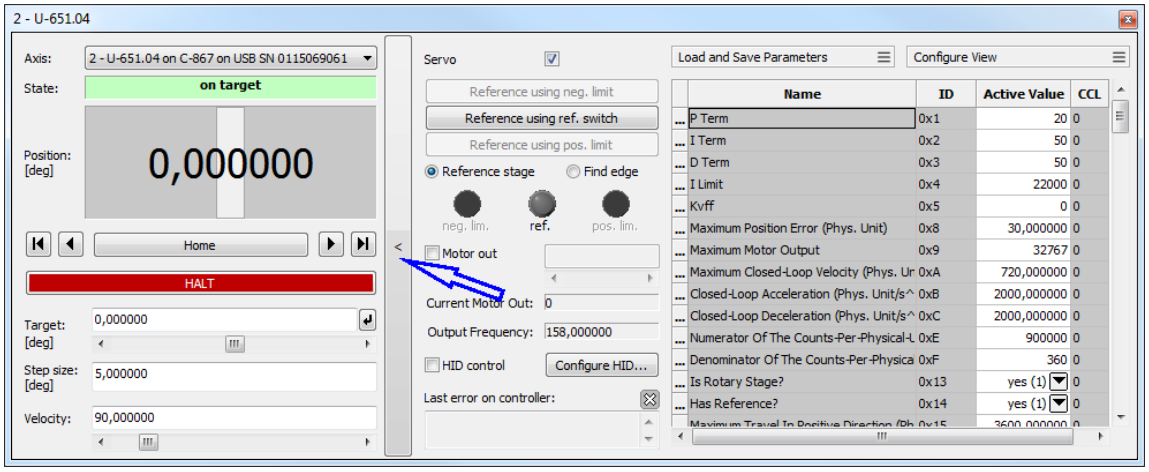
In the ***single-axis*** window you can enter target position, velocity and step size for closed-loop operation (servo on). If supported by the controller, a target value can also be set for open-loop operation (servo off). The target value in open-loop operation is to be interpreted as follows:

- ***OL Target***: is the open-loop target position for axes driven by PiezoWalk® drives (controller supports the OMA command)
- ***OL Value***: is an dimensionless open-loop value for axes driven by piezo actuators (controller supports SVA command). Depending on the controller it may correspond to a position or to a voltage (approximately).

If you right-click on any free area in the window you can see the ***Axis*** menu (p. 29) for the axis displayed in the window.

If supported for the controller, a ***single-axis*** window can be expanded by clicking the vertical **>** button. To collapse an expanded window, click the **<** button between the left-hand and the center pane (see picture below). As long as it is not expanded, a ***single-axis*** window can be docked to the left or right border of the main window by dragging it with the left mouse button pressed. Docked ***single-axis*** windows cannot be expanded.

If expanded, the ***single-axis*** window shows two additional panes.



Center pane

In the center pane, you can

- Switch servo on/off using the appropriate checkbox
- Initiate a referencing procedure:
  If the ***Reference stage*** and ***Find Edge*** radio buttons are present (depends on controller), make sure that ***Reference stage*** is selected. To start the reference move, click one of the ***Reference using…*** buttons (see also "Referencing" (p. 175)). If the axis does not support referencing or a given type of referencing (e.g. has no reference switch), the respective button(s) will be dimmed.
- Initiate a "find edge" motion (availability depends on controller) which can help to detect or check the travel range of the mechanics:
  Select the ***Find Edge*** radio button. To start the "find edge" motion, click one of the ***Find …*** buttons. The axis then moves to the corresponding switch (limit or reference). The reference state of the axis remains unchanged.
- Check the state of limit and reference switches using the appropriate LEDs. If the axis is not equipped with such switches, the respective LEDs are displayed as disabled.
- Set the output voltage to the motor directly:
  Activate the ***Motor Out*** checkbox (servo is then switched off automatically). Enter a value in the field next to the checkbox or use the slider below the field. 32767 is equivalent to 100% output voltage in positive direction and should cause a rapid move to the positive end of the travel range (appropriate value for negative direction is -32767).
  If the axis does not support setting the motor output directly, the checkbox and the appropriate controls are disabled.
- If supported by the controller, the last valid control value of the axis is shown in the field ***Current Motor Out***.
- For PILine® controllers, the ***Output Frequency*** field in addition shows the frequency of the piezo voltage in kHz (set via parameter 0x51).

NOTICE

Damage due to crashes!

Limit switches may not be effective when the motor ouput is set directly, so that the stage may run into the hard stop. Rapid motion may occur which might cause damage to the loads connected to the stage.

➢ Set the motor output directly only for testing purposes.

- Activate ***HID control*** for the axis, if supported by the controller. The configuration dialog for an HID connected to the controller can be called directly from here via the button ***Configure HID...***.
- Check the last controller error in the display below ***Last error on controller***. After an error occurred, the display is highlighted in red for one second. Then the display color changes to gray. This recurs with every new error (the former error code is overwritten). You can clear the display using the ***x*** button in its top right corner. Note that errors which are provoked by incorrect use of the controls in PIMikroMove will be displayed in separate message windows.

Right-hand pane

There may be some parameters of the stage or controller which often have to be checked or changed. For that purpose, the rightmost pane of the expanded ***Single Axis*** window provides a parameter list. See below for details.

3.12.1 Changing Parameter Values

To check or change parameters of stage or controller, you can use the parameter list in the right-hand pane of the expanded single-axis window.

Configure the parameter list:

You can configure the content of the parameter list using the ***Configure View -> Select parameters…*** function which opens the same dialog as used for the column selection in the PIMikroMove main window (see "Select Columns to be displayed" (p. 26) for details). It is possible to save or load certain list configurations under user-defined names via the list box in the top right corner.

Edit parameter values:

Type new values in the parameter fields. By leaving the field or pressing Enter the value is sent to the controller's volatile memory.

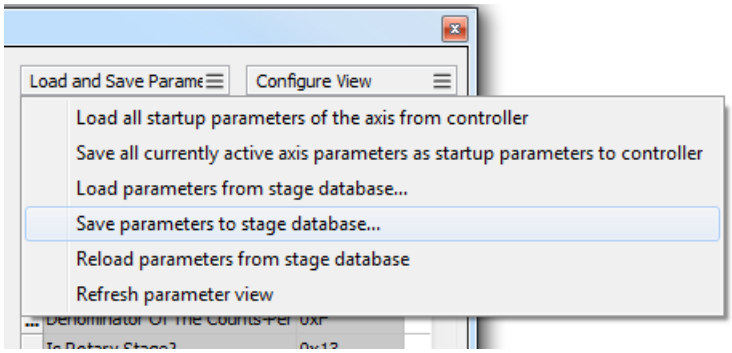
Depending on the controller, parameters may be write-protected by their command level setting. A padlock symbol indicates that a parameter is write protected, and changing the parameter may require the command control level (CCL) 1. Clicking on the parameter will open the password dialog where you can select a command level and enter the corresponding password. The password for command control level 1 is "advanced". Note that command levels higher than 1 are reserved for PI service personnel.

Save parameter changes:

If stage databases on the host PC are supported for the controller, the ***Save parameters to stage database…*** functionality is accessible from the single-axis window. With this function the current stage parameter values can be stored as a new user-defined stage type in the stage database. If there is already a stage type of the same name in the stage database, its settings will be overwritten when the new ones are saved.

Proceed as follows:

1. Click on ***Load and Save Parameters -> Save parameters to stage database...***.

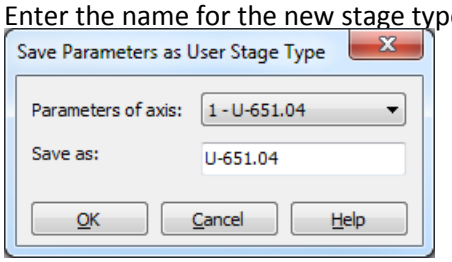


   The ***Save Parameters as User Stage Type*** dialog opens.

2. In the ***Save Parameters as User Stage Type*** dialog, save the modified parameter values as a new stage type:

   a) Leave the entry in the ***Parameters of axis*** field unchanged.

   b) Enter the name for the new stage type in the ***Save as*** field.



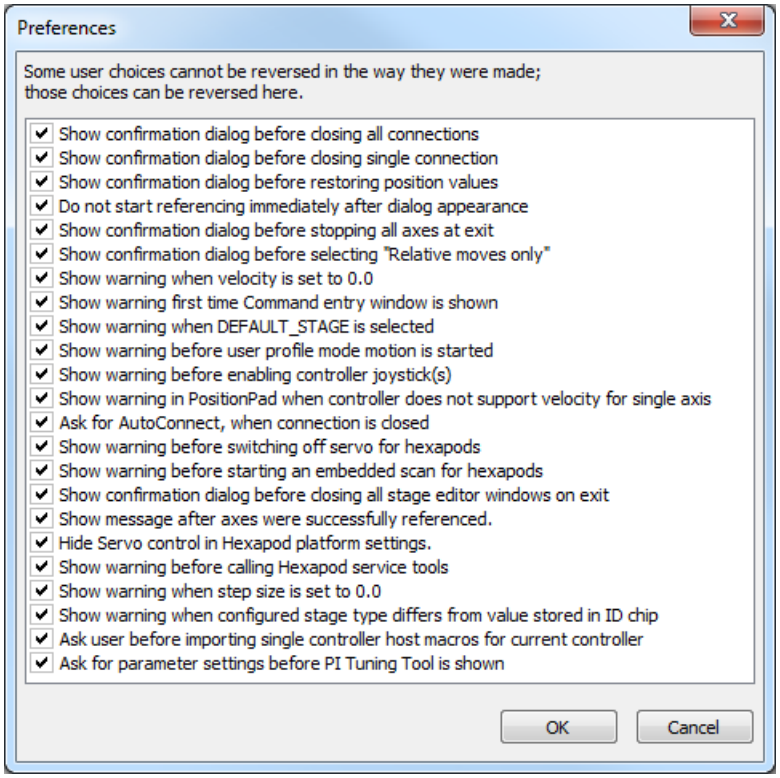
   c) Click ***OK***.

You can also save parameter changes using the following functions of the controller menu (p. 12) in the PIMikroMove main window (availability depends on controller):

- ***Save parameters as User Stage type…***
- ***Save parameters to non-volatile memory***

3.13 Preferences

Controls the behavior of the software user interface. In particular, the requirement for confirmation of various actions can be turned on or off. Note that the confirmation dialog boxes themselves have a checkbox to turn off future appearances of the dialog, but the ***Preferences*** window is the only place to turn them back on.



4 Controller-Specific Windows

Windows which are in any way controller-specific are accessible from the corresponding controller menu.

Note that windows which are available for all controllers can be opened from the main window using the items of the ***Tools*** and ***View*** menus. Some can also be opened with a key combination or toolbar button. See "Additional Common Windows" (p. 61).

In this Chapter

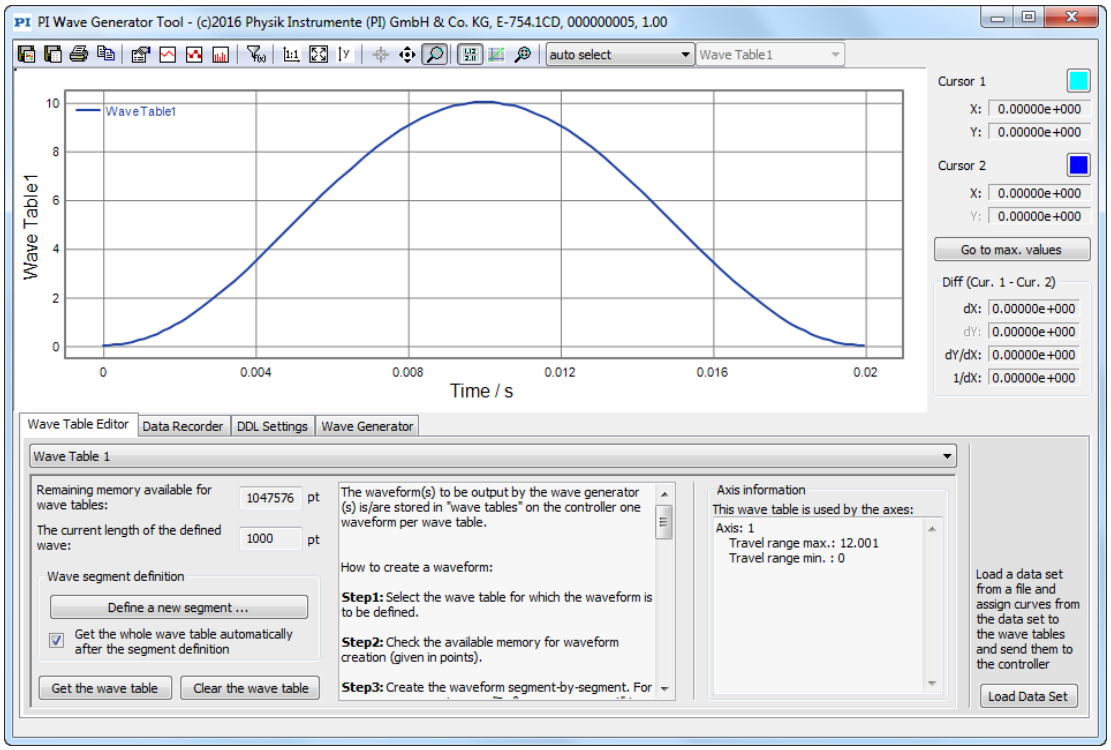
4.1 PI Wave Generator Tool

The ***Show wave generator...*** menu item is on the controller menu of supporting controllers. It gives access to a separate ***PI Wave Generator*** Tool window where you can

- Create waveforms for later output by the wave generator(s) and save them to data files, see "Wave Table Editor Tab Card" (p. 104)
- Configure the data recorder, see "Data Recorder Tab Card" (p. 106)
- Use the Dynamic Digital Linearization (DDL) feature, a standard or optionally purchased password-protected feature which can be activated before or after initial delivery, see "DDL Settings Tab Card" (p. 107)
- Start the wave generator(s) with the selected waveform(s) and options, see "Wave Generator Tab Card" (p. 110)

A graphics pane is always available, either displaying the data related to the currently active tab

card or data loaded from a file. With the toolbar on top of the ***PI Wave Generator*** Tool window, you can configure the graphics pane (p. 68) and import/export data to/from the display. A short description will be displayed if the mouse cursor hovers over a toolbar button.



See the controller's user manual for the controller-specific properties, e.g. for the number of wave tables, data recorder tables and DDL tables, the number of points per table, the assignment of wave tables to axes and wave generators and the assignment of DDL tables to axes.

4.1.1 Wave Table Editor Tab Card

The waveform(s) to be output by the wave generator(s) are stored in wave tables on the controller—one waveform per wave table. It is also possible to save waveforms to data files on the host PC and to load them back to the wave tables at a later time.

The available tables and wave types depend on your controller. See the controller's user manual for more information.

How to create a waveform:

1. Select the wave table in which the waveform is to be stored.
2. Check the maximum possible length of the waveform (given in points).
3. Create the waveform segment by segment. For every new segment, click ***Define a new segment…*** to open the separate definition dialog. See below for details.
   The order of the segments in a waveform can not be changed later. To change individual segments or modify their order, the complete waveform must be recreated segment by segment.
4. Check the waveform in the graphics display. To refresh the display, click ***Get the wave***

table.
It is not possible to select individual segments for display. Only the complete waveform can be displayed.

5. Optionally: Save the content of the graphics display (i.e. the waveform) to a data file on the host PC using the ***Export data*** icon in the toolbar.

To delete the complete wave table content, click ***Clear the wave table***. Segments cannot be deleted separately.

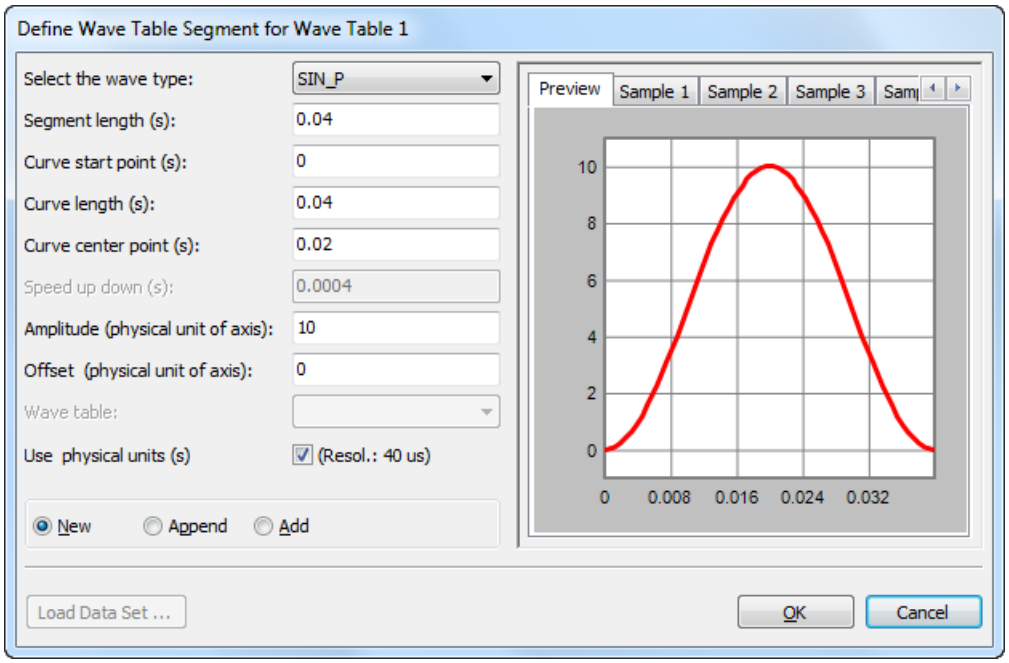
To assign wave tables to wave generator(s) and to start the wave generator output, go to the ***Wave Generator*** tab card (p. 110). The ***Axis information*** pane on the ***Wave Table Editor*** tab card shows the assignment of the current active wave table to the axes according to the settings on the ***Wave Generator*** tab card.

INFORMATION

The wave table content is lost when the controller is powered down or rebooted.

Waveforms which are supposed to run at the same time (each with a different wave generator) should have the same length. Otherwise all waveform output will be adapted to have the same cycle length—in most cases cut to the length of the shortest waveform currently running (depends on controller).

Define Wave Table Segment



Define segment handling:

- When the ***New*** option is activated, the content of the wave table will be completely overwritten by the new segment.
- The ***Append*** option will append the currently defined segment to the previously defined wave table content (i.e. concatenates newly defined segment to existing waveform).

- For some controllers, another option is available:
  The ***Add*** option will add the point values of the newly defined segment to those already in the wave table point by point (i.e. modifies the waveform amplitude).

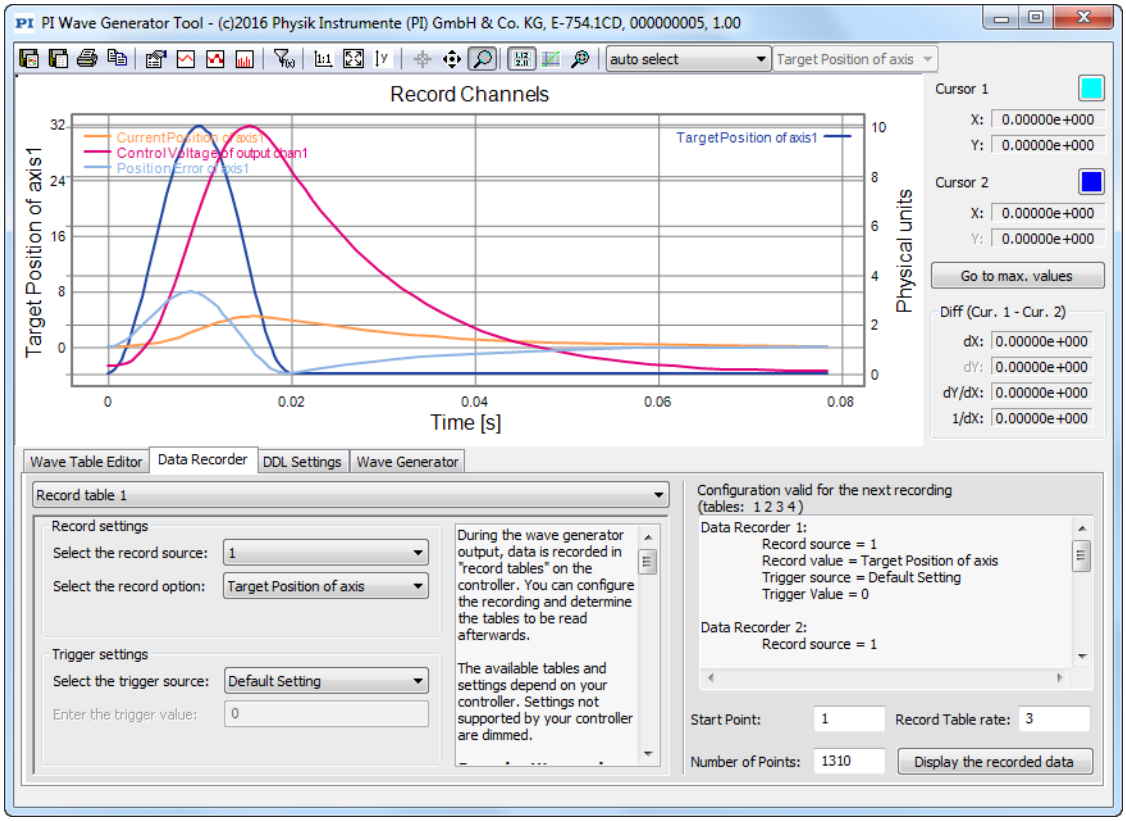
Define the curve shape for the segment:

1. Select the wave type—the fields necessary for the segment definition are then adapted depending on the wave type selected. The WAV_PNT wave type permits loading user-defined curves from data files on the host PC.
2. Enter suitable values in the available definition fields. See the examples in the right pane of the definition dialog for more information.
   For the WAV_PNT wave type, click ***Load Data Set…*** to load points from a file on the host PC. If the loaded file contains multiple waveforms, select one in the ***Wave table*** field. ***Curve start point*** gives the starting point in the wave table and ***Curve length*** the length of the wave (in points).
3. Click ***OK*** to send the defined segment to the wave table on the controller.

4.1.2 Data Recorder Tab Card

During wave generator output, data is recorded in record tables on the controller. You can configure the recording and select tables to be read afterwards.

The available tables and settings depend on the controller. Settings not supported by the controller are dimmed.



INFORMATION

Record settings and the graphics display may differ:

The graphics display always shows the record tables listed in the ***Configuration*** pane. But while the display shows the last recorded content of the tables, the settings listed in the ***Configuration*** pane will be applied to the next recording. For this reason:

- Use the graph labeling in the display to identify the displayed content.
- All data you want to see in the display must be configured before the recording is started.

How to configure the next recording:

1. Select the record table to configure.
2. Select the record source.
3. Select a suitable record option.
   If you select ***disable***, the record table is no longer listed in the ***Configuration*** pane, and it will be excluded from the display the next time you click ***Display the recorded data***.
4. Select the trigger source and enter a suitable trigger value.
5. Optionally: Set the number of points to display.
   By default, this value will be set to the length of the waveform which is output by the wave generator.
   The ***Number of Points*** value is valid for every record table listed in the ***Configuration*** pane. It is not possible to vary the displayed number of points from table to table.

To load the last recorded data to the display, click ***Display the recorded data***.

Recording is started automatically when a wave generator is started on the ***Wave Generator*** tab card (p. 110). As long as a wave generator is running, recording can be restarted by clicking ***Display the recorded data***.

INFORMATION

Depending on the controller, data may be recorded in a table even when the record option is set to ***disabled*** for that table. If this is the case, you can load that data to the display by setting any other record option for the table and clicking ***Display the recorded data***. Use the graph labeling in the display to identify the displayed content.

4.1.3 DDL Settings Tab Card

INFORMATION

On most controllers, the Dynamic Digital Linearization (DDL) feature must be ordered specifically. You can activate it after purchase and without opening the device. See the user manual of the controller for more information.

DDL is used in addition to the standard servo control algorithm and makes it possible to achieve significantly better position accuracy for dynamic applications with periodic motion. A DDL table

is used to compensate for the tracking error of an axis. The tables and settings available depend on the type of controller. See the user manual of the controller for more information.

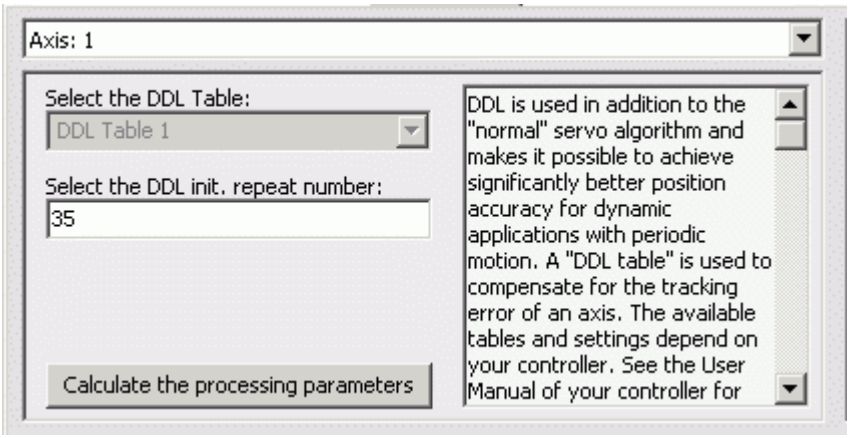
An initialization phase is required to fill the DDL table with data. DDL initialization must be repeated in the following cases:

- New stage connected
- Servo parameters were changed, e.g., due to load changes
- Waveform was changed

INFORMATION

Dynamic Digital Linearization (DDL) will amplify residual system resonances and cause oscillation to build up—the more wave generator cycles used for DDL initialization, the stronger the effect.

Before you work with DDL, eliminate oscillation by adjusting the notch filter frequency, servo-loop P-term (loop gain), I-term (time constant), and slew rate. See "Servo controller Dynamic Calibration" in the controller's user manual for more information.



Left pane of the DDL Settings tab card

How to initialize and use DDL for an axis:

1. Make sure that servo is switched on.
2. On the ***Wave Table Editor*** tab card (p. 104), define the waveform.
3. In the left pane of the ***DDL Settings*** tab card, configure DDL for the axis:
   a) Select the axis.
   b) Select the DDL table to use for the axis (presently, the factory default can not be changed).
   c) Set the number of wave generator cycles to use for DDL initialization (***DDL init. repeat***, factory default = 35).
   d) If the number of DDL initialization cycles and/or the servo parameters were changed since the last time the internal DDL processing parameters were calculated, click ***Calculate the processing parameters***.

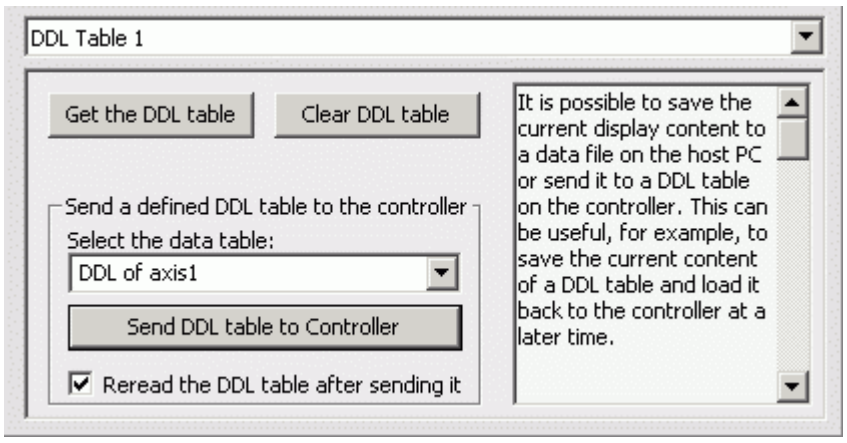
4. On the ***Wave Generator*** tab card (p. 110):
   a) Select the wave generator and hence the axis (the same axis as selected on the DDL Settings tab card).
   b) Assign the waveform (i.e. the wave table) to the axis.
   c) Activate the ***Use and reinitialize*** DDL flag.
   d) Start the wave generator and thus also the DDL initialization/usage.
5. Optionally: Examine the contents of the DDL table in the graphics display. On the ***DDL Settings*** tab card, click ***Get the DDL table*** to load the data.

As long as your application does not change, you can use the current DDL table content without new initialization. In this case, deactivate the ***Use and reinitialize DDL*** flag on the ***Wave Generator*** tab card and start the wave generator with the ***Use DDL*** flag activated.

INFORMATION

Starting DDL initialization for all axes at the same time is recommended. Each new initialization will stop all running initialization processes.
The DDL table content and the calculated processing parameters will be lost when the controller is powered down or rebooted.
Depending on the controller, it may be possible to save the DDL processing parameters to non-volatile memory.

You can examine and manage the DDL table content using the graphics display and its toolbar buttons and the controls in the right pane of the DDL Settings tab card.



Right pane of the DDL Settings tab card

It is possible to save the current display content to a data file on the host PC or to send it to a DDL table on the controller. This can be useful if you want, for example, to save the current content of a DDL table and load it back to the controller at a later time.

To save the display content to a file:

1. Select a DDL table.
2. Click ***Get the DDL table*** to load the content of the selected DDL table from the controller to the display.

3. Click the ***Export data*** button in the toolbar of the ***PI Wave Generator Tool*** window to save the display content to a file on the host PC.

To send the current display content to a DDL table on the controller:

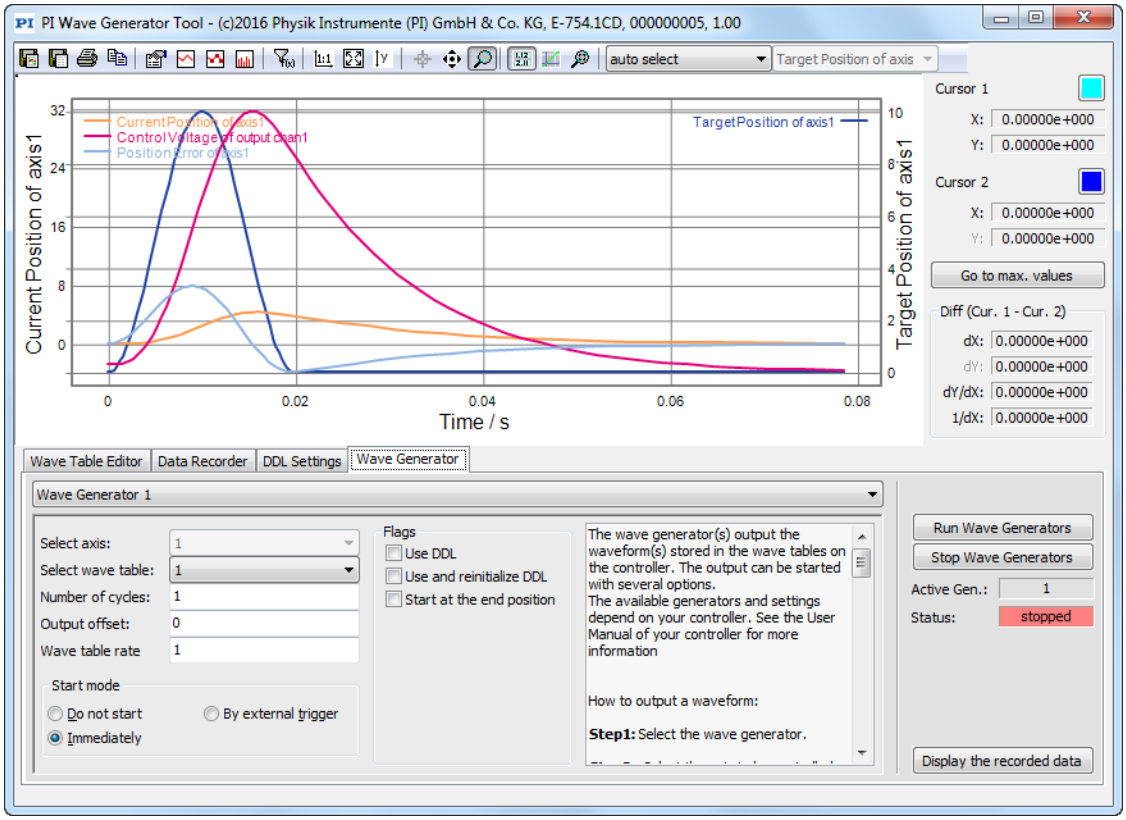
1. Select a DDL table (to this table the data will be written on the controller).
2. Optionally: Load DDL data to the display from a file on the PC using the ***Import data*** button in the toolbar of the ***PI Wave Generator Tool*** window.
3. If the display contains multiple DDL tables, select one in the ***Select the data table*** field.
4. To send the (selected) data from the display to the controller, click ***Send DDL table to controller***.
5. Optionally: If the ***Reread the DDL table after sending it*** checkbox was not activated, you can load the new DDL table content to the display using ***Get the DDL table*** to check if data was written successfully.

To delete the content of the currently selected DDL table on the controller, click ***Clear DDL table***.

4.1.4 Wave Generator Tab Card

A wave generator can output a waveform stored in a wave table on the controller to an axis. The output can be started with several options.

The number of wave generators and the settings available depend on the type of controller. See the controller's user manual for more information.



How to output a waveform:

1. Select a wave generator; all the settings made in Steps 2 through 7 will refer to this wave generator.
2. Select the axis to be controlled by the wave generator (presently, the factory default can not be changed).
3. If selection is supported by the controller, select the wave table and hence the waveform to be output by the wave generator.

   The waveform must have been previously defined on the ***Wave Table Editor*** tab card (p. 104).

   The same wave table can be selected for multiple wave generators.
4. Set the duration of the wave generator output by setting the number of output cycles. If ***Number of cycles*** is set to 0, the wave generator will run non-stop until you click ***Stop Wave Generators***.
5. Optionally: Set a position offset which will be added to the waveform.
6. Set the start mode for the wave generator. This does not start the wave generator output yet!
7. Optionally: Set options for the wave generator output by activating the corresponding flag(s).
8. Start wave generator output by clicking ***Run Wave Generators***. This starts all wave generators whose ***Start mode*** is different from ***Do not start***.

You can stop all wave generator output by clicking ***Stop Wave Generators***.

When the wave generator output is started, data recording starts automatically. To load the last-recorded data to the display, click ***Display the recorded data***. As long as a wave generator is running, recording is then restarted. Recording can be configured on the ***Data Recorder*** tab card (p. 106).

INFORMATION

If multiple wave generators are running simultaneously, with some controllers, their output cycles will be adapted to have the same length—on most controllers, cut to the length of the shortest waveform currently running.

If the ***Start at the endpoint*** flag is activated, the ***Output Offset*** value will be incremented internally with each output cycle. When the wave generator is stopped, the current valid offset value will be automatically loaded back to the ***Output Offset*** field.

Servo must be on if the wave generator is started with the ***Use DDL*** or ***Use and reinitialize DDL*** flags activated. Otherwise the DDL feature will not work.

Wave generator output will continue even if the ***PI Wave Generator*** Tool is closed or if the high voltage output is deactivated.

The assignment of wave tables to wave generators is displayed on the ***Wave Table Editor*** tab card, in the ***Axis information*** pane.

Depending on the controller, it may be possible to modify the wave generator output by setting the wave table rate. See the controller's user manual for more information.

4.2 PI Frequency Generator Tool

INFORMATION

In the case of controllers with wave generator functionality, each axis can be controlled by a wave generator that outputs waveforms. The wave generator is particularly suitable for dynamic applications, in which the axis performs arbitrary motion profiles.

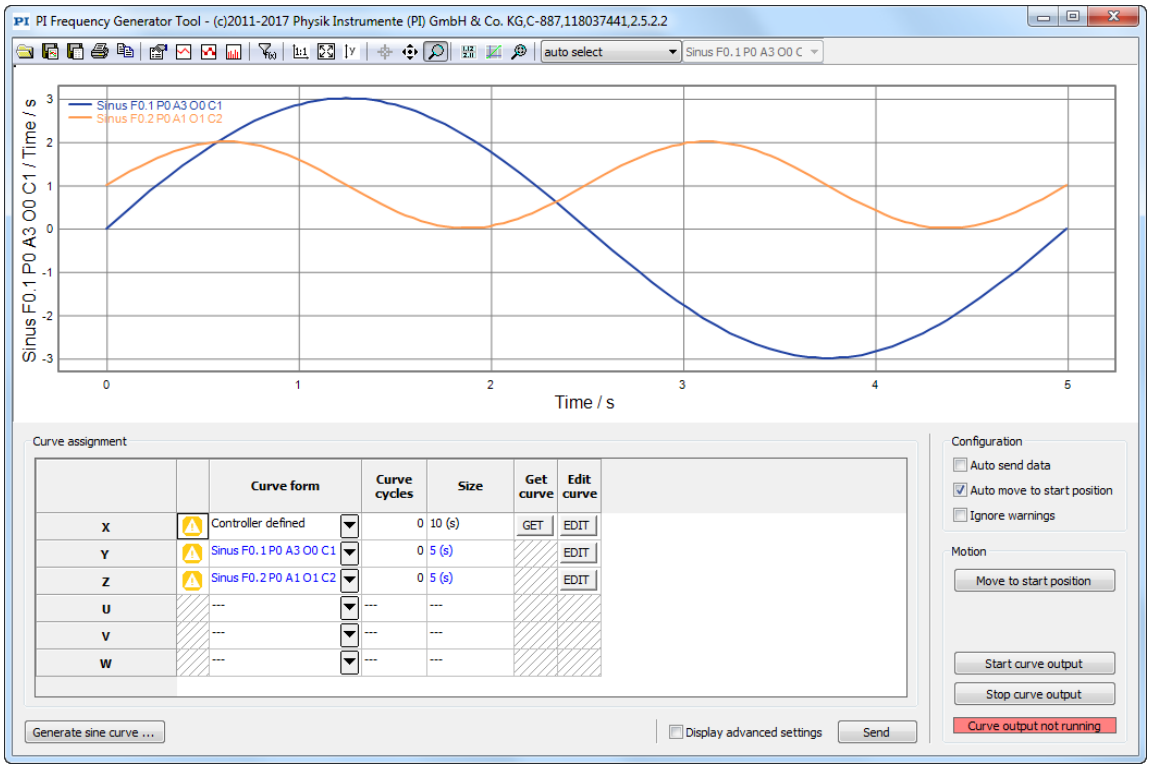
Waveforms can be defined and temporarily saved in the controller. While the wave generators are permanently assigned to the controller axes, the waveforms can be assigned to the wave generators and thus to the axes as desired. A waveform can be used by several wave generators at the same time.

Waveforms are saved in wave tables in the controller. Each wave table contains the data of one waveform.

Further information can be found in the documentation of the controller used.

If you use the ***PI Frequency Generator Tool*** window in PIMikroMove to work with the wave generator, you must only know which axis is to be moved and which waveform you want to use. You can also save waveforms on the PC and load them again later. This considerably simplifies work with the wave generator.

The ***PI Frequency Generator Tool*** window is opened via the ***Show frequency generator …*** menu function in the controller menu. When the window is opened, PIMikroMove reads out the current settings of the controller and displays them (available waveforms and their assignment to axes).



The ***PI Frequency Generator Tool*** window consists of two panels:

- Graphics panel: The available waveforms are displayed as graphs. Via the toolbar of the graphics panel you can load, save, and hide waveforms, among other things.
- Panel for ***Curve assignment***, ***Configuration***, and ***Motion***: Waveforms can be defined and assigned to the axes, and the output of the wave generator can be configured, started, and stopped.

INFORMATION

Warning or information signs in the ***Curve assignment*** table indicate that there are notes for the current configuration.

➢ Click the warning or information sign to display the note.

Waveforms can be displayed and hidden in the graphics panel via the ***Configure Display Properties*** toolbar button:  . The displayed waveforms can be assigned to axes in the ***Curve assignment*** panel. Waveforms that are hidden in the graphics panel are not available for assignment.

Waveforms can be loaded to the graphics panel and thus made available for axis assignment as follows:

- Definition of sine curves in the ***Define new Sine Curve*** dialog (p. 114), opened with the button ***Generate sine curve ...***
- Loading any waveforms from a file (p. 115) on the PC via the ***Import data*** button: 
- Loading the current wave table contents from the controller (p. 116) via the button ***GET*** in the curve assignment table

For the assignment of waveforms to axes see "Assigning Waveforms to Axes" (p. 117).

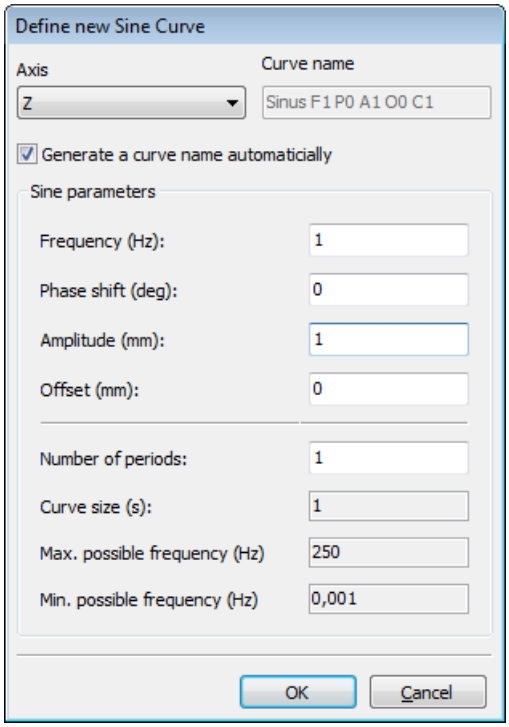
For the execution of moves see "Starting Motion" (p. 118).

For storing waveforms on the PC or via macros on the controller see "Saving Waveforms" (p. 120).

To configure the data display in the graphics pane and perform operations with the data use the window's toolbar buttons (p. 68).

4.2.1 Defining and Modifying Sine Curves

New sine curves can be defined in the ***Define new Sine Curve*** dialog.



In the ***Axis*** field, select the axis which the waveform is to be assigned to.

The name of the waveform is automatically generated by the ***PI Frequency Generator Tool*** but can also be entered manually if the option ***Generate a curve name automatically*** is not checked.

INFORMATION

The name of the waveform is used within the PI Frequency Generator Tool and is still retained when the waveform is saved in a file on the PC (p. 120). However, the name of the waveform is not sent to the controller. The name is therefore no longer present and is replaced by **Wave table n** when the waveform is loaded back to the ***PI Frequency Generator Tool*** from the controller (n = identifier of the wave table in which the waveform was saved at the time of loading (p. 116)).

The parameters of the sine curve (frequency in Hz, phase shift in degrees, amplitude, and offset in the physical unit of the axis) are entered in the fields in the ***Sine parameters*** panel. The number of periods of the sine can be entered in the ***Number of periods*** field.

INFORMATION

The number of periods refers to the waveform and therefore applies to each axis (= each wave generator) to which the waveform is assigned. In contrast, the settings for the number of output cycles and the output rate are linked to the individual wave generators (see "Starting Motion" (p. 118)).

OK ends the definition and closes the ***Define new Sine Curve*** dialog. The waveform is now displayed in the graphics panel of the ***PI Frequency Generator Tool*** window and can be assigned to axes. If an axis was selected when a waveform was defined, the waveform has already been assigned to this axis but the assignment has not been sent to the controller yet.

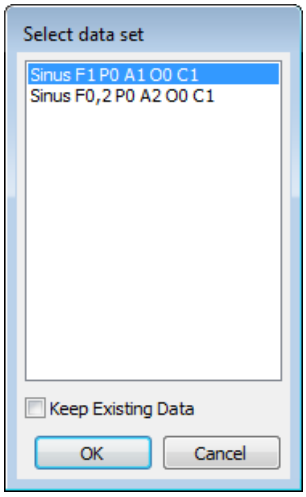
After definition, the sine curves that are displayed in the graphics panel can be changed in the ***Edit Sine Curve*** dialog at any time. If a sine curve is to be changed, it first has to be assigned to an axis in the ***Curve assignment*** panel (p. 117) (sending to the controller is not necessary). The ***EDIT*** button then appears for the axis in the ***Edit curve*** column. This button can be used to open the ***Edit Sine Curve dialog***, which is identical to the ***Define new Sine Curve*** dialog (except the dialog's caption).

4.2.2 Loading Waveforms from File

Any desired waveforms that are defined pointwise can be loaded from files on the PC to the ***PI Frequency Generator Tool***. The data must exist in GCS array format.

A waveform can be loaded by clicking the ***Import data*** button in the ***PI Frequency Generator Tool*** window's toolbar. A file selection window opens in which the desired file must be selected (file extension .dat). The further procedure depends on whether the selected file contains different waveforms with separate header entries:

- The file contains different waveforms with separate header entries: The ***Select data set*** window opens, in which you can select one or more waveforms.



  The ***Keep Existing Data*** option must be checked if the waveforms already present in the ***PI Frequency Generator Tool*** are to be retained. If the checkbox is not marked, all waveforms that were previously displayed in the graphics panel of the ***PI Frequency Generator Tool*** (and possibly already assigned to axes) will be deleted.

- The file contains a single waveform or different waveforms without separate header entries: A window with the query "Keep existing data?" opens. Regardless of whether the query is answered with yes or no, all waveforms contained in the file are then loaded.

The loaded waveforms are displayed in the graphics panel and can be assigned to axes in the ***Curve assignment*** panel (p. 117).

INFORMATION

Data in GCS array format contains header information on the sample time of the waveform (example: # SAMPLE_TIME = 0.001). This sample time must be observed when the waveform is output by the wave generator. If necessary, the ***PI Frequency Generator Tool*** therefore interpolates the data when it is loaded from the file. The interpolation takes place based on the specified sample time and the "basic" cycle time of the controller (servo cycle time or the smallest possible multiple of this for the controller).

4.2.3 Loading Waveforms from the Controller

Waveforms that are already present in the controller when the ***PI Frequency Generator Tool*** window is opened can be loaded to the ***PI Frequency Generator Tool***. A waveform is loaded by clicking the ***GET*** button in the ***Curve assignment*** panel. After the waveform has been loaded, the ***GET*** button disappears.

The loaded waveforms are displayed in the graphics panel and their axis assignment remains. Loaded waveforms can also be assigned to other axes in the ***Curve assignment*** panel (p. 117).

INFORMATION

Waveforms that are already present in the controller but have not yet been loaded to the ***PI Frequency Generator Tool*** are shown under the name **Controller defined**. During loading, the name is replaced by **Wave table n** (n = identifier of the wave table in which the waveform was saved at the time of loading).

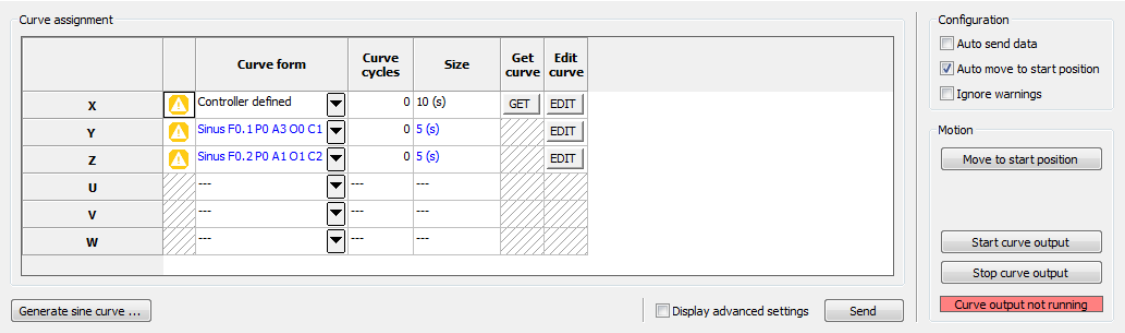
INFORMATION

When a waveform is loaded from the controller, the current contents of the wave table that is assigned to the corresponding axis (= the wave generator) in the controller are queried. For this purpose, the ***PI Frequency Generator Tool*** the `GWD?` command.

Command descriptions and further information can be found in the documentation of the controller used.

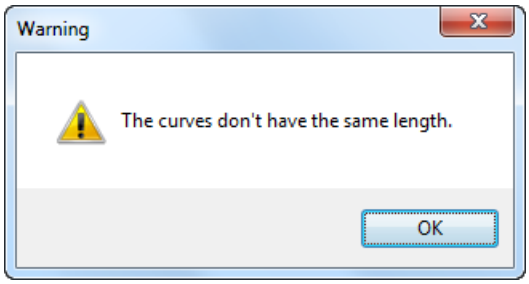
4.2.4 Assigning Waveforms to Axes

The waveforms that are displayed in the graphics panel are available for axis assignment.



The available waveforms are assigned to the axes in the ***Curve form*** column of the ***Curve assignment*** panel. Each line in the ***Curve assignment*** table belongs to an axis of the controller. The axis identifiers are shown in the column on the left margin of the panel.

Warning signs in the table indicate that there are notes for the current configuration. Clicking the warning sign displays the note, e.g.:



Most frequent cause: The axes were assigned waveforms with different lengths. The behavior in the case of different curve lengths depends on the controller: While some controllers (e.g. hexapod controllers) output the waveforms as defined, other controllers (e.g. E-712) shorten the output to the length of the shortest waveform. Further information can be found in the documentation of the controller used.

For the assignment to take effect, it must be sent to the controller. The assignment is sent to the controller by clicking the ***Send*** button underneath the table. Assigned waveforms that have not yet been sent to the controller are displayed in a blue font. Once the waveforms have been sent, their font color changes from blue to black.

INFORMATION

Clicking ***Send*** does not start the motion. For starting the motion, see "Starting Motion" (p. 118).

When the ***Auto send data*** checkbox in the ***Configuration*** panel is marked, the assignment is automatically sent to the controller when the wave generator output is started (in the ***Motion*** panel, see "Starting Motion" (p. 118)). Clicking ***Send*** is then not necessary.

INFORMATION

When sending, the following information for the waveforms is temporarily written to the controller:

- The definition of the waveform is written to a particular wave table of the controller (corresponding command: WAV).
- The wave table (and thus the waveform) is assigned to a wave generator (and thus to an axis) of the controller (corresponding command: WSL).

Command descriptions and further information can be found in the documentation of the controller used.

When the ***Display advanced settings*** checkbox (next to the ***Send*** button) is marked, more columns are added to the ***Curve assignment*** table.

Output rate and ***Interpolation*** columns: see "Starting Motion" (p. 118)

Table ID column: In this column, you can select the wave table of the controller to which the waveform is to be written when it is sent. A new waveform is always automatically written to the next free wave table of the controller.

4.2.5 Starting Motion

The motion of the mechanical system connected to the controller according to the assigned waveforms is started and stopped with the buttons in the ***Motion*** panel:

- ***Start curve output*** starts the wave generator output and the motion. The wave generator output is always started for all axes to which a waveform is assigned. If an axis should **not** move, it must not be assigned a waveform.

  Notes of the ***PI Frequency Generator Tool*** on the current configuration can be suppressed by marking the ***Ignore warnings*** checkbox in the ***Configuration*** panel.
- ***Stop curve output*** stops the wave generator output and the motion. The wave generator output is always stopped for all axes.
- ***Move to start position*** moves the axes at the currently valid velocity to the starting point of the respectively assigned waveform. When the ***Auto move to start position*** checkbox in the Configuration panel is marked, the motion to the starting point takes place automatically when the wave generator output starts. Clicking ***Move to start position*** is then not necessary.

If the axes are **not** moved to the starting point with one of the two options, it is possible that they will not reach the starting point fast enough when the wave generator output starts and the wave generator output will fail.

INFORMATION

To start and stop the wave generator output, the ***PI Frequency Generator Tool*** sends the WGO command.

To start, WGO is sent with the start mode "Immediate start of the wave generator output, synchronized by servo cycle" (bit 0). If another start mode and/or additional start options are

to be used, the wave generator output must be started outside of the ***PI Frequency Generator Tool*** window, e.g., in the ***Command entry*** window of PIMikroMove, with the corresponding WGO command.

Command descriptions and further information can be found in the documentation of the controller used.

In the ***Curve assignment*** panel, the following settings can be made for the wave generator output:

- Number of output cycles of the waveform
- Wave generator output rate
- Interpolation type

Changed values are displayed in a blue font. They do not become valid until they are sent to the controller by clicking the ***Send*** button (underneath the ***Curve Assignment*** table). Once the changed values have been sent, their font color changes from blue to black. When the ***Auto send data*** checkbox in the ***Configuration*** panel is marked, the settings are automatically sent to the controller when the wave generator output is started.

Number of output cycles

The number of output cycles of the waveform can be specified in the ***Curve cycles*** column of the ***Curve assignment*** table. The wave generator output is stopped once the specified number of cycles has been output.

The default setting for the number of output cycles is 0. With the default setting, the waveforms are output for an unlimited time until they are stopped with the ***Stop curve output*** button.

INFORMATION

To set the number of output cycles, the ***PI Frequency Generator Tool*** sends the WGC command.

Command descriptions and further information can be found in the documentation of the controller used.

Wave generator output rate

The output rate of the wave generator can be set in the ***Output rate*** column of the ***Curve assignment*** table. The ***Output rate*** column is only available when the ***Display advanced settings*** option (next to the ***Send*** button) is checked.

The output cycles of the waveform can be extended by changing the output rate. The duration of an output cycle for the waveform can be calculated as follows:

Output duration = servo cycle time of the controller * output rate * number of points of the waveform

INFORMATION

To set the output rate, the ***PI Frequency Generator Tool*** sends the WTR command.
Command descriptions and further information can be found in the documentation of the controller used.

Interpolation type

The interpolation type used by the wave generator can be specified in the ***Interpolation*** column in the ***Curve assignment*** table. The ***Interpolation*** column is only available when the ***Display advanced settings*** option (next to the ***Send*** button) is checked.

The wave generator uses the interpolation for outputs between wave table points when the output rate is greater than 1.

INFORMATION

To set the interpolation type, the ***PI Frequency Generator Tool*** sends the WTR command.
Command descriptions and further information can be found in the documentation of the controller used.

INFORMATION

The behavior during the wave generator output depends on the controller: Some controllers (e.g. hexapod controllers) continuously check during the motion whether the commanded target positions can actually be reached. If it is likely that the working area limits will be exceeded, the motion is interrupted and an error code is set. Other controllers (e.g. E-712) limit the target positions to the minimum/maximum allowed values without setting an error code if it is likely that limits will be exceeded. Further information can be found in the documentation of the controller used.

INFORMATION

Error messages from the controller during wave generator output are also displayed in the status line of PIMikroMove's main window.

- During the wave generator output, check the status line of PIMikroMove.

4.2.6 Saving Waveforms

Saving Waveforms on the PC

The waveforms that are displayed in the graphics panel can be saved on the PC as ASCII data or as a graphics file. The buttons to perform these functions can be found in the toolbar above the graphics display:

This button opens the ***Save data*** window, in which you can select a directory on the PC and save the data of the waveform in comma separated values (.csv) format or GCS array format (.dat). If you want to load the data to the ***PI Frequency Generator Tool*** later (p. 115), you must save it in GCS array format.

This button opens a dialog in which you can specify the size of the graphic. After the size has been specified, the ***Save image*** window opens, in which you can select a directory on the PC and save the graphics display in an image format (e.g. bmp).

Saving Waveforms on the Controller via Macros

Waveforms are only temporarily present on the controller after being sent. When the controller is switched off or rebooted, the waveforms are lost.

In the case of controllers with macro functionality, the waveforms and the wave generator configurations can be permanently saved on the controller via macros.

Further information can be found in the documentation of the controller used.

4.3 Wave Table Operation

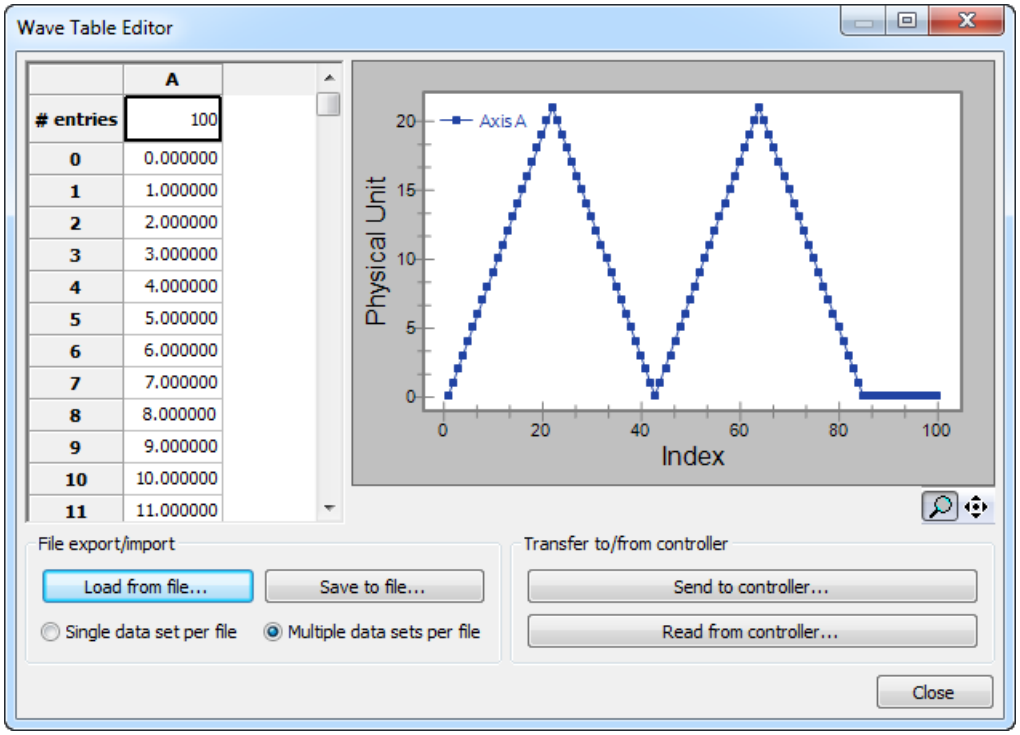
Some controllers can be controlled by an internal "wave table" which outputs user-specified target points. The interpretation of these target points (e.g. as positions or voltages) depends on the controller. The wave table feature is especially important in dynamic applications which require periodic motion of the axes.

The following menu items on the controller menu of supporting controllers refer to wave table operation:

- ***Show wave table editor…***: Define wave table points for later output and save them to the controller or to data files on the host PC. A graphics pane is available. See "Wave Table Editor" (p. 122) for details
- ***Start wave table output…***: Wave table output can be started immediately or by external trigger pulses. See "Start Wave Table" (p. 123) for details
- ***Stop wave table output…***: See "Stop Wave Table Output" (p. 124) for details

4.3.1 Wave Table Editor

Use the ***Show wave table editor…*** item in the controller menu to open the ***Wave Table Editor*** window. In this window, you can define wave table points for later output, check them in a graphics pane and save them to the controller or to data files on the host PC.



The table in the leftmost pane of the ***Wave Table Editor*** window lists the wave table content, i.e. the target points for all currently connected axes which support wave table operation. The table content is displayed as a graph in the graphics pane.

INFORMATION

When the ***Wave Table Editor*** window is opened, the table does not show the actual values from the controller(s) yet. To load the data for the individual axes from the controller(s), click the ***Read from controller…*** button below the graphics pane. Reading can take some time, depending on the number of points to read.

The leftmost column of the table shows the index of the target points. The other columns belong to individual axes, with the table header showing the axis identifier. In the first row of the axis columns (***# entries***), you can determine the number of points to be written for each axis. The maximum number of points depends on the controller. All subsequent rows of the axis columns contain the values of the target points. The interpretation of the target points depends on the controller used and on the current servo mode.

To change a value, click into the corresponding field and enter the new value, or, with the ***# entries*** fields, use the arrow buttons which appear. The graphics pane is updated immediately.

INFORMATION

Values entered in the table are not written to the controller or to a file on the host PC until you click the corresponding button.

To write the current table content to the controller and hence overwrite the values which may already be present there, click the ***Send to controller…*** button below the graphics pane. Depending on the controller, the written points may be automatically saved to non-volatile memory (see controller's user manual for details).

You can save the data from the table to a file on the host PC using the ***Save to file…*** button (wave table file with the extension .tbl). The current table content is considered as a single data set. By activating the corresponding radio button, you can save every data set to a new file, or save multiple data sets to the same file. To load data from a file on the host PC, use the ***Load from file …*** button. You can assign the data sets to the individual axes when loading.

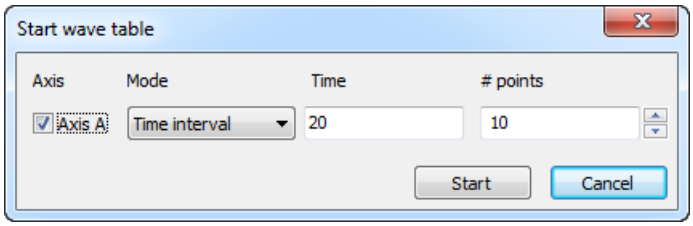
INFORMATION

Wave table content which has not been sent to the controller or saved to a file on the host PC is lost when the ***Wave Table Editor*** window is closed.

To start the wave table output, close the ***Wave Table Editor*** window and select the ***Start wave table output…*** function from the controller menu. See "Start Wave Table Output" (p. 123) for details.

4.3.2 Start Wave Table

Use the ***Start wave table output…*** item in the controller menu to open the ***Start wave table*** window. Each line in this window is dedicated to one axis.



Via the ***Axis*** checkbox, you can determine if the wave table output is to be started for the axis when clicking the ***Start*** button. Removing the check for an axis means that the state of the wave table output for this axis will remain unchanged.

In the ***Mode*** field, you can select the following start options (default configuration, for other wave table output configurations see the controller's user manual):

- ***Time interval*** means that the wave table output is started immediately when clicking the Start button. Each point will be output for the amount of time specified in the ***Time*** column (in milliseconds). Output will roll over for the points specified by ***# points*** and continue until stopped, e.g. by using the ***Stop Wave Table Output*** menu item in the controller menu.

- ***External trigger*** means that the wave table output is to be started by external trigger pulses after the ***Start*** button was clicked. By default, one wave table point is output each time an external trigger pulse is received. Make sure that the controller is configured to accept trigger input, and that a suitable trigger signal is available. See the user manual of your controller for details.

In the ***Time*** field specify the amount of time in milliseconds each wave table point is to be output (for "time interval" mode). Note that wave table output is disabled when ***Time*** is set to 0.

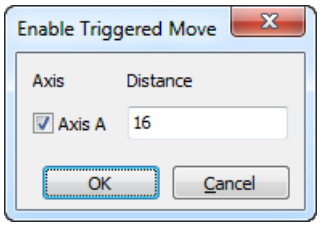
In the ***# points*** field specify the number of wave table points to be used for output. Note that wave table output is disabled when ***# points*** is set to 0. See the user manual of your controller for the maximum number of points.

4.3.3 Stop Wave Table Output

Use the ***Stop wave table output…*** item in the controller menu to open the corresponding axis selection window. To stop the wave table output for an axis, check the corresponding checkbox and click ***OK***.

You can stop the wave table output for an axis also in the ***Start wave table window*** by entering 0 in the corresponding ***# points*** field (see "Start Wave Table Output" (p. 123)).

4.4 Enable Triggered Move



With some controllers, axis motion can be controlled by external trigger pulses. Each pulse causes a relative step of a predefined size.

The ***Enable triggered move…*** item is on the controller menu of supporting controllers. It opens the corresponding window where each line belongs to one axis.

Before enabling triggered motion, make sure that the controller is configured to accept trigger input, and that a suitable trigger signal is available (see the user manual of your controller for details).

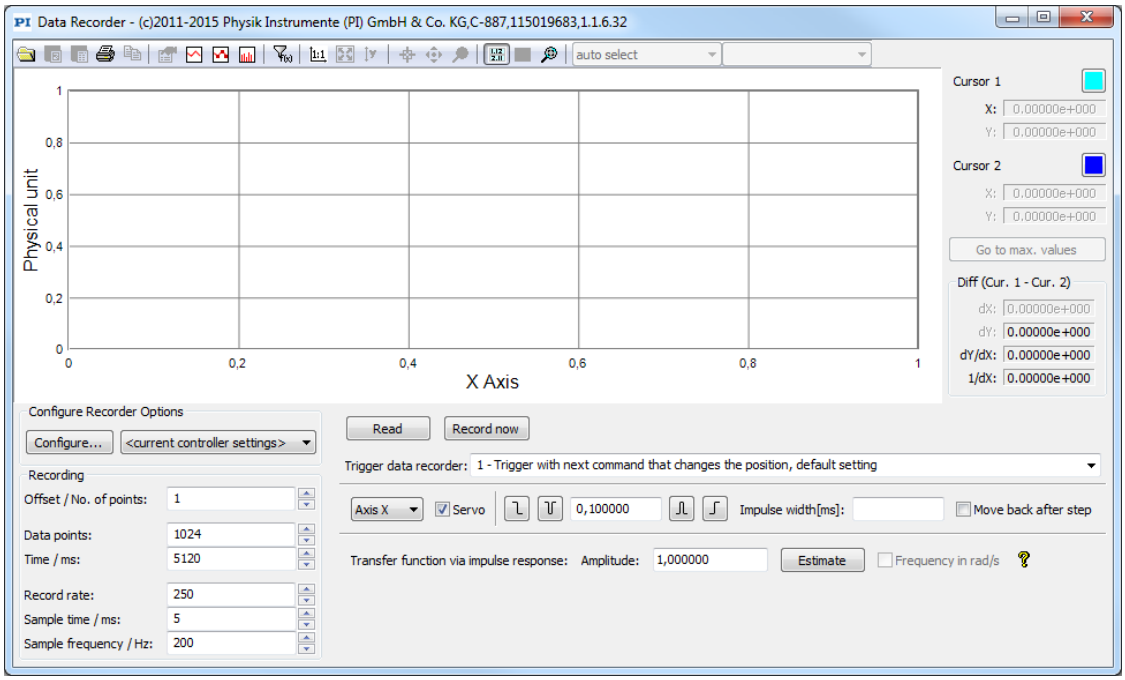
To enable or disable triggered motion for the individual axes, use the ***Axis*** checkboxes. Triggered motion is enabled for an axis when its box is checked. To set the step size to be used, type the corresponding value(s) in the ***Distance*** field(s). The interpretation of the step size value (e.g. as position in µm or as voltage in volts) depends on the controller.

INFORMATION

Depending on the controller, triggered motion and other control sources (e.g. target values given on the ***Axes*** tab card or wave table output) may be mutually exclusive. If this is the case, other control sources will cause an error for the axis as long as triggered motion is enabled for this axis.

4.5 Data Recorder

The ***Show data recorder...*** menu function is on the controller menu of supporting controllers. It gives access to the ***Data Recorder*** window.



In the ***Data Recorder*** window you can:

- Configure recording (p. 127) and define actions to be applied by PIMikroMove to the recorded data. For this click the button ***Configure...***. If you already stored configuration settings for recordings, you can select a stored configuration via the drop-down list next to the ***Configure...*** button (shown with the text "<current controller settings>" in the picture).
- Start data recording via step and impulse buttons (p. 134) and via the ***Record now*** button (p. 135). Note that depending on the controller, data recording may also be triggered by actions done outside of the ***Data Recorder*** window.
- Read recorded data (p. 135) from the controller to display it in the graphics pane. For data reading it is irrelevant if recording was started inside or outside of the ***Data Recorder*** window.

- Configure the data display in the graphics pane and perform operations with the data via the window's toolbar buttons (p. 68).

In the ***Recording*** panel you can define the starting point and the duration of the recording:

Offset / No. of points defines the starting point of the recording.

The duration of the recording is calculated as follows:
Recording duration = servo cycle time * record rate * number of data points
It can be set via these options:

Via ***Data points*** the number of data points to be read from the controller is specified, and via ***Time / ms*** the duration of the recording.

In the ***Record Rate*** field specify the number of servo cycles before the next data point is recorded. The value must be an integer ≥ 1. The servo cycle time is set with either ***Sample time / ms*** or ***Sample frequency / Hz***.

The servo cycle time and the number of data points (length of the data recorder table) depend on the controller.

If supported by the controller, the ***Trigger data recorder*** field is displayed: Here you can specify how recording is to be triggered.

INFORMATION

Do not select option 2 - "Trigger with next command, resets trigger settings to 0". Because of the background processing done by PIMikroMove recording would start immediately.

See the controller's user manual for the controller-specific data recorder properties, e.g. for available record options and trigger settings, for the number of data recorder tables and the maximum number of points per table.

INFORMATION

Depending on the controller, there may be additional tools which use the data recording functionality (for example, "PI Tuning Tool" or "PI Wave Generator Tool"). The ***Data Recorder*** window can not be opened as long as any of these tools is in use.

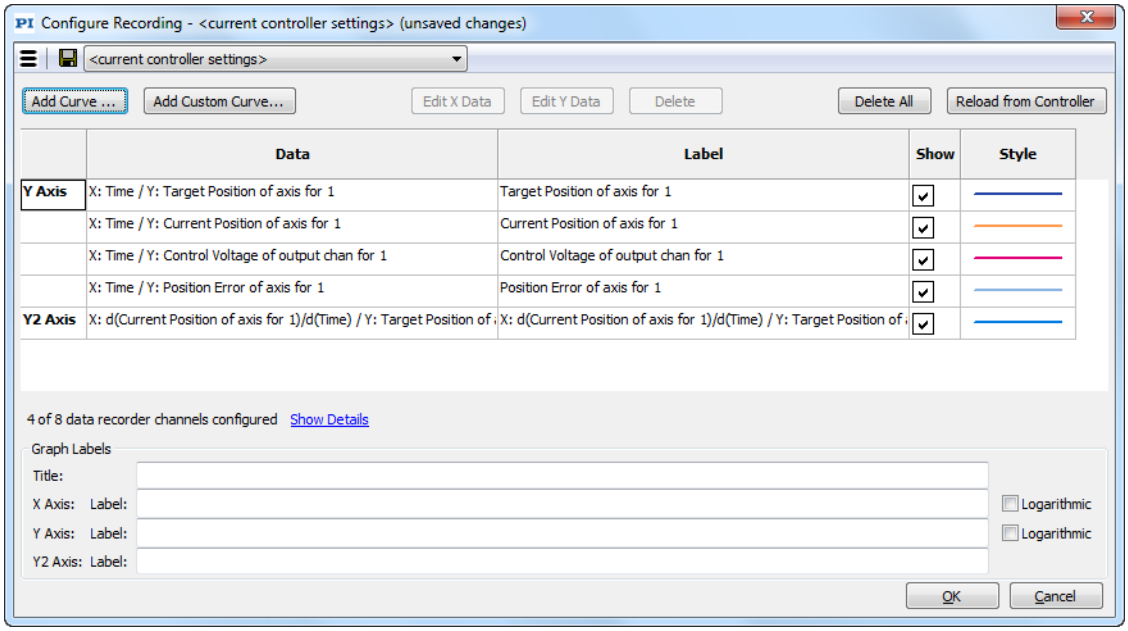
INFORMATION

The display always shows the **last recorded content** of the data recorder tables. Data source and record option for the data you want to see in the display must be configured before the recording is started.

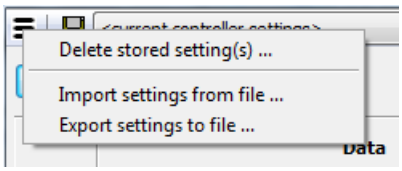
Configuration changes subsequent to recording are possible for actions performed by PIMikroMove.

4.5.1 Configure Recording

The ***Configure...*** button in the ***Data Recorder*** window opens the ***Configure Recording*** window.



Via the menu and toolbar of the window, you can save and load configuration settings for recording:



If there are stored settings for recordings they can be deleted via the function ***Delete stored setting(s) ...***. In the dialog that is opened select the configurations you want to delete.

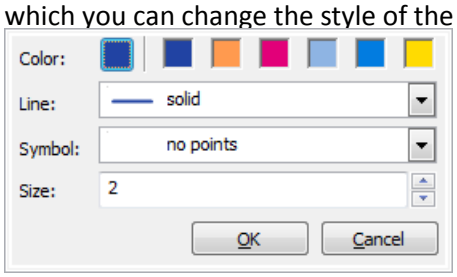
Defined settings for recordings can be exported to file with ***Export settings to file ...***, and settings exported to file can be imported with the function ***Import settings from file ...***.

Use this button to save the defined configuration settings: They will be stored by PIMikroMove so that they can be loaded via the drop-down list next to the button (shown with the text "<current controller settings>" in the picture).

The current recording configuration is displayed in the table:

- Each line represents a curve.
- The leftmost column contains an entry for each Y axis of the graph: ***Y Axis*** = the left Y axis of the graph (standard), ***Y2 Axis*** = the right Y axis of the graph. Below each entry, the assigned curves are listed.
- ***Data*** shows data for X and for Y that makes up the curve.
- ***Label*** shows under which name the curve will be displayed in the graph.

- By checking/unchecking the ***Show*** checkbox, you can tell PIMikroMove to display/hide the curve in the graph.
- ***Style*** defines how to display the curve in the graph. Click in the field to open a dialog in which you can change the style of the curve:



Right-clicking a table entry opens its **context menu** which provides the following functions:

- ***Delete***
  Deletes the selected curve definition.
- ***Edit X Data*** / ***Edit Y Data***
  Opens the dialog ***Select Data to be Recorded*** (see below) where you can change the data for the curve's X / Y axis.
- ***Move to Y2 axis*** / ***Move to Y axis***
  Assigns the selected curve definition to the Y2 / Y axis of the graph.

  A curve's Y axis assignment can also be changed via drag & drop: Click in the leftmost table column of the curve you want to move, and, with the left mouse button pressed, drag the entry to the desired axis.

For the definition of new standard and custom curves the buttons ***Add Curve...*** and ***Add Custom Curve...*** are available (see below). Use custom curves if you want to define actions to be performed by PIMikroMove to the recorded data immediately after reading it from the controller. The results of these actions are only present on the host PC but not in the data recorder tables of the controller.

The recording settings can be edited by clicking the button ***Edit X Data*** or ***Edit Y Data***. This opens the dialog ***Select Data to be Recorded*** (see below) for the corresponding axis, in which you can change the record option and axis assignment. Changed settings will always be applied to the next recording, but not to the data which was already recorded.

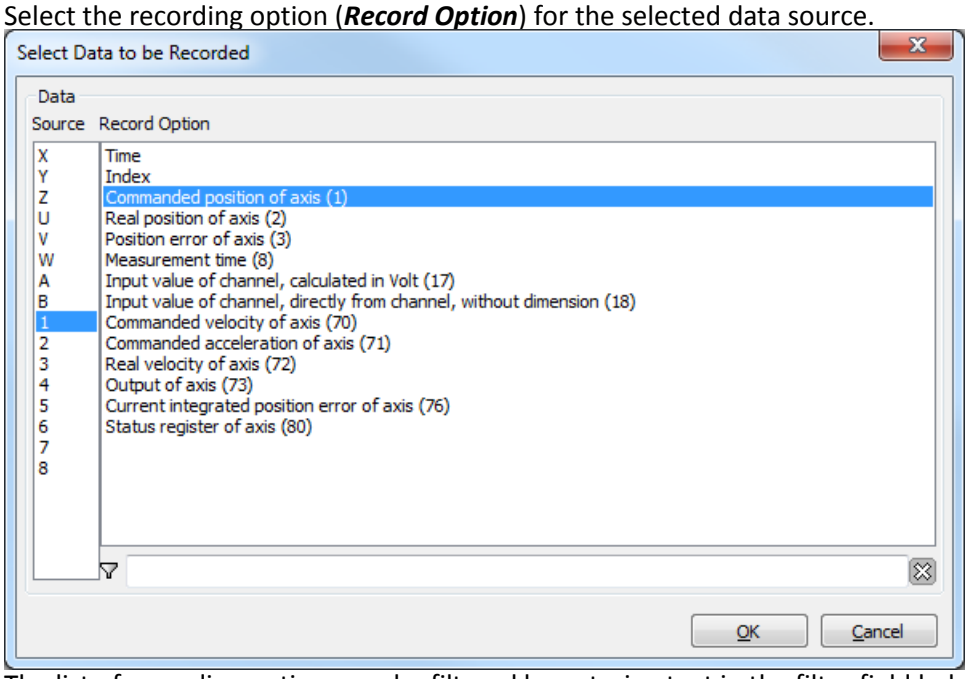
To delete a curve definition, mark the line in the table and click ***Delete***. To delete all curve definitions click the button ***Delete All***.

Reload from Controller sets the recording configuration to the current controller settings.

Adding standard curves

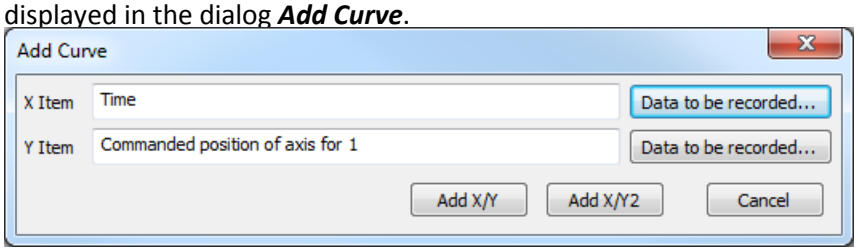
1. Click the button ***Add Curve...***. This opens the dialog ***Add Curve*** as well as the dialog ***Select Data to be Recorded***.
2. In the dialog ***Select Data to be Recorded*** define the recording options:
   a) Select a data source (***Source***), e.g. a controller axis, an input channel, an analog input line.

b) Select the recording option (***Record Option***) for the selected data source.



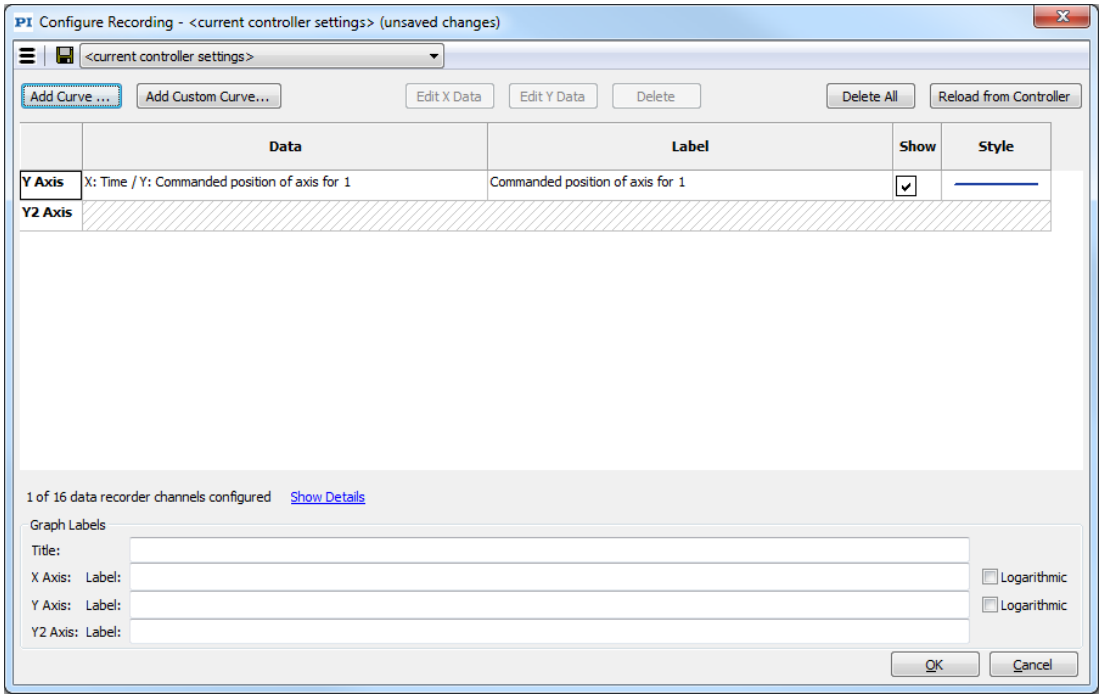
The list of recording options can be filtered by entering text in the filter field below the list.

c) Click ***OK***. The dialog ***Select Data to be Recorded*** is closed and the selected data is displayed in the dialog ***Add Curve***.



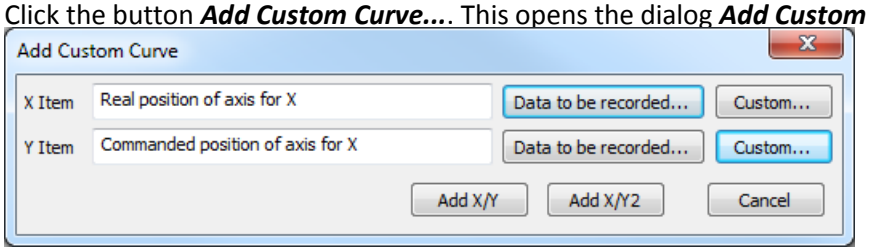
3. In the dialog ***Add Curve*** click ***Add X/Y*** to assign the selected data to the X axis and the left Y axis of the graph. To assign the data to the X axis and the right Y axis of the graph, click ***Add X/Y2***. The dialog ***Add Curve*** is closed and the newly defined curve is shown in the table.

Result:

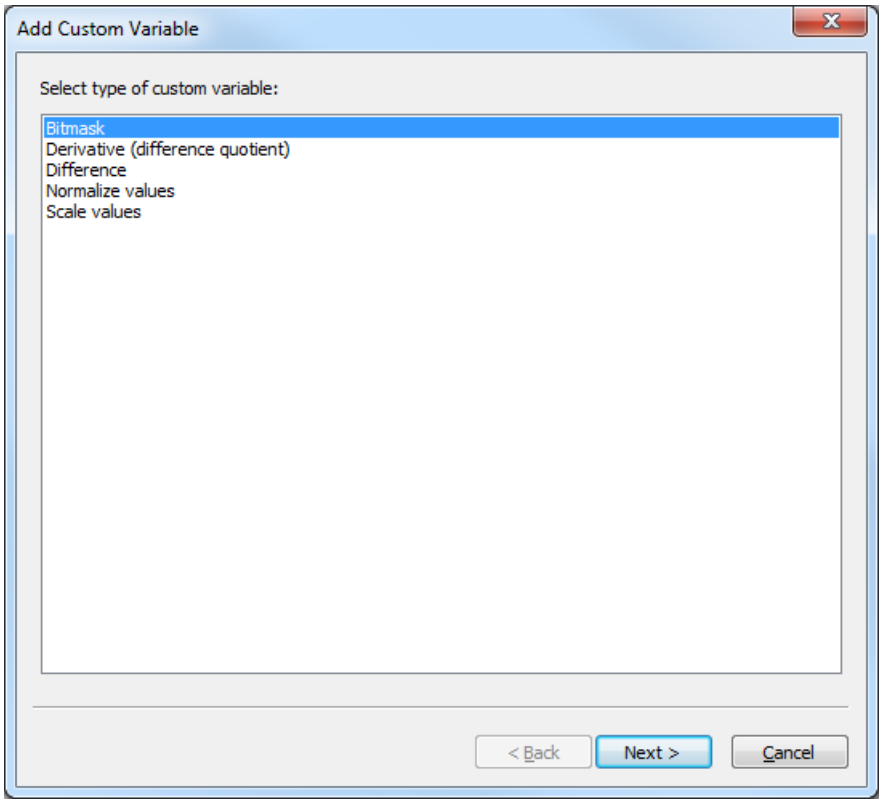


Adding custom curves

1. Click the button ***Add Custom Curve...***. This opens the dialog ***Add Custom Curve***.



2. Click the button ***Data to be recorded...*** to define recording options. This opens the dialog ***Select Data to be Recorded*** for the corresponding axis of the graph.
3. In the dialog ***Select Data to be Recorded*** define the recording options:
   a) Select a data source (***Source***), e.g. a controller axis, an input channel, an analog input line.
   b) Select the recording option (***Record Option***) for the data source.
   c) Click ***OK***. The dialog ***Select Data to be Recorded*** is closed and the selected data is displayed in the dialog ***Add Custom Curve***.
4. In the dialog ***Add Custom Curve*** click the button ***Custom...*** to define actions to be performed by PIMikroMove to the recorded data. This opens the dialog ***Add Custom Variable*** for the corresponding axis of the graph.



5. In the dialog ***Add Custom Variable*** define the custom variable:

   a) Select the type of variable to be applied to the recorded data:

   – ***Bitmask***: Bitmask filter to be applied to the recorded data. Application of this filter makes sense only if the recorded data is bit-mapped, e.g. if you have recorded the content of the status register.

   – ***Derivative (difference quotient)***: Derivative of the recorded data to be calculated. The values of the recorded data will be calculated as difference quotient: d[value of first component] / d[value of second component]. Note that you can also calculate derivatives later using the corresponding button in the data recorder window's data toolbar (p. 68).

   – ***Difference***: Difference of the recorded data to be calculated. The values of the recorded data will be calculated as difference: value of first component - value of second component. Note that you can also perform arithmetic operations on the recorded data later using the corresponding button in the data recorder window's data toolbar.

   – ***Normalize values***: Normalization of recorded data values using a selectable method:

     ▫ Normalize to -1...1 by maximum
     The values are normalized to +/-1. The maximum value corresponds to 1.
     Example: If the value range of the data is -28 to 75, each value will be multiplied by 1/75.

     ▫ Normalize to -100...100 by maximum
     The values are normalized to +/-100. The maximum value corresponds to 100.

Example: If the value range of the data is -28 to 75, each value will be multiplied by 100/75.

- Normalize to -1...1 by max value
  The values are normalized to +/-1, related to the specified <max value>. The maximum value corresponds to <max value>.
  Example: If the value range of the data is -28 to 75, each value will be multiplied by (<max value>)/75.
- Normalize to -100...100 by max value
  The values are normalized to +/-100, related to the specified <max value>. The maximum value corresponds to <max value>.
  Example: If the value range of the data is -28 to 75, each value will be multiplied by (100*<max value>)/75.

– ***Scale values***: Scaling of recorded data values:

- Scale with value: The values are scaled by a definable scaling factor.
- Invert (scale with -1): The values are scaled by the scaling factor -1.

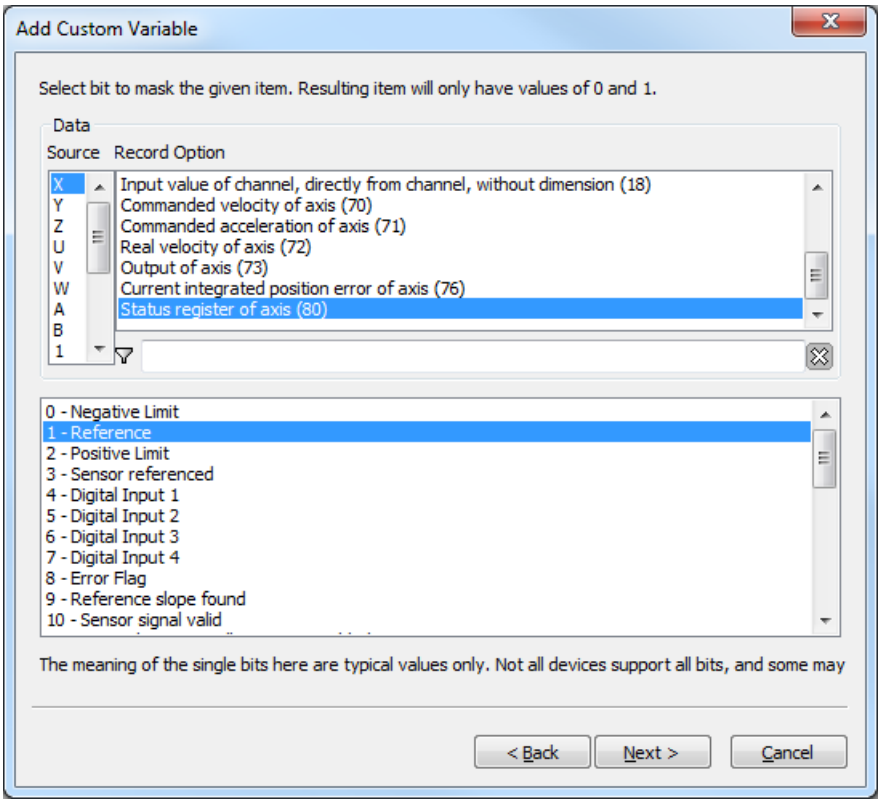
a) After having selected the type of variable click ***Next >***. The dialog switches to a page on which details for the selected variable have to be specified.

6. On the detail page of the ***Add Custom Variable*** dialog specify the details for the selected custom variable:

   a) Specify the variable-specific details:

   – **Bitmask**

   - Select a data source (***Source***).
   - Keep the preset ***Record Option*** "Status register of axis", or select another option that returns bit-mapped data.
   - Select the status bit you want to create a separate curve for.

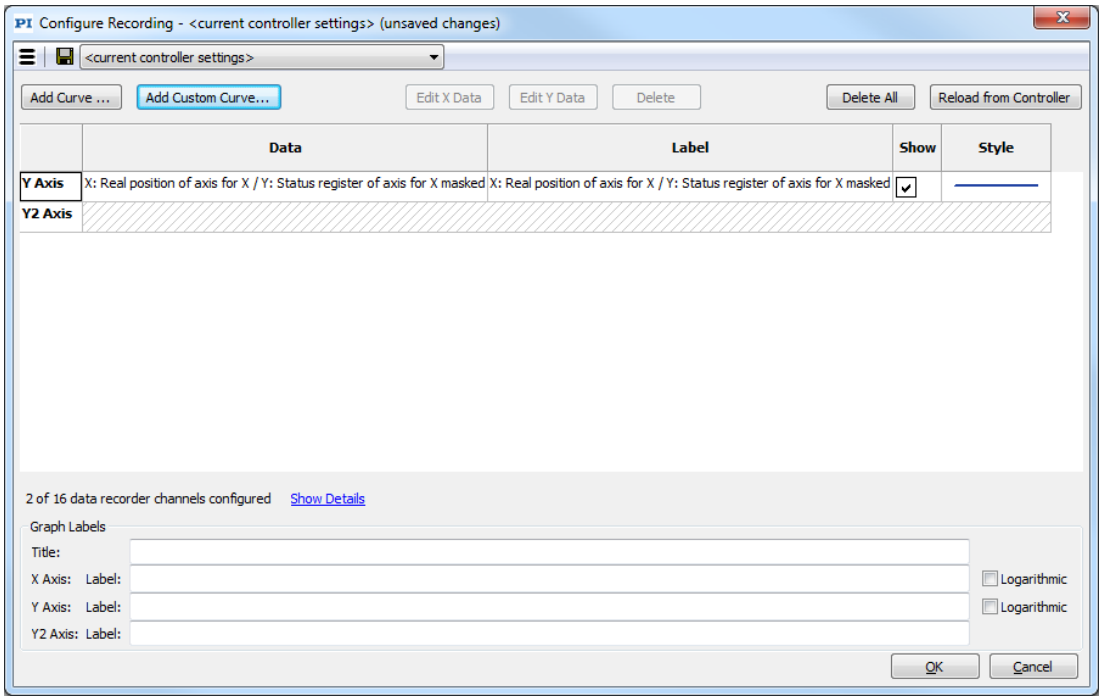


– **Derivative (difference quotient)**
  - Select a ***Source*** and ***Record Option*** to get the data for the first component.
  - Select a ***Source*** and ***Record Option*** to get the data for the second component.
– **Difference**
  - Select a ***Source*** and ***Record Option*** to get the data for the first component.
  - Select a ***Source*** and ***Record Option*** to get the data for the second component.
– **Normalize values**
  - Select a ***Source***.
  - Select a ***Record Option***.
  - Select a method for normalizing the recorded data and, for a normalization by max. value, enter the maximum value.
– **Scale values**
  - Select a ***Source***.
  - Select a ***Record Option***.
  - Select a method for scaling the recorded data and, for the method ***Scale with value***, enter the scaling factor.

a) Click ***Next >***. The dialog switches to a page on which you can complete the definition of the custom curve by specifying a name for it.

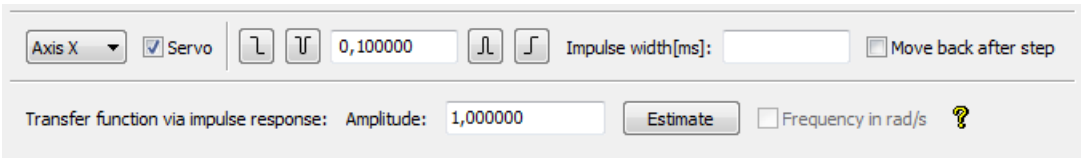
7. Specify a name for the custom curve on the completion page of the ***Add Custom Variable*** dialog:
   a) Keep or change the proposed name for the custom curve.
   b) Click ***Finish***. The dialog ***Add Custom Variable*** is closed and the selected data is displayed in the dialog ***Add Custom Curve***.
8. In the dialog ***Add Custom Curve*** click ***Add X/Y*** to assign the selected data to the X axis and the left Y axis of the graph. To assign the data to the X axis and the right Y axis of the graph, click ***Add X/Y2***. The dialog ***Add Custom Curve*** is closed and the newly defined custom curve is shown in the table.

Result:



4.5.2 Perform Step or Impulse Measurements

Via the functions of the ***Data Recorder*** window shown below you can start a step or impulse response measurement. The recorded data will be read and displayed after the recording has finished.



Control description, from left to right:

- Axis selection field: Select the axis for which the measurement is to be performed.

- ***Servo*** checkbox: Select the servo state. Depending on the controller, the measurement can be done in closed-loop or open-loop operation (i.e. with servo on or off).
- Start buttons for step and impulse with negative offset: Start motion in negative direction and recording. Buttons are disabled if the corresponding action is not supported by the controller.
- Offset input field: Enter the amplitude to be used for the step or impulse. The unit of the amplitude in open-loop and closed-loop operation depends on the controller. See the User manual of the controller for more information.
- Start buttons for step and impulse with positive offset: Start motion in positive direction and recording. Buttons are disabled if the corresponding action is not supported by the controller.
- ***Impulse width*** field: For an impulse measurement specify the impulse width in milliseconds.
- ***Move back after step*** checkbox: Check if you want the axis to move back to the starting position after the motion was performed.

Transfer function via impulse response:

- Specify the impulse height in the ***Amplitude*** field.
- Use the button ***Estimate*** to calculate the resulting output signal via the transfer function.
- For the representation of the frequency as angular frequency you can specify that the result of the transfer function is given as ***Frequency in rad/s***.

4.5.3 Start Recording Without Motion

If supported by the controller, you can start recording without any axis motion using the ***Record now*** button. PIMikroMove will then send a predefined suitable command to the controller. The recorded data will be read and displayed after the recording has finished.

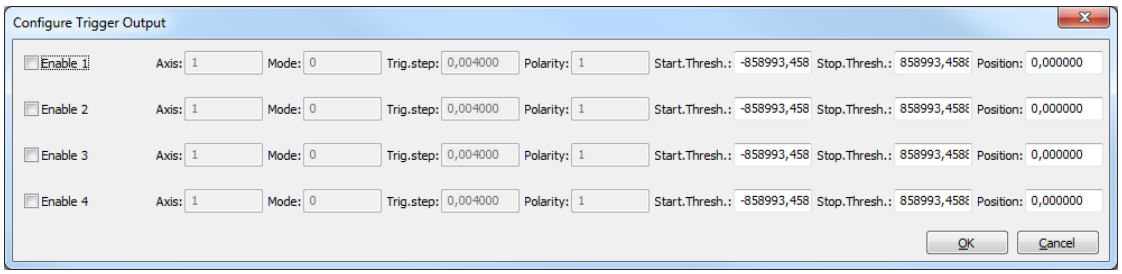
4.5.4 Read Data from Controller and Apply Actions to the Data

To read the data and display it in the graphics pane, click the ***Read*** button. Reading can take some time depending on the number of points to be read. For data reading it is irrelevant if recording was started inside or outside of the ***Data Recorder*** window.

If you have configured actions to be applied to the recorded data, e.g. calculation of derivatives, sums or differences, or application of bitmasks, they are (re)applied when you click ***Read***. See "Configure Recording" (p. 127) for more information regarding such actions performed by PIMikroMove.

4.6 Configure Trigger Output

The ***Configure trigger output...*** menu item is on the controller menu of supporting controllers. It gives access to a separate ***Configure Trigger Output*** window where you can specify trigger conditions for the digital output lines present on the controller.



INFORMATION

In certain controllers, e.g. C-867, C-663, programmable digital outputs and trigger outputs share the same circuitry. Make sure that the same lines are not activated for both functions. Digital I/O is configured from the ***View*** menu. Should lines be called using alphabetic characters in the ***Digital I/O*** Window, then A is the same as line 1 in the ***Configure Trigger Output*** window, B is 2, etc.

Each line in the ***Configure Trigger Output*** window is dedicated to one digital output line of the controller. The fields shown depend on the controller (see the description of the CTO command in the controller's user manual for more information):

- If ***Enable*** checkboxes are present, the trigger settings only become active for a digital output line when the box is checked.
- The ***Mode*** setting specifies the trigger mode to be used. The available trigger modes depend on the controller. The mode specified determines which of the other fields are required / ignored:

  0 = Position distance; axis-related. With this trigger mode, a trigger pulse is output whenever the axis has covered the ***Trigger step*** distance.

  1 = Position distance + wait time; axis-related. With this trigger mode, a trigger pulse is output a specified time (***Delay time***) after the axis has covered the ***Trigger step*** distance.

  2 = OnTarget; axis-related. With this trigger mode, the on-target status of the selected axis determines the state of the digital output line (active-high logic; this status can also be read with the ONT? command in the ***Command entry*** window).

  3 = MinMaxThreshold; axis related. With this trigger mode, values for ***Min.Threshold*** and ***Max.Threshold*** must be defined. When the axis position of the selected axis is inside the band specified by the ***Min.Threshold*** and ***Max.Threshold*** values, the digital output line is set high, otherwise it is set low.

  4 = Generator Trigger; related to the wave generator output. With this trigger mode, the trigger actions on the digital output line must be defined for certain waveform points using the TWS command in the ***Command entry*** window.

5 = MotionError; the ***Axis*** setting is irrelevant with this trigger mode. The selected trigger line becomes active when a motion error occurs. The line will stay active until the error code is reset to 0 (by a query).

6 = InMotion; axis related. The selected trigger line is active as long as the selected axis is in motion (the in-motion state can also be read with the #4, #5 or SRG? commands in the ***Command entry*** window).

7 = Position+Offset; axis related. With this trigger mode, the first trigger pulse is written when the axis has reached the position given by ***Position***. The next trigger pulses each are written when the axis position equals the sum of the last valid trigger position and the increment value given by ***Trigger step***. Trigger output ends when the axis position exceeds the value given by ***StopThreshold***. The sign of the ***Trigger step*** value determines for which direction of motion trigger pulses are to be output. Trigger processing is done by the DSP of the controller.

8 = SingleTrigger; axis related. With this trigger mode, a trigger pulse is written when the axis has reached the trigger position given by ***Position***.

9 = HardwareTrigger; axis related. This trigger mode is the same as the Position+Offset mode, but done by the FPGA of the controller and therefore much faster. This trigger mode can only be selected for output lines 1 and 2. Output line 1 must be connected to the first axis and output line 2 to the second axis via the ***Axis*** fields.

- For the axis-related trigger modes, the digital output line is associated with the controller axis whose identifier is entered in the ***Axis*** field.

See the controller's user manual for more information (e.g. command descriptions, I/O connector pinout and examples).

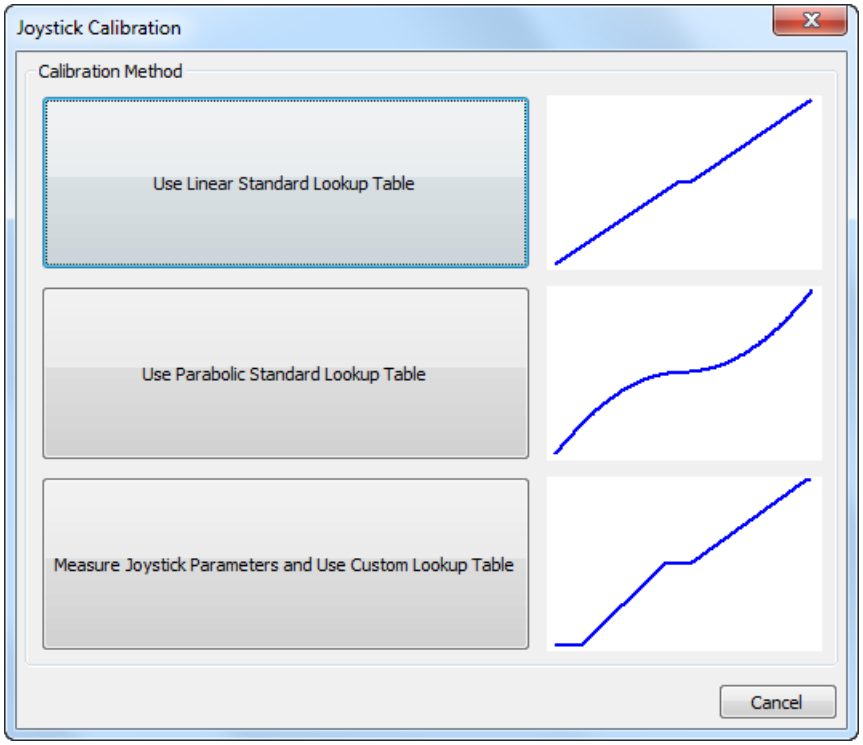
4.7 Joystick Calibration

The ***Calibrate controller joystick...*** menu item on the controller menu of supporting controllers opens the ***Joystick Calibration*** window for the selected HID (human interface device). In this window, you can "calibrate" the individual axes of the HID by changing their lookup tables in the controller as follows:

- Select the linear or the parabolic standard lookup table. The parabolic lookup table offers more sensitivity during slow motions.
- Measure HID parameters and write a custom lookup table.

INFORMATION

Only HIDs directly connected to the controller are affected here. If you have an HID connected to the host PC see "Configure PC Joysticks Window" (p. 93) for how to test and calibrate it.



The calibration of HID axes is necessary in the following cases:

- The HID response behavior is not suitable for your application.
- You are using the Z-axis of a C-819.30 joystick: This joystick axis cannot be used with the linear or parabolic standard lookup tables and must be calibrated via the measurement of the joystick parameters which are then written to a custom lookup table. Calibration must be repeated whenever the Z-axis of the joystick is connected to another controller.

INFORMATION

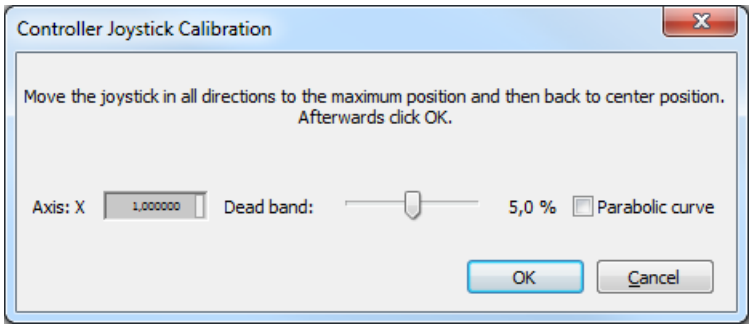
If the HID provides operating elements for the mechanical adjustment of its axes, use them before you calibrate the HID axes with PIMikroMove. For more information, see the user manual of your controller.

Proceed as follows for calibration:

1. Select the HID axes to be calibrated by checking the corresponding boxes in the ***Joystick Axes*** pane of the ***Joystick Calibration*** window.

   Depending on the controller, multiple HID axes may share the same lookup table. When one of the HID axes is selected in the ***Joystick Axes*** pane, all other joystick axes sharing the same lookup table become grayed out.

2. Select the preferred calibration method using the appropriate button in the ***Calibration Method*** pane of the ***Joystick Calibration*** window:
   - If you want to use the linear lookup table type for the joystick axes, click on ***Use Linear Standard Lookup Table***. This loads the corresponding lookup table type in the controller, and the calibration is finished.
   - If you want to use the parabolic lookup table type for the joystick axes, click on ***Use Parabolic Standard Lookup Table***. This loads the corresponding lookup table type in the controller, and the calibration is finished.
   - If you have connected the Z-axis of an C-819.30 joystick device or if you just want to map the joystick behavior to a custom lookup table on the controller, click on ***Measure Joystick Parameters and Use Custom Lookup Table***. This opens the ***Controller Joystick Calibration*** window.



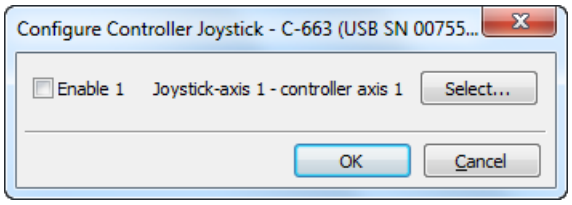
3. In the ***Controller Joystick Calibration*** window, proceed as follows:
   a) Move the axis of the human interface device to all extreme positions. The custom lookup table values are determined in this way.
   b) Let go of the axis.
   c) If you want to change the neutral area of the axis (i.e., the area around the middle position of the axis where no change in the controlled motion variable is triggered), set the ***Dead band*** slider in the window correspondingly.
   d) If the values in the user-defined lookup table are to describe a parabolic waveform, click the ***Parabolic curve*** checkbox.
   e) Click ***OK*** to write the lookup table values to the volatile memory of the controller. For each lookup table that is written, a separate window opens where you can monitor the write process. Calibration is finished after all windows were closed automatically.

4.8 Configure Controller Joystick

The ***Configure controller joystick(s)...*** menu item on the controller menu of supporting controllers opens the ***Configure Controller Joystick*** window where you can activate/deactivate direct HID (human interface device) control for individual controller axes.

INFORMATION

Only HIDs directly connected to the controller(s) are affected here. If you have an HID connected to the host PC see "Configure PC HID Control (p. 93)".



NOTICE

Uncontrolled motion!

If no physical HID is connected or if the HID's axes are not properly calibrated, uncontrolled motion of the controller axes assigned to that HID can occur. Uncontrolled motion can damage the positioner or the application.

- Do **not** activate an HID with no physical HID connected.
- Make sure that the HID is properly calibrated. See "Calibrate Controller Joystick" (p. 137) for details.

In the ***Configure Controller Joystick*** window, each ***Enable*** checkbox is dedicated to one HID connected to the controller.

Selecting/deselecting the ***Enable*** checkbox for an HID activates/deactivates HID control for all controller axes assigned to the axes of that HID. The assignment of controller axes to HID axes is displayed next to the ***Enable*** checkbox(es) and can be changed using the ***Select…*** buttons.

All settings not supported by the controller are disabled. Changed settings are applied after clicking ***OK***. HID control remains active until you uncheck the associated ***Enable*** checkbox or exit PIMikroMove.

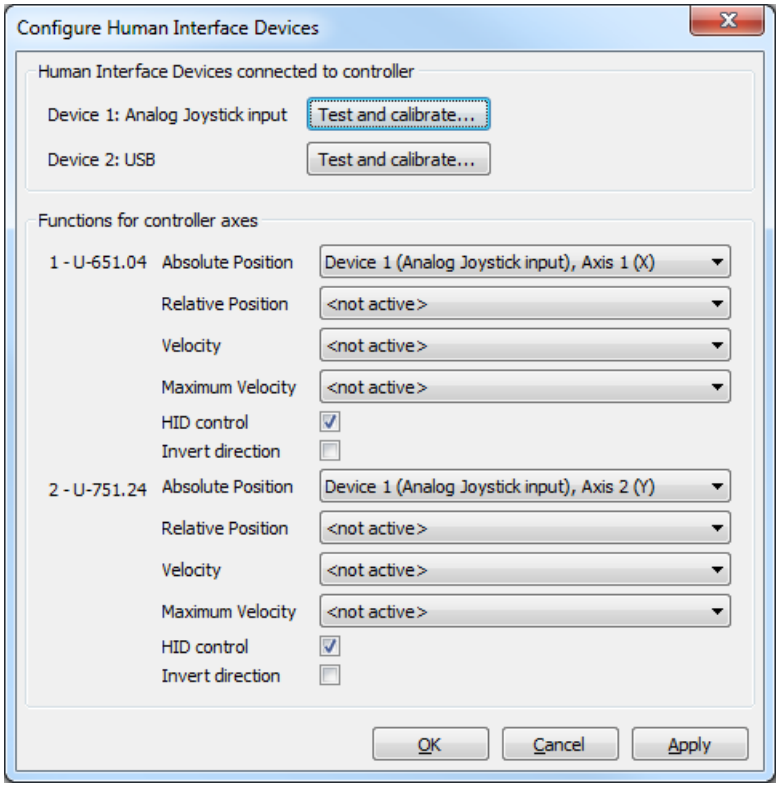
See the user manual of your controller for more information regarding HID control.

4.9 Configure Human Interface Devices

The ***Configure Controller HIDevice(s)*** menu item on the controller menu of supporting controllers opens the ***Configure Human Interface Devices*** window where you can activate/deactivate direct HID control for individual controller axes.

INFORMATION

Only HIDs directly connected to the controller(s) are affected here. If you have an HID connected to the host PC see "Configure PC HID Control (p. 93)".



The ***Configure Human Interface Devices*** window consists of two panes:

- ***Human Interface Devices connected to controller***

  The upper pane contains all the HIDs connected to the controller. They are displayed with their respective device IDs. You can test and calibrate an HID via the respective button ***Test and calibrate...***.

- ***Functions for controller axes***

  For each axis of the controller, the lower pane contains the motion variables that can be controlled via an axis of an HID, as well as checkboxes to activate HID control and to invert the direction of motion. Via these options, HID control for the controller's axes can be configured and activated.

NOTICE

Uncontrolled motion!

If no physical HID is connected or if the HID's axes are not properly calibrated, uncontrolled motion of the controller axes assigned to that HID can occur. Uncontrolled motion can damage the stage or the application.

- ➢ Do **not** activate an HID with no physical HID connected.
- ➢ Make sure the HID is properly calibrated.

See the user manual of your controller for more information regarding HID control.

Configuring and activating HID control for a controller axis

1. From the drop-down list of the motion variable to be controlled, select the HID axis you want to use for control.
2. Activate HID control for the axis by checking the ***HID control*** checkbox.
3. If the direction of motion is to be inverted during HID control, check the ***Invert direction*** checkbox.
4. Send the settings for setting up HID control to the controller by clicking the ***OK*** button.

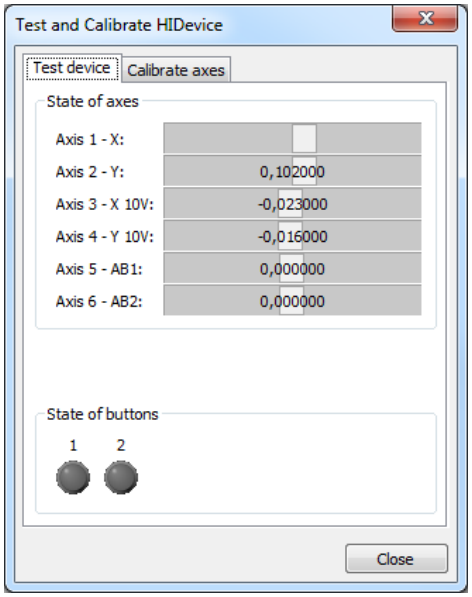
4.9.1 Test and Calibrate HIDevice

The button ***Test and calibrate...*** in the ***Configure Human Interface Devices*** window opens the dialog ***Test Calibrate HIDevice*** for the connected HID. This dialog fulfills two functions:

- Testing of HIDs
- Calibration of HID axes

Testing HIDs

On the ***Test device*** tab of the window, the operating elements of the HID can be tested.



To test the operating elements of the connected HID:

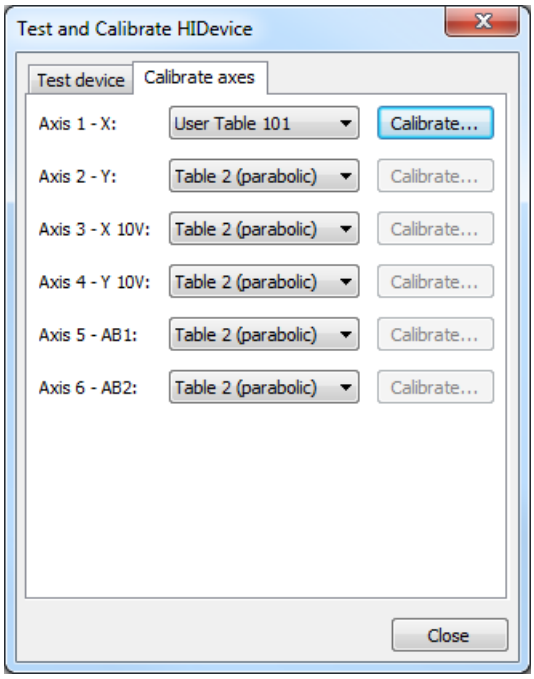
- Move the axes of the HID and observe the status displays in the ***State of axes*** area.
- Press the buttons of the HID and observe the status displays in the ***State of buttons*** area.

Calibrating HID Axes

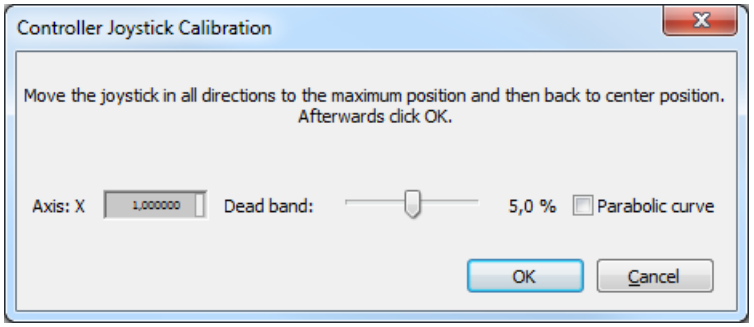
On the ***Calibrate axes*** tab of the window, the axes of the connected HID can be calibrated.

To calibrate an HID axis proceed as follows:

1. Select a lookup table with the designation "User Table" from the axis' selection field.



2. Calibrate the HID axis by filling the assigned user-defined lookup table with values:
   a) Click the axis' ***Calibrate...*** button to open the ***Controller Joystick Calibration*** window.

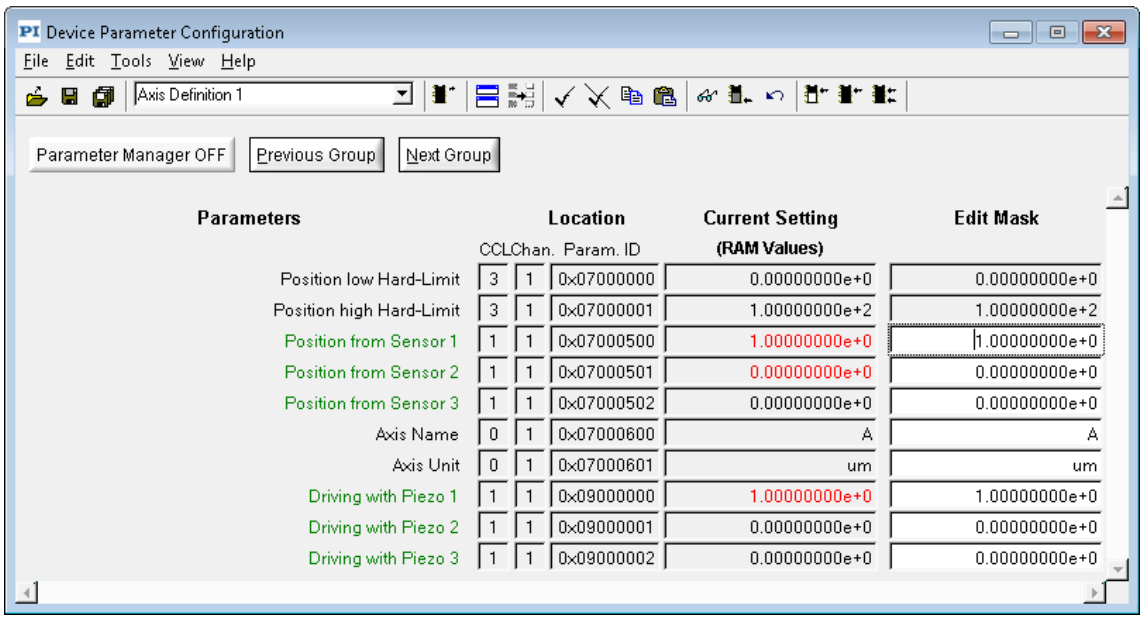


   b) Move the axis of the human interface device to all extreme positions. The custom lookup table values are determined in this way.
   c) Let go of the axis.
   d) If you want to change the neutral area of the axis (i.e., the area around the middle position of the axis where no change in the controlled motion variable is triggered), set the ***Dead band*** slider in the window correspondingly.
   e) If the values in the user-defined lookup table are to describe a parabolic waveform, click the ***Parabolic curve*** checkbox.
   f) Click ***OK*** to write the lookup table values to the volatile memory of the controller.

4.10 Device Parameter Configuration

The ***Parameter Configuration...*** menu item on the controller menu of supporting controllers opens the ***Device Parameter Configuration*** window in which you can check, modify and save controller parameters. Working with the ***Device Parameter Configuration*** window is helpful e.g. when you want to adapt the controller to your application.

The individual parameters apply to different elements, e.g. to the entire system, to individual axes or to individual input/sensor channels and output/piezo channels (for parameter details see the controller's user manual). The parameters available depend on the controller firmware. In the ***Device Parameter Configuration*** window, they are grouped under different headings. In the toolbar, the ***Parameter Group*** selection box permits selecting the group of parameters that will be displayed in the window.



Parameters	Location			Current Setting	Edit Mask
	CCL	Chan.	Param. ID	(RAM Values)	
Position low Hard-Limit	3	1	0x07000000	0.00000000e+0	0.00000000e+0
Position high Hard-Limit	3	1	0x07000001	1.00000000e+2	1.00000000e+2
Position from Sensor 1	1	1	0x07000500	1.00000000e+0	1.00000000e+0
Position from Sensor 2	1	1	0x07000501	0.00000000e+0	0.00000000e+0
Position from Sensor 3	1	1	0x07000502	0.00000000e+0	0.00000000e+0
Axis Name	0	1	0x07000600	A	A
Axis Unit	0	1	0x07000601	um	um
Driving with Piezo 1	1	1	0x09000000	1.00000000e+0	1.00000000e+0
Driving with Piezo 2	1	1	0x09000001	0.00000000e+0	0.00000000e+0
Driving with Piezo 3	1	1	0x09000002	0.00000000e+0	0.00000000e+0

The ***Chan.*** column of the ***Device Parameter Configuration*** window always shows the index of the axis, input/sensor channel or output/piezo channel related to the parameter. These indices are also present in the group headings which are used for parameter group selection. Example: The first axis is named A, but the axis definition parameters are grouped under the ***Axis Definition*** 1 heading, and the ***Chan.*** column contains the value 1 instead of A.

The following buttons are available for parameter list control and parameter management:

- ***Previous Group*** and ***Next Group*** buttons, which permit switching between parameter groups faster than using the ***Parameter Group*** selection box.
- ***Parameter Manager***: indicator/button, when OFF indicates that the ***Parameter Manager*** functions are hidden and the parameter list is displayed as in the figure above. Clicking it reverses the situation, displaying the ***Parameter Manager*** controls and hiding the parameter list. See "Parameter Manager" (p. 146) for details.

The parameter values displayed are those from the controller's volatile memory (RAM values), i.e., the currently active settings.

4.10.1 Load / Edit / Save Parameter Values

NOTICE

Wrong parameter values may lead to improper operation or damage of your hardware!

➢ It is strongly recommended to save the parameter values of the controller to a file on the host PC before you make any changes in non-volatile memory. This way the original settings can be restored if the new parameter settings will not prove satisfactory. See the User manual of the controller for detailed instructions on how to create backup files.

Settings can only be edited in the ***Edit Mask*** column of the ***Device Parameter Configuration*** window. You have several options to handle the parameter values via the content of the ***Edit Mask*** column. Use the ***Parameter Manager*** controls (p. 146) or the items/icons of menu bar and toolbar (p. 147) for the following actions:

- You can load/save parameter values from/to a file on the host PC. Loaded parameter values will be written to the ***Edit Mask***, and only the values in the ***Edit Mask*** column can be saved to a parameter file. To save the current settings or the default settings you have to copy them first to the ***Edit Mask*** and then save to a file.
- To write a new value to the controller, write it in the corresponding ***Edit Mask*** field, select it and then use the appropriate write function. Multiple selections are possible. Only the selected ***Edit Mask*** values will be written to the controller.

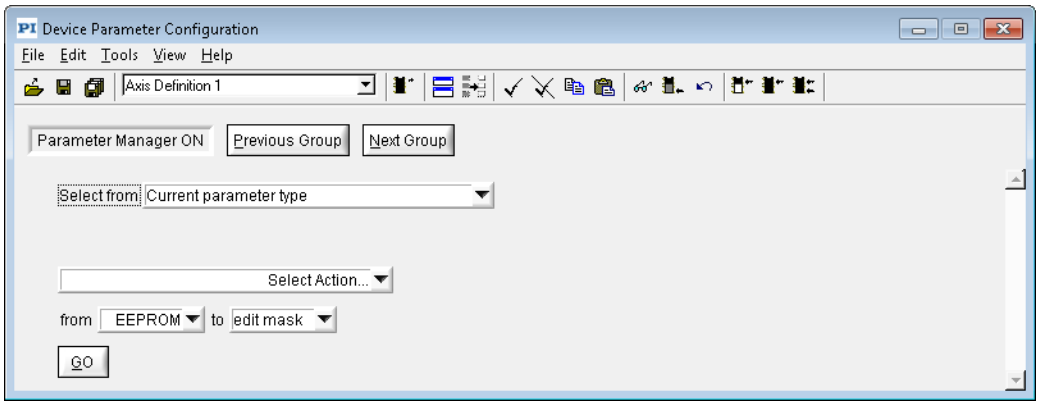
  For test purposes, write the value as current setting (i.e. to RAM which is volatile memory). After that, the corresponding ***Current Setting*** value will be displayed in red.

  If you write the value as default setting, it will be saved in the controller's non-volatile memory (EEPROM) where it becomes the power-on default.
- You can compare the parameter value from volatile memory (RAM) with the one from non-volatile memory (EEPROM) using the ***Compare Current Setting with Default Setting*** menu item. Parameters that are different in RAM and EEPROM when you perform this comparison are then displayed in red in the ***Current Setting*** column. This comparison is automatically performed in the ***Device Parameter Configuration*** window whenever the parameters are uploaded from the controller.

Many parameters have a password protection. The protection status of the individual parameters can be seen in the ***CCL*** column of the ***Device Parameter Configuration*** window. You can change only parameters with ***CCL*** values 0 or 1. Parameters with ***CCL*** values larger than 1 can be modified by PI service personnel only. While changing ***CCL*** 0 parameters requires no password, for changing ***CCL*** 1 parameters you have to enter "advanced" in the ***Password*** dialog which pops up automatically.

4.10.2 Parameter Manager

Using the ***Parameter Manager*** controls, you can easily copy the values of selected parameter types, and it is also possible to copy the default settings (EEPROM values) to the ***Edit Mask***, which is not possible with the menu items and toolbar icons described in "Menu Bar and Toolbar" (p. 147).



Parameter Manager controls:

- ***Parameter Manager*** indicator/button when ON, indicates that the ***Parameter Manager*** controls are displayed and the parameter list is hidden; it also permits toggling back to the parameter list.
- From the ***Select from*** drop-down list, you can select the parameter type to work with, e.g. axis-related parameters or piezo-channel-related parameters.
- Further drop-down lists permit to define the action to perform, that is: copying the values of parameters from EEPROM, RAM or ***Edit Mask*** to RAM or ***Edit Mask***.

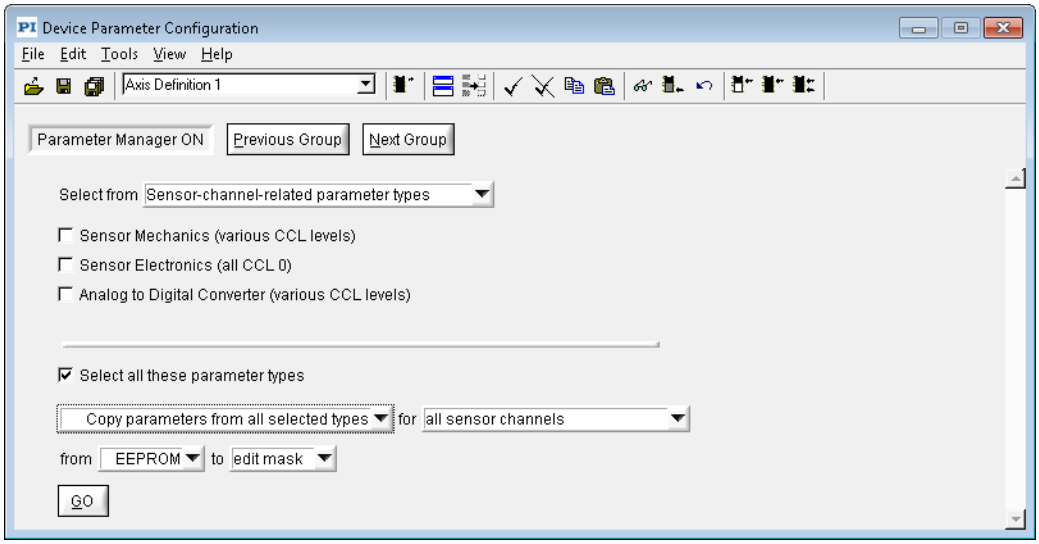
The window layout changes according to the item selected from the ***Select from*** drop-down list: The ***Current parameter type*** item refers to the item selected in the ***Parameter Group*** selection box (in the toolbar). If you select another item from the list, the window contains:

- Checkboxes for selecting all parameter types to take into account for the action to perform.
- The checkbox ***Select all these parameter types***, which permits taking all parameter types into account for the action to perform.

If the selection is neither ***Current parameter type*** nor ***All parameter types***, the window also contains a selection box permitting you to choose all or a single axis / piezo channel / sensor channel / wave generator, depending on the action to perform.

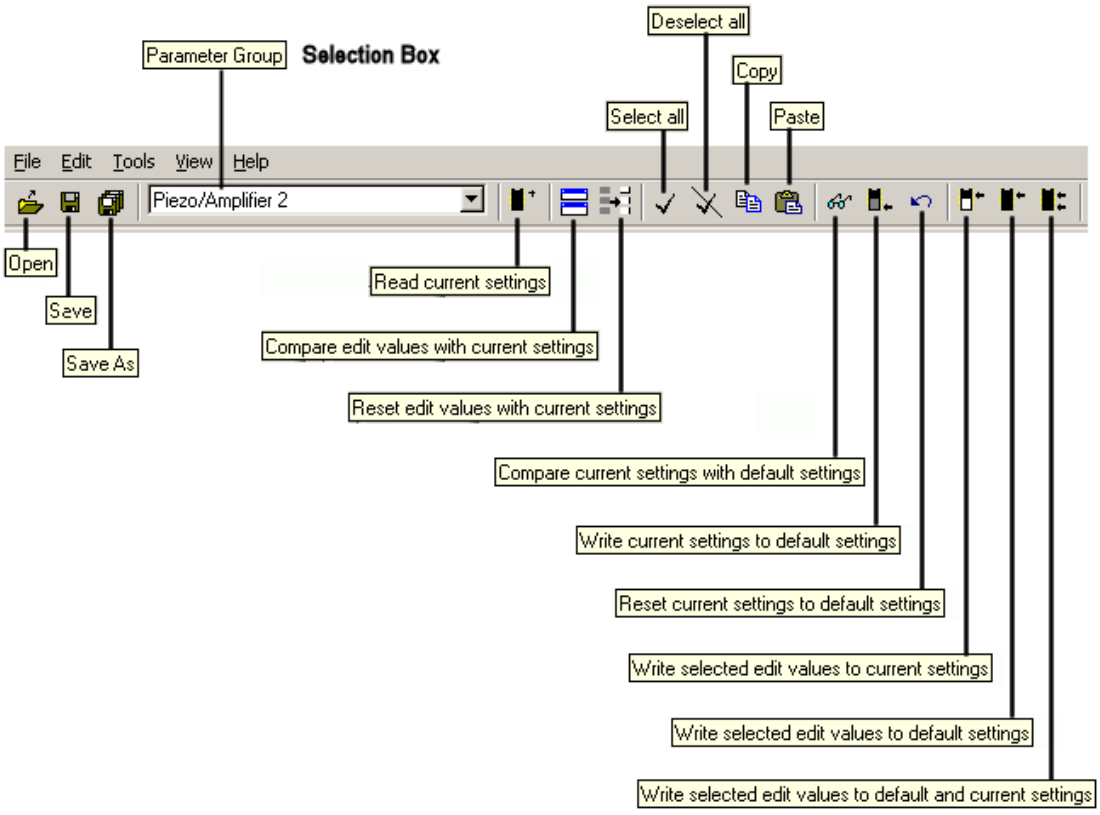
The selected action will be performed on the corresponding parameters when pressing the ***GO*** button.

An example of the ***Parameter Manager*** window configuration for copying all sensor-channel-related parameters from EEPROM to the ***Edit Masks*** is shown in the figure below.



4.10.3 Menu Bar and Toolbar

Most of the menu items are available on the toolbar, and all icons are available on the menus.



Menu bar and toolbar of the Device Parameter Configuration window

The items of the *File* menu are:

- ***Load parameter file*** permits loading all parameters from a parameter file (with the extension .pam) and writing them in the corresponding ***Edit Masks*** field.
- ***Load and select*** permits loading all parameters from a parameter file (with the extension .pam) and writing them in the corresponding ***Edit Mask*** fields. The corresponding ***Edit Mask*** values will be selected automatically.
- ***Save Edit Values As*** and ***Save Edit Values*** permit saving all parameters from the ***Edit Masks*** into a parameter file.
- ***Close*** permits closing the ***Device Parameter Configuration*** window.

The items of the *Edit* menu are:

- ***Copy*** permits copying the selected ***Edit Mask*** parameter values from the current parameter group to the clipboard.
- ***Paste*** permits writing the parameter saved in the clipboard to the same position in the ***Edit Mask*** of the current parameter group. The parameter group where the parameters have been copied from and the one where the parameter will be pasted must be compatible. If the parameter groups are not compatible a corresponding message will pop up.
- ***Select all*** and ***Deselect all*** permit selecting or deselecting all ***Edit Mask*** values. In both cases, this applies either to the current parameter group or to all parameters, depending on what you select in the associated submenu.

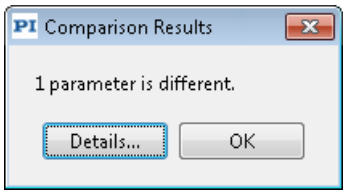
  Note: Individual parameter edit values can be selected by clicking it with the left mouse button or deselected by clicking it with the right mouse button. Selected parameters are displayed in white in a grey background. Only selected parameters will be used in write operations.
- ***Search…*** permits searching for a parameter via a part of its name, value or ID.
- ***Search Invalid Values…*** permits detecting invalid values from the ***Edit Mask*** and ***Current Setting*** fields (e.g. empty fields or—if present—"invalid" items from value selection boxes).

The results of ***Search…*** or ***Search Invalid Values…*** can be saved in a *.txt or a *.csv file.

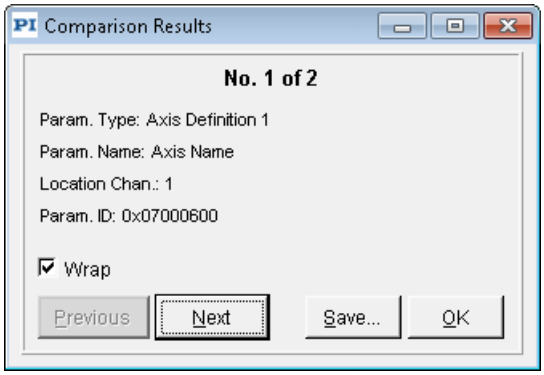
The items of the *Tools* menu are:

- ***Read Current Setting*** permits reading the RAM values from the controller for either the current parameter group or for all parameters, depending on what you select in the associated submenu:
- ***Write selected Edit Values*** permits writing values in the Edit Mask to the controller RAM or EEPROM or to both, depending on what you select in the associated submenu.
- ***Reset Edit Values with Current Setting*** permits resetting the values in the ***Edit Masks*** to the values from the ***Current Setting*** column. This applies either to the parameter that is currently activated with the cursor or to all parameters from the current parameter group, or to all parameters, depending on what you select in the associated submenu:

- ***Compare Edit Values with Current Setting…*** permits comparing the values in the ***Edit Mask*** with the RAM values for all parameters. At the end of this action, a dialog gives you access to precise information about the parameters that differ.



Pressing the ***Details…*** button opens a dialog box with more information about the first parameter that differs and a ***Next*** button for moving on to the next parameter (if there is any).

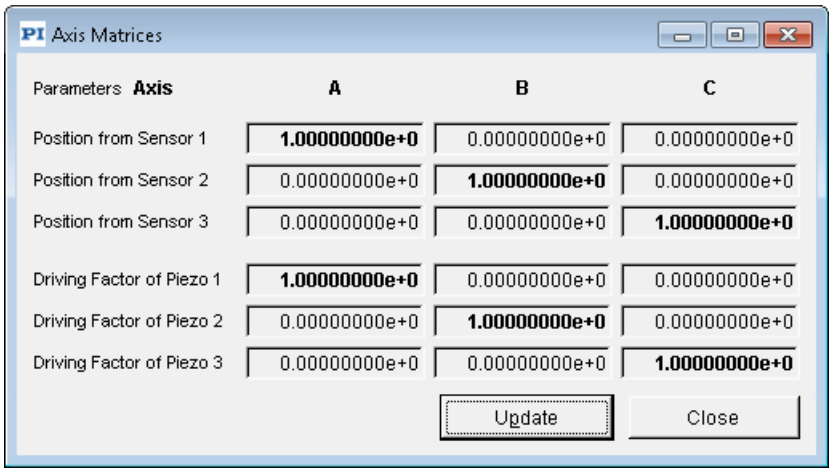


As you go through the different parameters, the parameter group of the respective parameter is selected in the ***Parameter Group*** selection box in the ***Device Parameter Configuration*** window, and the parameter in question is highlighted (selected), so that it can easily be found.

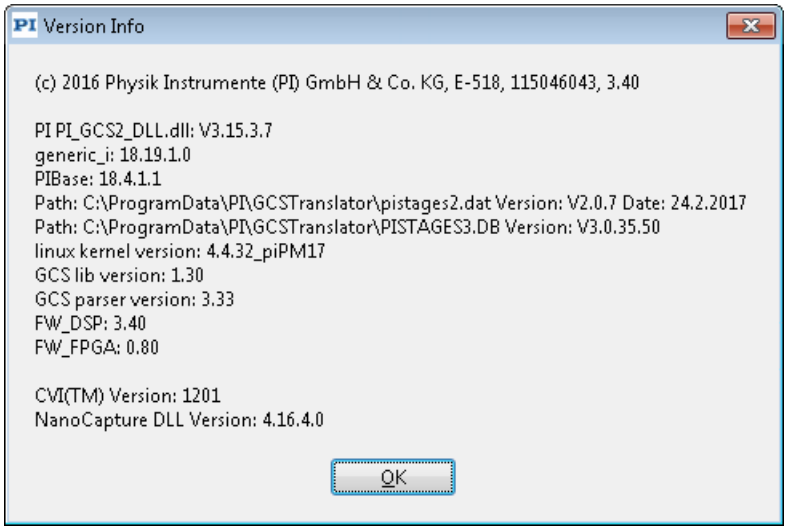
The results can be saved to a *.txt or a *.csv file.

- ***Compare Current Setting with Default Setting*** permits comparing RAM and EEPROM values for either the current parameter group or for all parameters, depending on what you selected in the associated submenu. The concerned current setting values are displayed in red if they differ from the corresponding default setting value.
- ***Reset Current Setting to Default Setting*** permits resetting the RAM values to the EEPROM values for all parameters.
- ***Write Current Setting to Default Setting*** permits writing the RAM values to the EEPROM for all parameters.

The ***View*** menu contains only one item, ***Axis Matrices***. It opens the ***Axis Matrices*** window where you can check (and edit, if supported by the controller) the assignment of sensor channels and piezo channels to the motion axes of the controller.

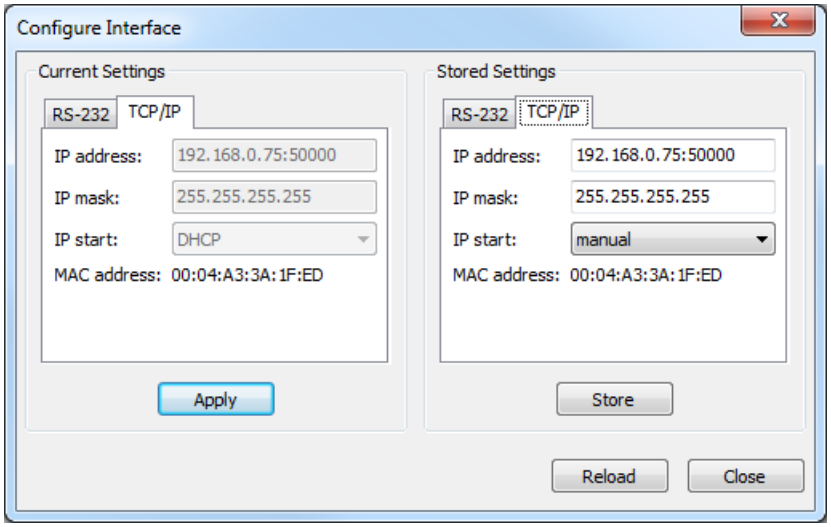


The ***Help*** menu also contains only one item, ***Version Info***. It pops up a window displaying the identification string obtained from the controller, including firmware version number and—if supported for the controller— the versions of the underlying drivers and libraries.



4.11 Configure Interface

The ***Configure interface...*** menu function in the controller menu of supporting controllers opens the ***Configure Interface*** window where you can specify the settings for the controller side of the communications interface.



Depending on the controller, you can specify the current settings and/or the power-on default settings.

A set of default communication parameters is stored on the controller. These values are loaded into the controllers RAM on power-up or reset and become the active settings. The default settings can be read and changed in the ***Stored Settings*** pane. To keep changes click the ***Store*** button. They become active only with the next power-up or reset.

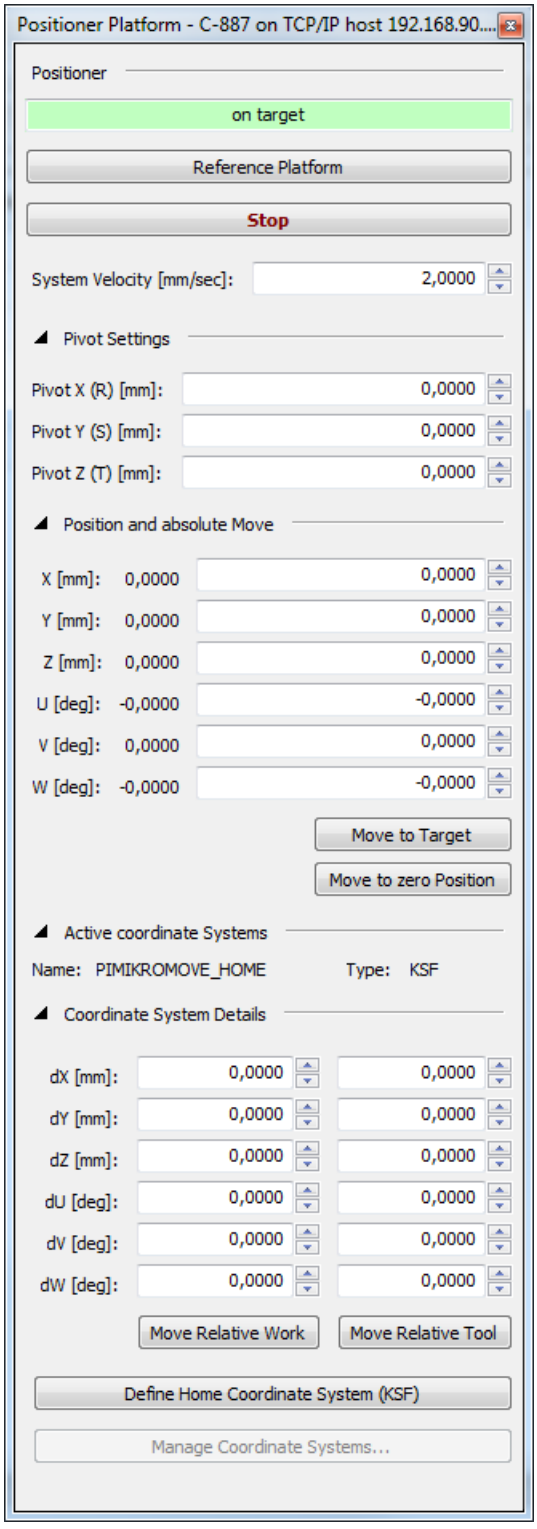
The current active settings can be changed in the ***Current Settings*** pane—the new settings become active immediately when you click the ***Apply*** button. The host PC may need to change its interface configuration to maintain communications - close the connection and reconnect with the new settings. When the controller is powered down, settings made in the ***Current Settings*** pane are lost.

The settings used for the current active communications and the last-stored default settings can be reloaded in both panes by clicking the ***Reload*** button.

See the controller's user manual for communications details and default settings.

4.12 Positioner Platform

For every parallel kinematics system (controller or simulation) connected to PIMikroMove, a separate ***Positioner Platform*** window by default is displayed in the main window. You can reopen the ***Positioner Platform*** window using the ***Show Positioner Platform Setting*** item of the controller menu.



The current state of the parallel kinematics positioner platform is shown in the status field, e.g. "on target".

To start a reference move of the parallel kinematics positioner platform use the ***Reference platform*** button. The reference move can also be started via the ***Start up axes…*** (p. 175) item of the controller menu or the ***Axis*** menu.

Using the ***Stop*** button, you can stop motion started by the ***Move to Target*** or ***Move to zero position*** buttons.

An ***Axis*** menu (p. 29) for all axes of the parallel kinematics positioner platform can be displayed from the ***Positioner Platform*** window by right-clicking on any free area in the window. The ***Clear Error*** item of this menu can be helpful when the state field indicates an error.

INFORMATION

During a motion of the parallel kinematics positioner platform, all controls of the ***Positioner Platform*** window except for the ***Stop*** button are disabled.

Motion settings

You can configure, start and stop motion of the parallel kinematics positioner platform as follows:

- ***System Velocity***: Set the system velocity if necessary:

  Enter a new value in the ***System Velocity*** field and press Enter or move the cursor to another field. This sends the value to the controller, and the text color changes from blue to black again. If a new value is not accepted by the controller, the old value reappears in the field.

- ***Pivot Settings***: Change the pivot point coordinates if necessary:

  Enter a new value in a ***Pivot …*** field and press Enter or move the cursor to another field. This sends the value to the controller, and the text color changes from blue to black again.

The pivot point can only be changed if the parallel kinematics positioner platform is not tilted (U, V and W must be 0). See the user manual for your parallel kinematics system for more information.

- ***Position and Absolute Move***: Start motion to variable target positions or to the zero position:

  – Start motion to variable target positions:
  Enter target positions for the axes of the parallel kinematics positioner platform in the corresponding ***<axis>*** fields and start the motion using the ***Move to Target*** button.
  If a node of the calculated trajectory or the target position cannot be reached, the motion is not started, and an error message is displayed.
  The ***<axis>*** fields and the ***Move to Target*** button are deactivated as long as the parallel kinematics positioner platform has not performed a reference move successfully.

  – Start motion to the zero position (X = Y = Z = U = V = W = 0):
  Start the motion using the ***Move to zero position*** button.
  The ***Move to zero position button*** is deactivated as long as the parallel kinematics positioner platform has not performed a reference move successfully.

Coordinate systems

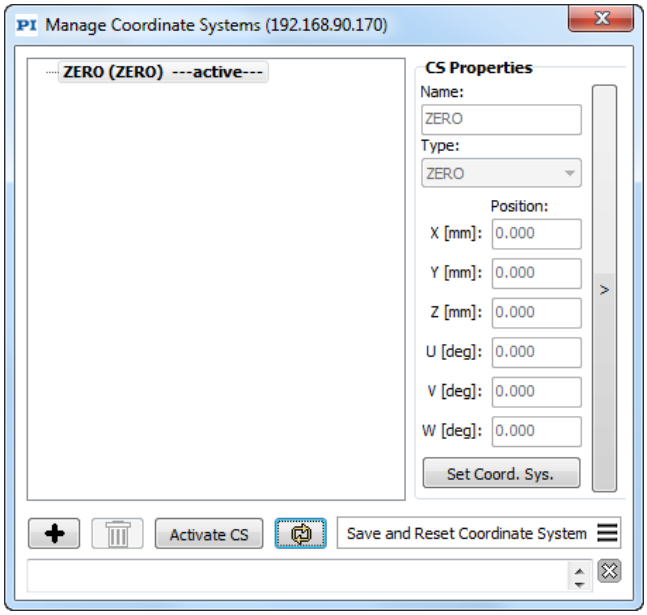
The currently active coordinate system is shown below ***Active Coordinate Systems***, its detailed axis settings below ***Coordinate System Details***. The settings for each axis can be changed by using the up/down buttons or by directly entering a value. To apply changed values use the buttons ***Move Relative Work*** or ***Move Relative Tool***.

To define a home coordinate system click the button ***Define Home Coordinate System (KSF)***. The button ***Manage Coordinate Systems*** opens a dialog (p. 154) in which you can define and manage coordinate systems.

For detailed information on coordinate systems for Hexapod microrobots and parallel-kinematic positioners see Technical Note C887T0007.

4.12.1 Manage Coordinate Systems

The ***Manage Coordinate Systems*** dialog is opened for the connected controller with the button ***Manage Coordinate Systems*** in the ***Positioner Platform*** window.



The dialog can be expanded by clicking the vertical **>** button. In expanded mode, the dialog shows an additional pane with fields for setting the lower and upper soft limits, the step size, and the pivot point for the individual axes of the coordinate system (see figure below).

In the dialog you can do the following:

Click the ***+*** button to define a new coordinate system. Set the name (default name: NewCS1) and type for the new system and specify the position values for the axes. Then click ***Set Coord. Sys.*** to save the new coordinate system to the controller's nonvolatile memory. After a refresh of the display in the dialog, you can see the new system as a new entry in the list of coordinate systems.

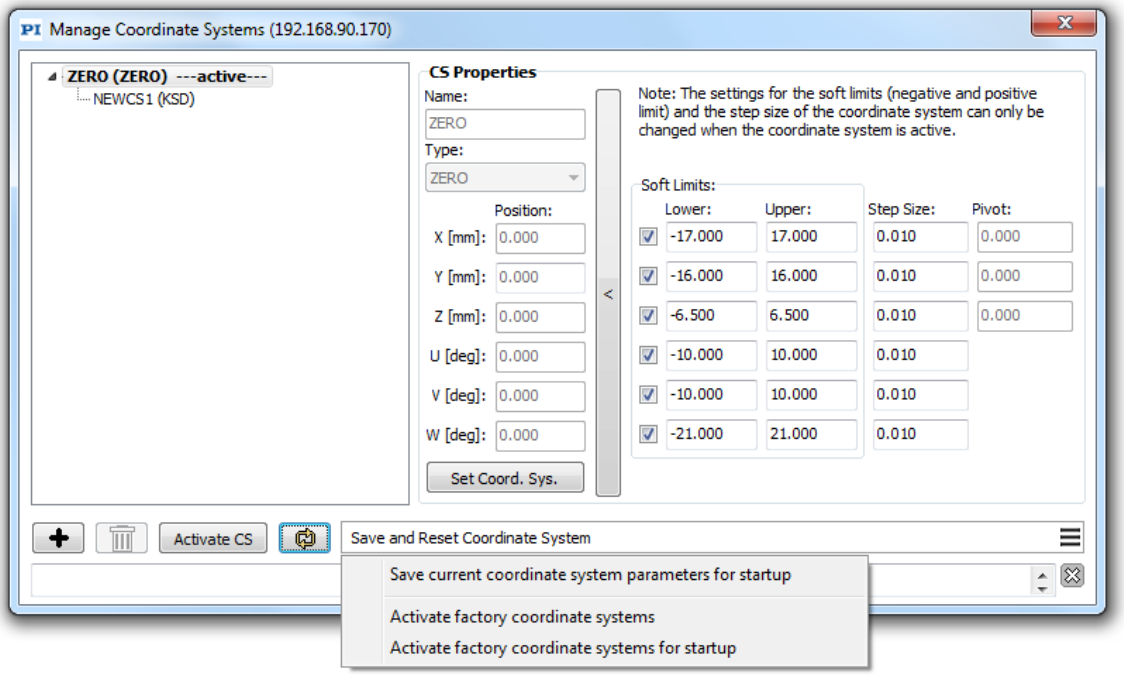
With the button you can delete the coordinate system selected in the list.

A coordinate system selected in the list can be activated for the parallel-kinematics positioner with the button ***Activate CS***.

Use the button to refresh the display in the dialog.

In the menu ***Save and Reset Coordinate System*** these functions are available:

- Save current coordinate system parameters for startup
- Activate factory coordinate systems
- Activate factory coordinate systems for startup



For detailed information on coordinate systems for Hexapod microrobots and parallel-kinematic positioners see Technical Note C887T0007.

4.13 Embedded Scan and Align Algorithms

The ***Show Embedded Scan Window*** menu item is on the controller menu of supporting controllers (e.g. Hexapod controllers or Hexapod system simulations). It gives access to the special scan and align algorithms which are implemented in the controller firmware.

With the ***Show Embedded Scan Window*** menu item you have to select the algorithm to be used. For each of the selected algorithms, a separate window opens. Multiple windows can be open at the same time.

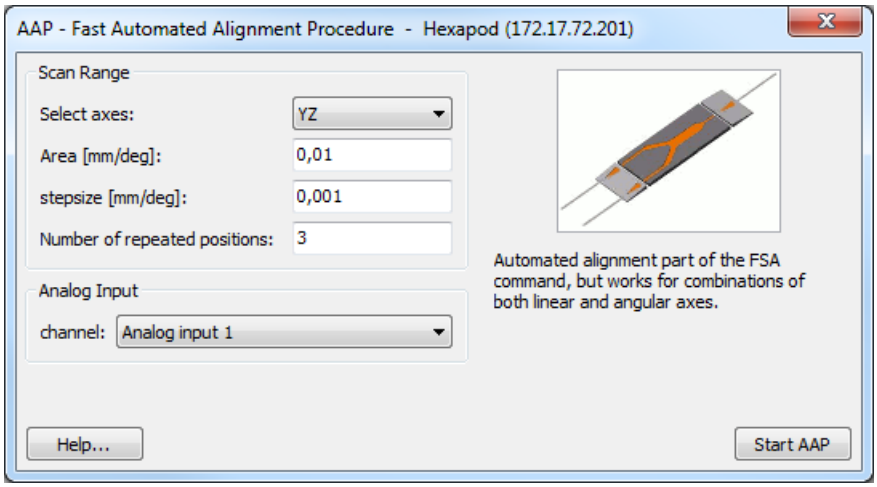
See the user manual of your Hexapod system or of your Hexapod controller for the supported algorithms and a more detailed description.

INFORMATION

There are algorithms for line scans and for area scans. Although with line scans only a distance to be scanned along an axis must be given, the corresponding field is labled ***Area*** for compatibility reasons. With area scans, the ***Area*** field in fact gives the side length of the square area to be scanned.

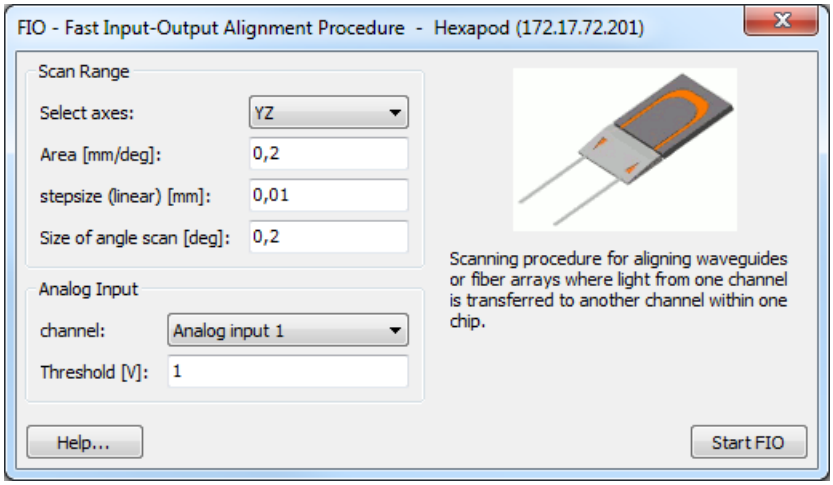
Examples for scan and alignment procedures

AAP - Fast Automated Alignment Procedure



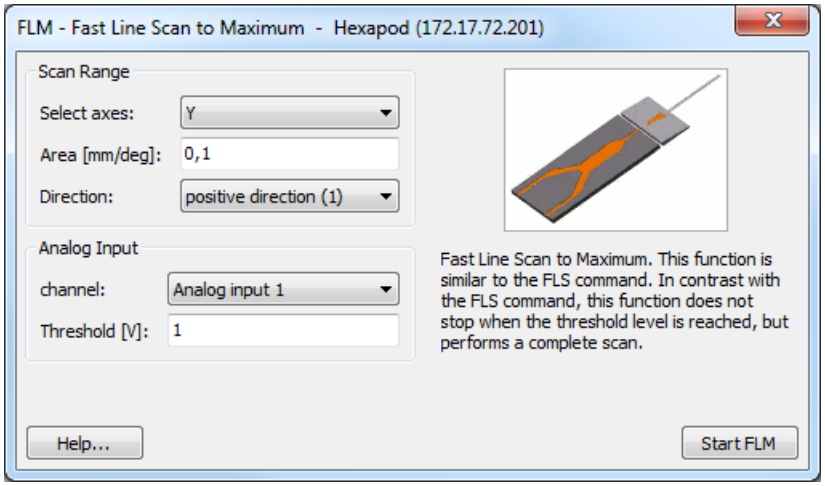
Starts a scanning procedure for a detailed determination of the maximum intensity of an analog input signal. The scanning procedure started with AAP corresponds to the "fine portion" of the scanning procedure that was started with FSA.

FIO - Fast Input-Output Alignment Procedure



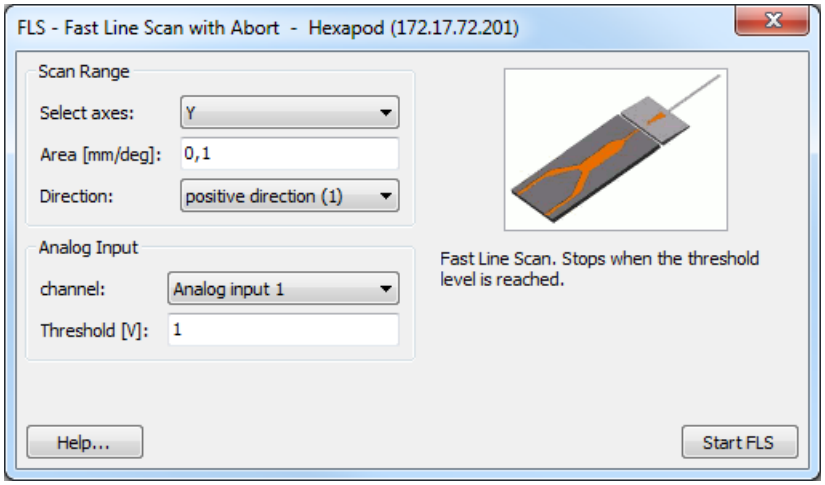
Starts a scanning procedure for the alignment of optical elements (e.g. optical fibers), the input and output of which are on the same side. Within the element, light is transmitted from the input to the output. In order for a sensor that is connected with the analog input channel of the controller to receive the signal at the output of the optical element with maximum intensity, the input and output of the optical element must be aligned at the same time.

FLM - Fast Line Scan to Maximum



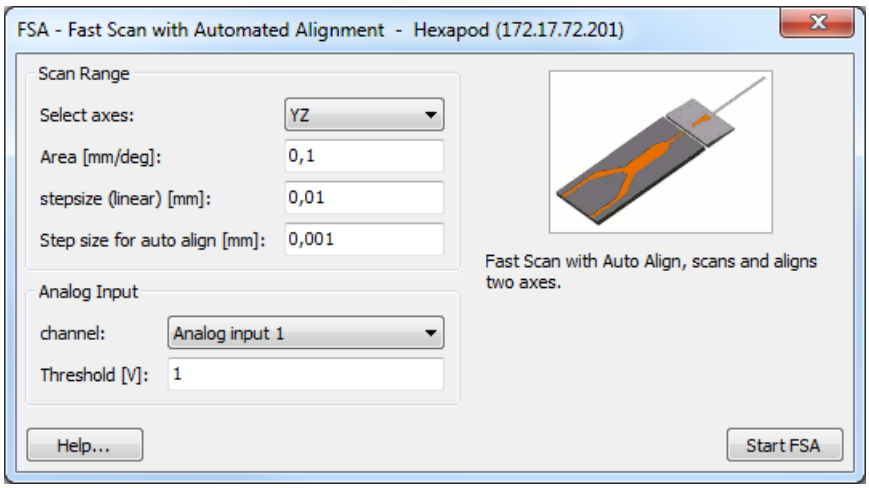
Starts a scanning procedure to determine the global maximum intensity of an analog input signal. The scanning procedure scans a specified distance along an axis for the intensity of the analog input signal. In the case of several maximum intensities, this prevents that only a local maximum is found instead of the global maximum.

FLS - Fast Line Scan with Abort



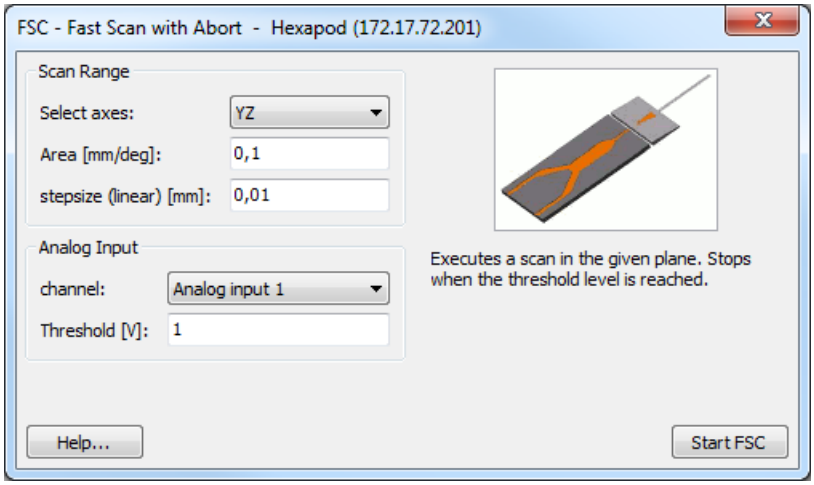
Starts a scanning procedure which scans a specified distance along an axis until the analog input signal reaches a specified intensity threshold. When the intensity threshold has already been reached at the initial position (current position at the time the command is received), a different start position will not be approached.

FSA - Fast Scan with Automated Alignment



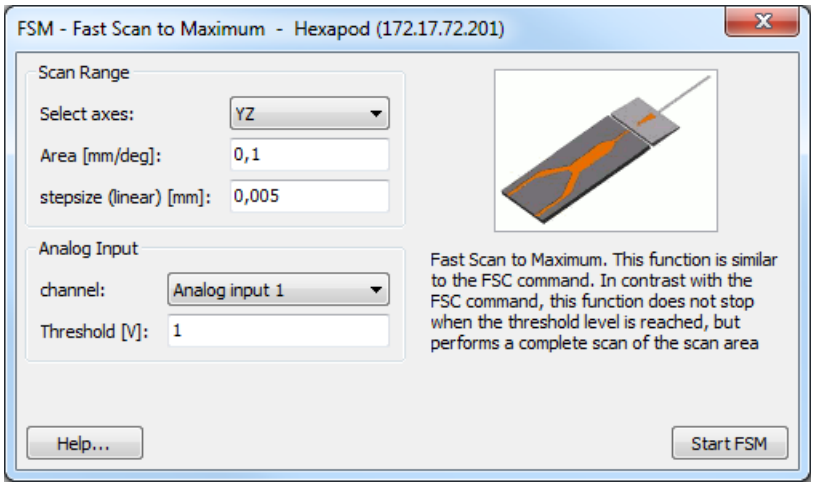
Starts a scanning procedure to determine the maximum intensity of an analog input signal in a plane. The search consists of two subprocedures: "Coarse portion" corresponds to the procedure that is started with the FSC command; "fine portion" corresponds to the procedure that is started with the AAP command. The fine portion is only executed when the coarse portion has successfully finished beforehand.

FSC - Fast Scan with Abort



Starts a scanning procedure which scans a specified area until the analog input signal reaches a specified intensity threshold. The scanning procedure started with FSC corresponds to the "coarse portion" of the scanning procedure that is started with the FSA command.

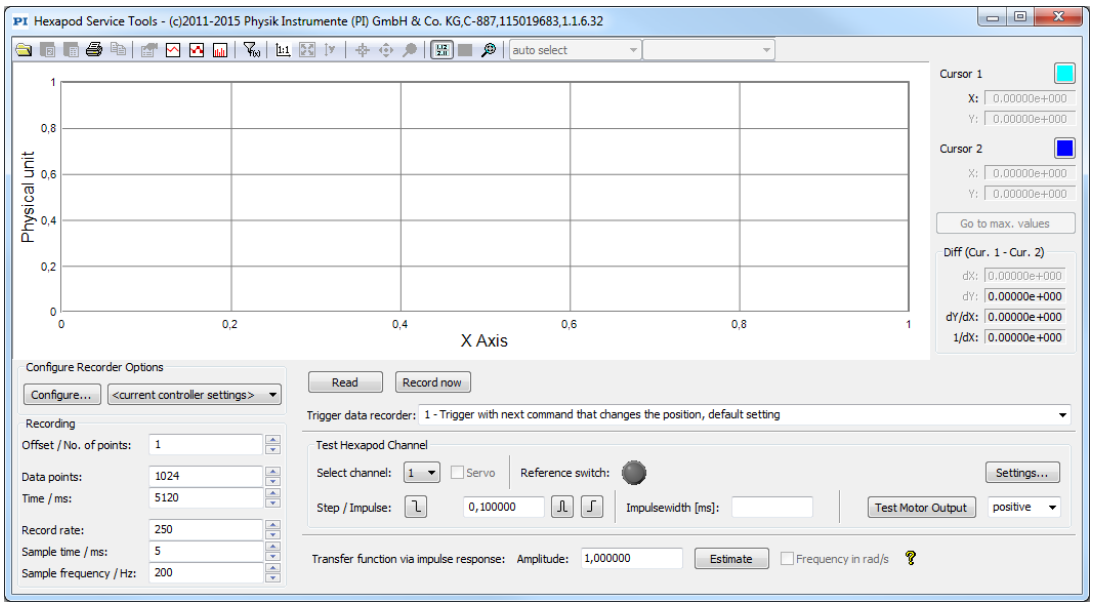
FSM - Fast Scan to Maximum



Starts a scanning procedure to determine the global maximum intensity of an analog input signal in a plane. The scanning procedure scans a specified area for the intensity of the analog input signal. In the case of several maximum intensities in the scanning area, this prevents only a local maximum from being found instead of the global maximum.

4.14 Hexapod Service Tools

The ***Show service tools…*** menu function is available via the controller menu of supporting controllers, e.g. hexapod controllers.



Use the ***Show service tools…*** function only if failure of the hexapod system occurs. It gives access to the ***Hexapod Service Tools*** window where you can perform the following tests for the single hexapod struts:

- **Impulse response measurement** of a hexapod strut. Failures may be indicated by large overshot of the actual position, by oscillation of the actual position or by large differences between actual position and commanded position.
- **Measurement of the motor output during one spindle revolution**, the measurement result is given in percent of the maximum motor output and makes it possible to evaluate the control reserve. Failures may be indicated if the motor output value continuously exceeds 80 % or if it shows an abnormal high value at a certain point of the travel range.

To configure the data display in the graphics pane and perform operations with the data use the window's toolbar buttons (p. 68).

4.14.1 Performing Strut Tests

NOTICE

Damage due to collisions during the strut test!

During the strut test the hexapod moves in an unpredictable way. There is no collision check even if a configuration for avoiding collisions was stored on the controller using the PIVeriMove software. Soft limits set with the NLM and PLM commands for the moving platform of the hexapod are ignored during a strut test.

Therefore, colllisions between the hexapod, the load to be moved and the surroundings are

possible. Collisions can damage the hexapod, the load to be moved and the surroundings.

- ➢ Make sure that no collisions between the hexapod, the load to be moved and the surroundings are possible during a strut test.
- ➢ Do not place any objects in areas where they can get caught by moving parts during a strut test.
- ➢ Observe the hexapod during a strut test to be able to take actions if a fault occurs.

NOTICE

Damage due to unwanted position changes during a strut test!

During a strut test the hexapod strut can reach a limit switch. Therefore, the servo mode for the axes of the moving platform of the hexapod is automatically switched off.

If the actual load on the hexapod exceeds the maximum holding force which is based on the self-locking of the actuators, switching off the servo mode for the axes of the moving platform of the hexapod can cause unwanted position changes of the hexapod.

Therefore, colllisions between the hexapod, the load to be moved and the surroundings are possible. Collisions can damage the hexapod, the load to be moved and the surroundings.

- ➢ Before you start a strut test, make sure that the actual load on the moving platform of the hexapod does not exceed the maximum holding force which is based on the self-locking of the actuators.

INFORMATION

Recommendations for performing strut tests:

- ➢ If possible, command the hexapod to the reference position before a strut test to have the largest possible travel range in the positive and negative direction for each hexapod strut.
- ➢ Set the velocity to a suitable value with the VLS command before a strut test.

INFORMATION

Instead of the impulse response you can also measure and evaluate a step response of the hexapod strut. But after performing the step the hexapod strut does not move back to the starting position. If multiple steps are performed, the hexapod strut can therefore reach a limit switch. When the limit switch is reached the servo mode is switched off for the moving platform of the hexapod.

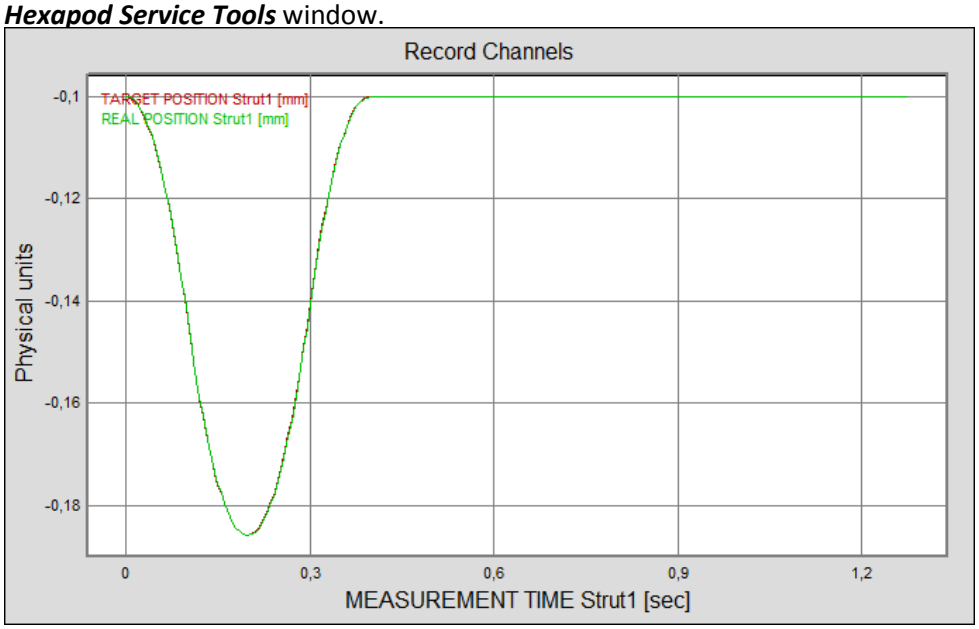
Proceed as follows for strut tests:

1. Select the ***Show service tools… item*** of the controller menu.

   A window opens with a message saying that damage is possible during the usage of the ***Hexapod Service Tools*** window.

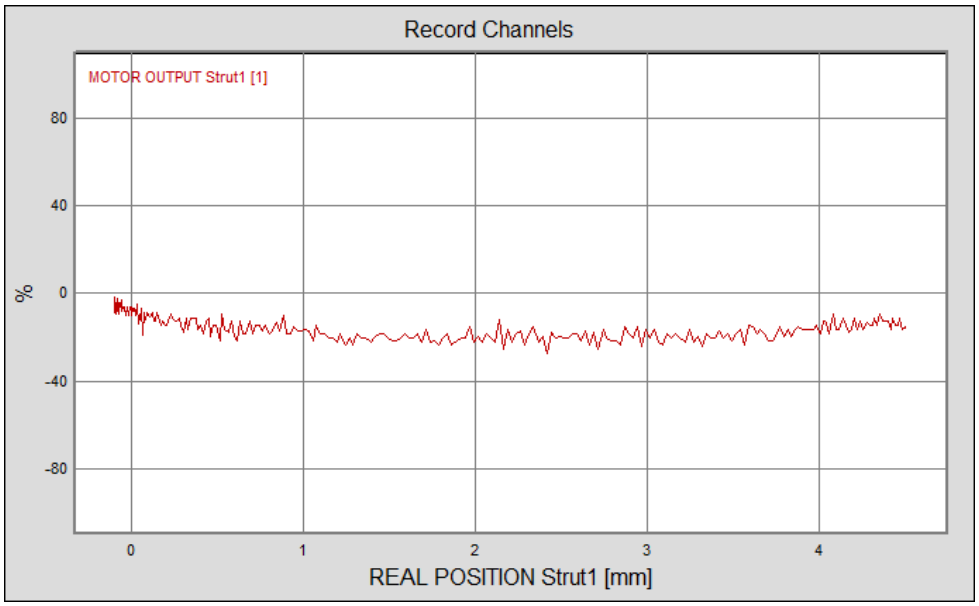
2. Click ***Show service tools*** in the message window to open the ***Hexapod Service Tools*** window.

3. In the ***Hexapod Service Tools*** window, measure the impulse response for a hexapod strut:
   a) In the ***Select channel*** field, select the strut for which the impulse response is to be measured.
   b) In the ***Impulsewidth [ms]*** field, enter the pulse width of the impulse in milliseconds. Recommended value: 100 ms.
   c) Click on the ⎍ or ⎍ button to start the impulse in positive or negative direction. Starting the impulse also starts the recording of the current position and the target position of the hexapod strut. The hexapod strut moves according to the given pulse width of the impulse and moves back to the starting position afterwards.
   d) Evaluate the impulse response using the curves in the graphics pane of the ***Hexapod Service Tools*** window.



   A step response measurement can be started using the ⎍ or ⎍ button. The step size is given by the entry in the field between the buttons for the impulse start.

4. In the ***Hexapod Service Tools*** window, measure the motor output during one spindle revolution of a hexapod strut:
   a) In the ***Select channel*** field, select the strut for which the motor output is to be measured.
   b) In the field to the right of the ***Test Motor Output*** button, select the direction (positive or negative) for the spindle revolution.
   c) Click on the ***Test Motor Output*** button to start one spindle revolution of the hexapod strut in the selected direction.
   Starting the spindle revolution also starts the recording of the current motor output and the current position of the hexapod strut. The hexapod strut does not move back to the starting position after the spindle revolution.
   d) Evaluate the recorded motor output using the curve in the graphics pane of the ***Hexapod Service Tools*** window.



If multiple spindle revolutions are performed in the same direction, the hexapod strut can reach the limit switch. When the limit switch is reached the servo mode is switched off for the moving platform of the hexapod.

5. If the test results indicate failure of the hexapod:
   a) Do not operate the hexapod system anymore.
   b) Send the test results to the PI customer service department for diagnosis (mailto:service@pi.de).

4.14.2 Configuration of Data Recording

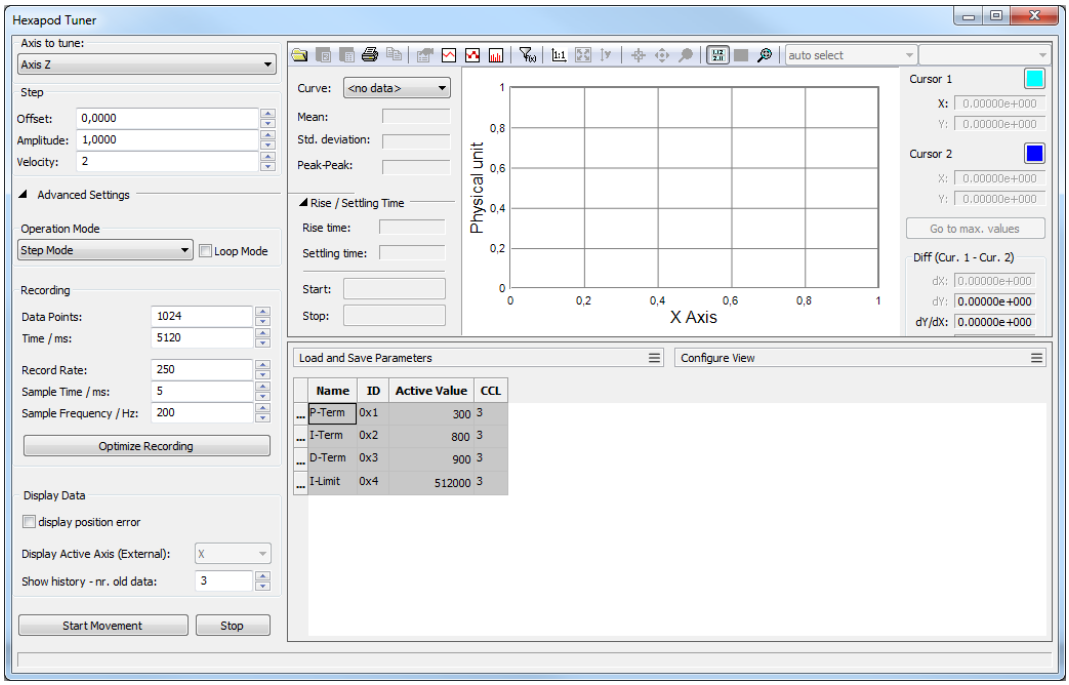
When you open the ***Hexapod Service Tools*** window, a suitable data recorder configuration is preset which cannot be changed. You can only change the following settings in a separate window which can be opened with the ***Settings…*** button:

- ***Record Rate*** field: Specify the number of servo-loop cycles before each next data point is recorded, must be an integer value ≥ 1. The larger the value, the longer the time period covered by recording. The duration of a recording can be calculated as follows:
  Rec. Duration = Servo Cycle Time * Record Rate * Number of Points
  where *Servo Cycle Time* and *Number of Points* (length of the data recorder table) depend on the controller, see controller user manual
  Note that the ***Record Rate*** is a global data recorder setting which will not only affect measurements in the ***Hexapod Service Tools*** window.
- ***# of data points*** field: You can set the number of data points to be read from the controller for the display.

4.15 Hexapod Tuner

The ***Hexapod Tuner*** is a tool for the easy optimization of a Hexapod's parameter settings. It displays all elements essential for the optimizing of a Hexapod's parameter settings in one window:

- Left pane: Settings for the recording of a motion
- Right pane - top: Graphics pane for the evaluation of the recorded data
- Right pane - bottom: Parameters which can be optimized



To optimize a Hexapod's parameter settings, a motion is performed that includes all struts of the Hexapod. The optimization is done for one Cartesian axis, i.e. one of the six degrees of freedom in which the Hexapod can move. Then the recorded data can be evaluated and the parameter values can be adjusted, if necessary. When a parameter is adjusted the new value will be set for *all* the struts (= the logical axes) of the Hexapod.

Recording a motion

In the left pane, the settings for the recording of a motion are configured.

1. Select the ***Axis to tune***.

   The axes that can be selected are the six Cartesian axes X, Y, Z (linear axes) and U, V, W (rotational axes).

   For the optimization of a Hexapod's parameter settings, we recommend to select axis Z (motion in vertical direction), in order to have all struts of the Hexapod make the same movement.

2. Configure the ***Step*** settings for a motion of the Hexapod:

a) ***Offset*** defines the point (in physical units) within the motion at which the recording is to be started.

b) ***Amplitude*** sets the amplitude to be used for the motion. The unit of the amplitude depends on the controller.

c) ***Velocity*** sets the velocity to be used for the motion (in physical units/s).

3. If necessary, open ***Advanced Settings*** by clicking on the arrow and further configure the recording of the data:

   – Select the ***Operation Mode***:

     ▫ ***Step Mode***: A motion is performed using the trajectory generator. The motion is a velocity-dependent linear step of the set amplitude.
     For this mode, wave table 100 is used, which will be overwritten when a Step Mode motion is applied for the first time. A corresponding warning message is displayed at the start of the ***Hexapod Tuner***.
     Select this mode if you want to optimize the Hexapod's settling behavior.

     ▫ ***Predef. Wave Generator***: A motion along one or more predefined waveforms is performed.
     The waveforms can be defined in the program's PI Wave Generator Tool (p. 103), for example.
     Select this mode if you want to optimize the Hexapod's motion behavior over an entire trajectory.

     ▫ ***Frequency Response***: A Step Mode motion is performed and displayed as a frequency diagram (FFT).

   – Check ***Loop Mode***, if you want to repeat the motion until stopped with the ***Stop*** button.
   During a motion in loop mode, you can change parameter values. As the changed settings take effect immediately, i.e., for the next cycle of motion execution, you can directly see the progress of the optimization.

   – Configure the ***Recording*** settings (for the description see "Data Recorder" (p. 125)).

   – By clicking the ***Optimize Recording*** button, you can optimize the number of data recorder points that will be read from the controller: The recording settings will be adjusted so that the entire motion is recorded and displayed.

   – Via the ***Display Data*** options, configure the settings for the display of the recorded data:

     ▫ Check ***display position error*** to view the position error curve alongside the curves of the commanded and real positions.

     ▫ The option ***Display Active Axis (External)*** is only enabled when the operation mode "Predef. Wave Generator" is active. Via this option you can view what happens on each of the Cartesian axes during the performed motion: Select the axis whose behavior you want to view and start the motion via the ***Start Movement*** button.

     ▫ Via the field ***Show history - nr. old data*** you can set the number of recordings to be displayed at the same time. When you change parameter values between recordings, this option lets you track performance improvement.

4. Start the motion and its recording with the ***Start Movement*** button.
   If loop mode is active, the recording must be stopped with the ***Stop*** button.

   The recorded motion data is displayed in the graphics pane.

   To configure the data display in the graphics pane and perform operations with the data use the window's toolbar buttons (p. 68).

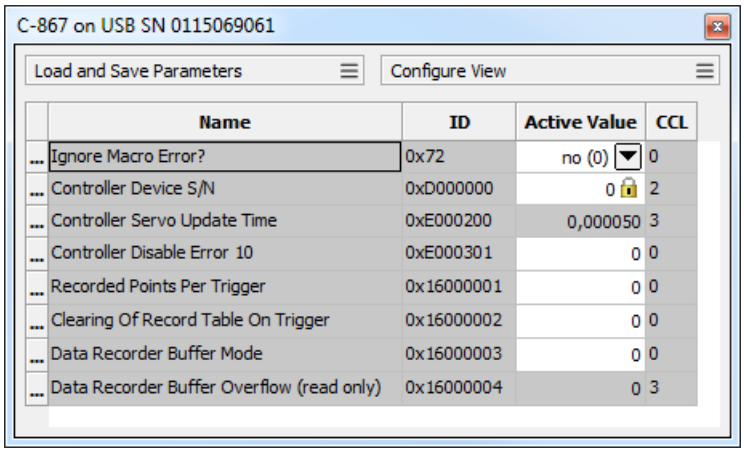
Optimizing parameter settings

The displayed data can then be evaluated, and - if necessary - parameter values can be changed. Start the recording again to evaluate the changed settings. For the description of changing parameter values see "Changing Parameter Values" (p. 99).

Please note that any changes in axis-related parameters you make in the ***Hexapod Tuner*** window are applied to *all* the struts of the Hexapod. If a further fine-tuning is necessary, the corresponding parameter values can be changed for individual struts via the `SPA` command in the program's ***Command entry*** window (p. 64) or in PITerminal.

4.16 System Parameters

In the system parameters window the values of the controller's system parameters are displayed. System parameters are the parameters of the controller that are independent of the connected mechanics.



The parameters are shown with their names (***Name***), the parameter IDs (***ID***), the current values (***Active Value***), and - if configured - their command levels (***CCL***).

- Parameters with command level (CCL) can be changed.
- Protected parameters (CCL > 0) are marked by a lock. For these, you have to change the command level before you can change the parameter's value: A dialog is opened in which you have to enter the password for changing the command level.
- Read-only parameters (CCL > 2) are displayed in gray. Their values cannot be changed.

To change a parameter value, click in the parameter's ***Active Value*** field and enter the new value.

Menu functions

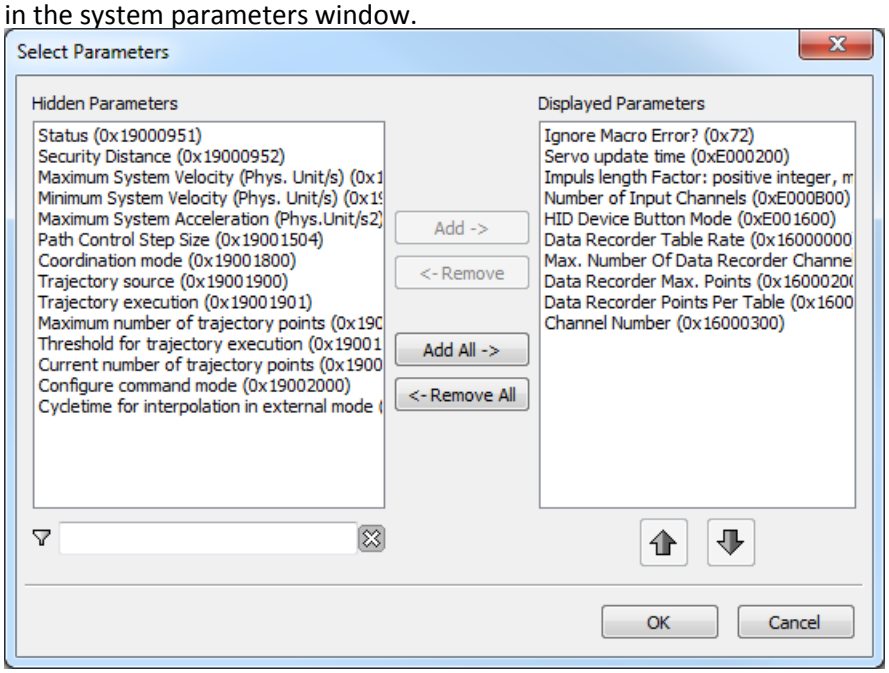
The menus above the parameter pane provide functions for working in the window. To open a menu click in the field or on the button to the right of the field.

The ***Load and Save Parameters*** menu provides the following functions:

- Refresh parameter view

The ***Configure View*** menu provides the following functions:

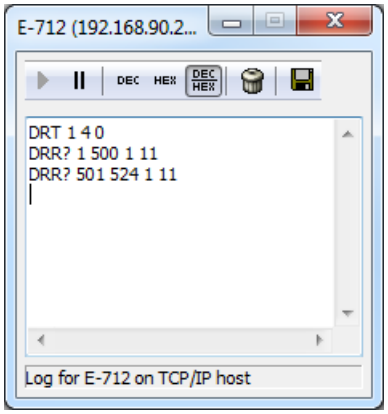
- ***Select parameters...***
  Opens the dialog ***Select Parameters*** in which you can select the parameters to display in the system parameters window.



- ***Store current view...***
  Saves the current parameter values as user-defined parameter settings. In the dialog that is opened, enter a name under which the settings are to be stored.
- ***Delete stored view...***
  Stored user-defined parameter settings can be deleted. In the dialog that is opened, select the settings to be deleted.
- ***Show command level (CCL)***
  Shows the parameters' command level.
- ***Show all***
  Shows all controller parameters.

4.17 Log Window

The ***Log window...*** function is available for every controller via its menu and opens a controller-specific log window. In this window, you can monitor the commands which are sent to the controller when you use the controls of PIMikroMove. This is a good way to see what commands are required for certain actions and to learn the command syntax.



With the ***Start*** and ***Pause*** icons, you can activate and deactivate logging.

The ***DEC***, ***HEX*** and ***DEC|HEX*** buttons modify the number format for parameter IDs used with SPA.

Use the buttons to the right to clear the window content or save it to a text file on the host PC.

INFORMATION

The following commands are not displayed in the log window:

- Status requests sent by PIMikroMove
- Responses from the controller
- Commands sent from the ***Command entry*** window of PIMikroMove

5 Start up Controller - Details

This section explains how to connect a controller to PIMikroMove and how to start up the connected stages. The ***Start up Controller*** dialog is shown at start of PIMikroMove, and can later be opened by a number of different menu sequences.

PIMikroMove guides you through the startup procedure which typically comprises multiple steps. It depends on the controller which of the following steps have to be performed:

- All controllers:
  Connect controller (p. 170)
- Controller which support loading settings from a stage database:
  Select connected stages (p. 172)
- E-518 interface module:
  Start up E-518 (p. 174)
- System whose axes have to be initialized (e.g. referenced) before normal use:
  Start up axes (p. 175)

If the hardware configuration remains unchanged (controller, communication interface, connected stages), you can automate the startup procedure using the AutoConnect functionality (p. 178). AutoConnect means that you define the startup steps once—thereafter you will no longer have to deal with them because the program performs them automatically.

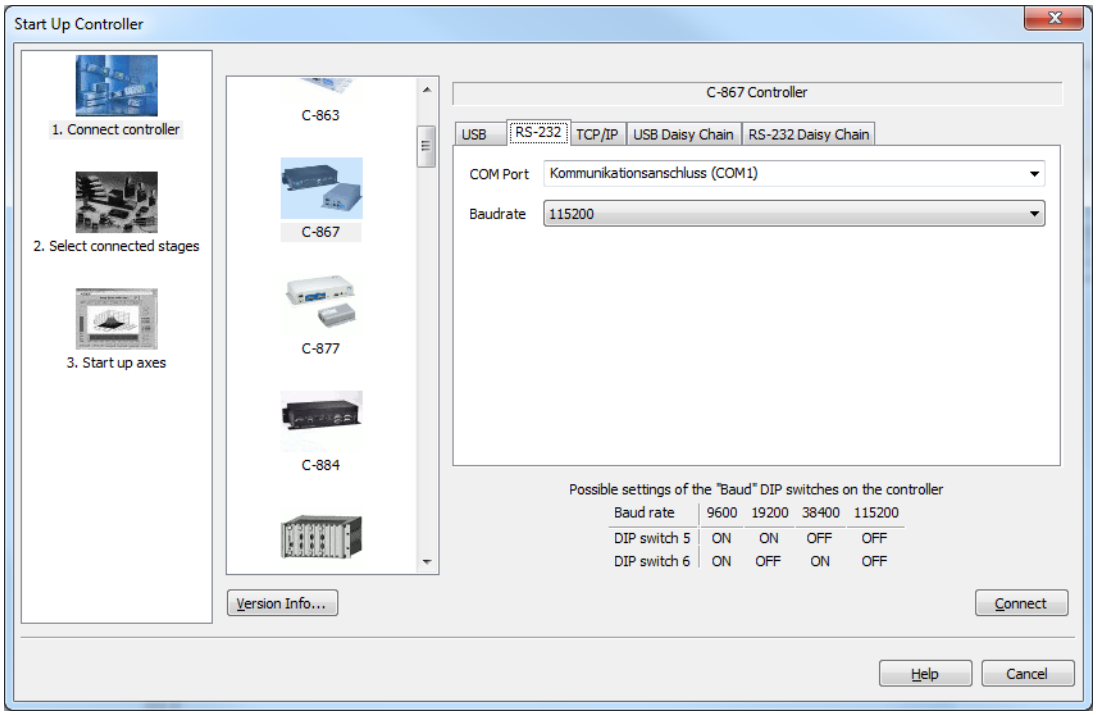
If you start PIMikroMove from a command line or via batch processing, you can set some options for the program start, e.g. to use a customized configuration file instead of the default file. See "Command Line Options" (p. 182) for more information.

In this Chapter

5.1 Connect Controller

You can show the ***Connect Controller*** dialog at any time with ***New connection*** on the toolbar or with the ***Connections > New…*** menu sequence. This way, you can connect additional controllers to PIMikroMove. It is not necessary to close an existing connection before additional controllers can be connected since PIMikroMove is designed to handle multiple connections. For example, the axes of all connected controllers will be listed in the main window.

5.1.1 General



The list box in the middle of the window shows symbols for all controllers known to the host system. At first, select the controller you want to use from that list. Then, select the tab card for the interface you want to use in the right pane of the window. On that tab card, choose the correct interface settings (baud rate, COM port, IP address, …). When all settings are done, click ***Connect*** to establish the connection. If the connection is established successfully, the next step of the ***Start up controller*** window will be shown automatically.

Please note that not all of the interfaces shown may be physically present on your system.

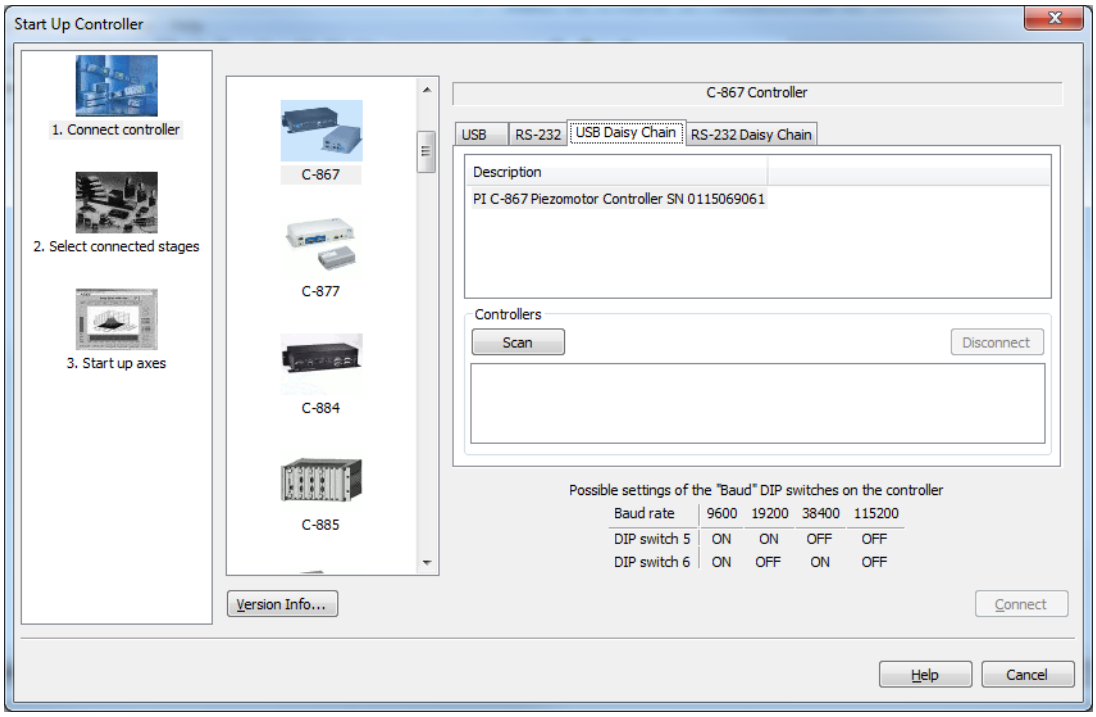
INFORMATION

If the controller is connected via a USB port, this USB interface might appear as an additional COM port in the port-selection list.

With ***Version info*** you can show the versions of the software components found by PIMikroMove. This information can be helpful in diagnosing problems.

5.1.2 Daisy Chain Connection

Some controllers can be connected to PIMikroMove via a daisy chain of multiple devices. See the controller's user manual for hardware connection details.



If you want to connect one or more devices which are part of a daisy chain, use one of the ***… Daisy Chain*** tab cards (availability depends on controller):

- ***USB Daisy Chain***:
  Click the ***Scan*** button to find all controllers in the daisy chain. Select one controller from the list and click the ***Connect*** button.
- ***RS-232 Daisy Chain***:
  Set the COM port to use on the host PC and the baud rate. Then click the ***Scan*** button to find all controllers in the daisy chain. Select one controller from the list and click the ***Connect*** button.

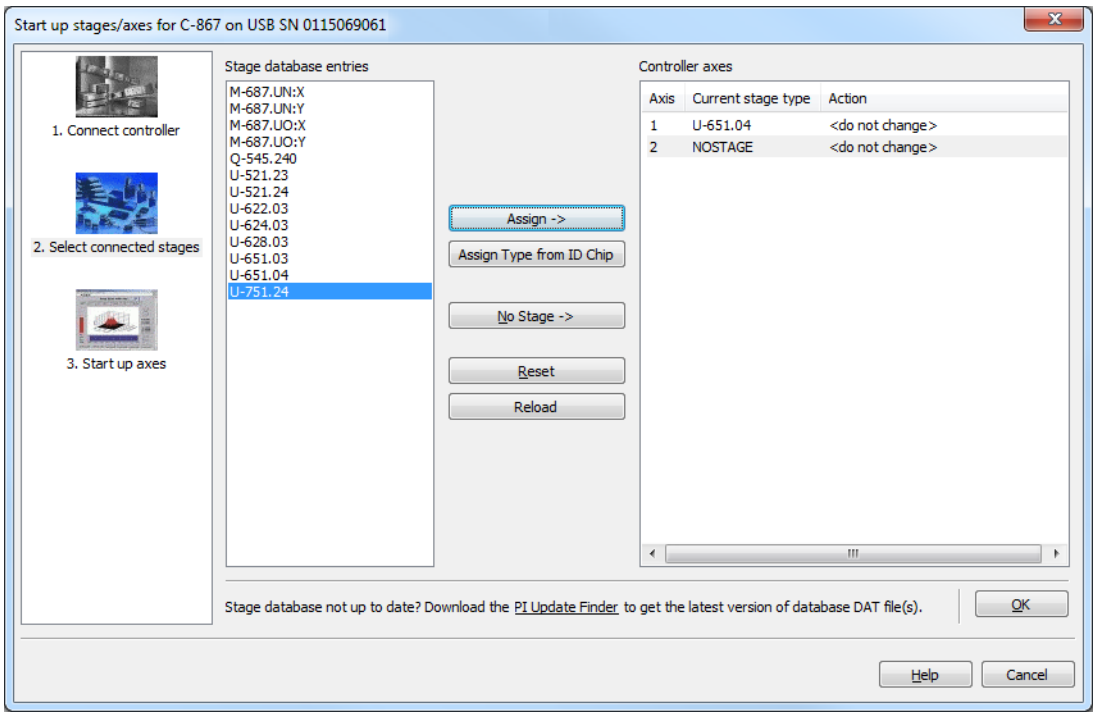
To connect other controllers on the daisy chain, use the main window ***Connections > New*** menu item to reopen the Start up controller window and, on the corresponding ***Daisy Chain*** tab card, select the desired controller and click ***Connect***.

To disconnect one of the controllers listed on a ***… Daisy Chain*** tab card, select it in the list and click ***Disconnect***. If the ***Disconnect*** button is dimmed, use the items of the ***Connections*** menu in the PIMikroMove main window to close the connection.

If you want to connect a controller with daisy chain feature as stand-alone device, do not use the ***… Daisy Chain*** tab card(s).

5.2 Select Connected Stages

You can manually open the ***Select connected stages*** window at any time with the ***Select stages*** item on the controller menu of the corresponding controller.



In this window you can select the types of the connected stages for the various axes and hence load the proper settings from a stage database, provided that the controller supports this feature. Two lists are provided:

- ***Stage database entries***: List of all stage database entries suitable for the controller
- ***Controller axes***: List of all axes of the controller
  The list shows axis identifiers, current valid stage types and the actions which will be performed when pressing ***OK***.

If you cannot find a PI stage in the list of stage database entries, download the latest version of the PI stage database from the PI Website. This can be done via the PI Update Finder, which can be started by clicking the link "Start update finder".

The following actions can be chosen for the controller axes marked in the list (actions will not be performed until you click ***OK***):

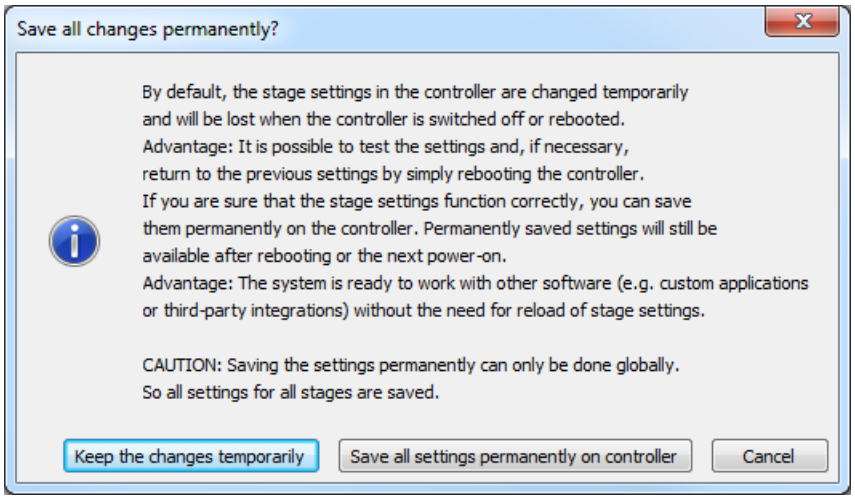
- Assign the entry selected in the stage database list to the selected axes using the ***Assign*** button. Alternatively, you can also double-click the stage database entry. When pressing ***OK***, the stage-specific settings will be loaded from the stage database to the controller.
  Action column of the ***Controller axes*** list shows "Set to: Stagename"
- Deactivate axes using the ***No Stage*** button (if supported by the controller): You can deactivate an axis if required by your application, e.g. if no stage is connected to the corresponding socket, or if motion of the axis is strictly forbidden. Deactivation of an axis means that this axis is not available for axis-related commands any more (e.g.

motion commands, position queries). Hence after pressing ***OK*** the axis will not be visible any more on the ***Axes*** tab card or in ***Single-Axis*** windows of PIMikroMove. You can undo axis deactivation at any time in the Select connected stages window by selecting a stage database entry and using the ***Assign*** button (see above).
Action column of the ***Controller axes*** list shows "Set to: NOSTAGE"

- Load the parameters of the current valid stage type again from the stage database using the Reload button. This is necessary if the settings of the current valid stage type have changed in the stage database. If you do not reload them, they will not come into ffect. When pressing ***OK***, the modified parameter values will then be loaded to the controller.
Action column of the Controller axes list shows "Reload: Stagename"
- Discard all changes using the ***Reset*** button. This will undo all action selections made before using the ***Assign***, ***No Stage*** or ***Reload*** buttons.
Action column of the ***Controller axes*** list shows "<do not change>"

OK performs the actions shown in the ***Action*** column. If <do not change> is shown in the ***Action*** column of the ***Controller axes*** list, pressing ***OK*** will not change any settings for the corresponding axis on the controller.

OK opens the ***Save all changes permanently?*** dialog, in which you have to specify how you want to load the parameter settings into the controller:



- Loading temporarily: Click ***Keep the changes temporarily*** to load the parameter settings into the volatile memory of the controller. The settings are lost when the controller is switched off or rebooted.
- Load as default values: Click ***Save all settings permanently on controller*** to load the parameter settings into the nonvolatile memory of the controller. The settings are available immediately after switch-on or reboot of the controller and do not need to be reloaded.

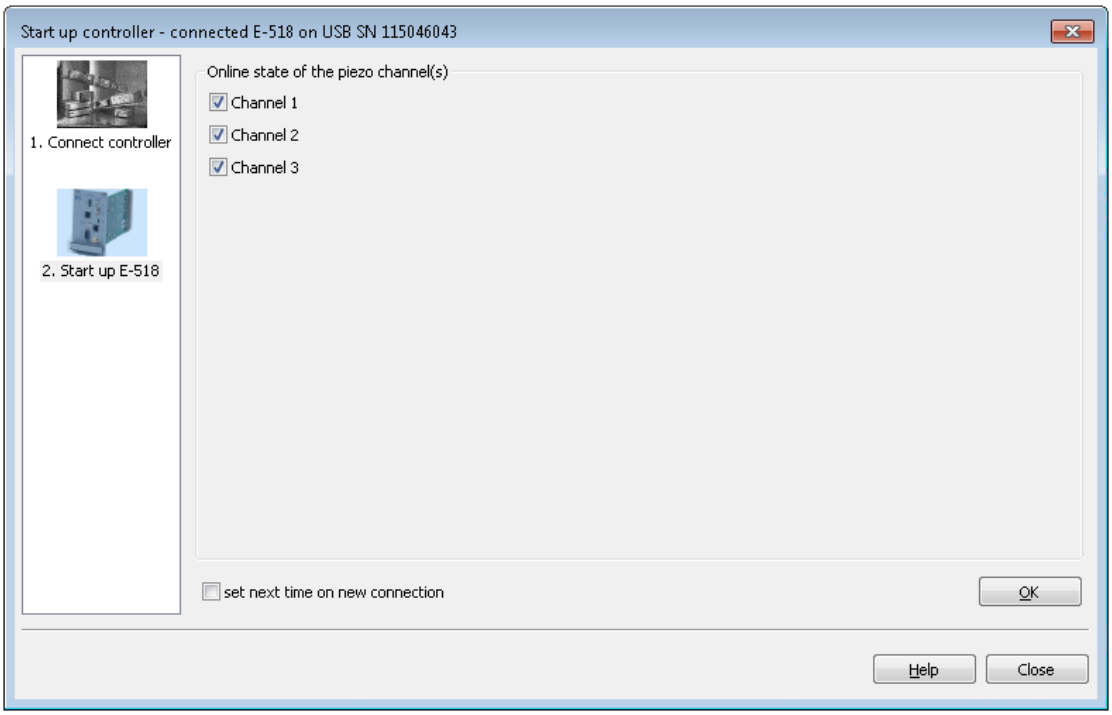
The ***Start up controller*** window changes to the ***Start up axes*** step.

INFORMATION

Controllers which are able to store parameter values in non-volatile memory may come with preset parameter values, especially when delivered with custom stages. The ***Current stage type*** column of the ***Controller axes*** list then already shows the name of the custom stage. In

this case, do not choose any stage database entry from the list. Before you click ***OK*** in the ***Select connected stages*** window, make sure that the ***Action*** column of the ***Controller axes*** list shows "<do not change>".

5.3 Start Up E-518



This dialog is only available if the connected controller is an E-518 interface module. The online state of a (piezo) channel determines the applicable control sources for the output voltage and hence for the axis motion:

- Checkbox selected = ONLINE mode:
  PIMikroMove controls the piezo channel and hence the motion of the corresponding axis. Any analog control input voltage and DC offset settings on the analog piezo control electronics will be ignored.
- Checkbox not selected = OFFLINE mode:
  The channel and hence the motion of the corresponding axis cannot be controlled by PIMikroMove but only by analog control input voltage and DC offset settings applied to the analog piezo control electronics.

INFORMATION

In ONLINE mode the SERVO switches of all channels must be set to OFF on the analog piezo control electronics.

The ***Start up E-518*** dialog will not appear again, and the current settings will be adopted if you select ***set next time on new connection***. You can reopen the dialog from the E-518 controller menu with the ***Start up axes…*** item.

5.4 Start Up Axes

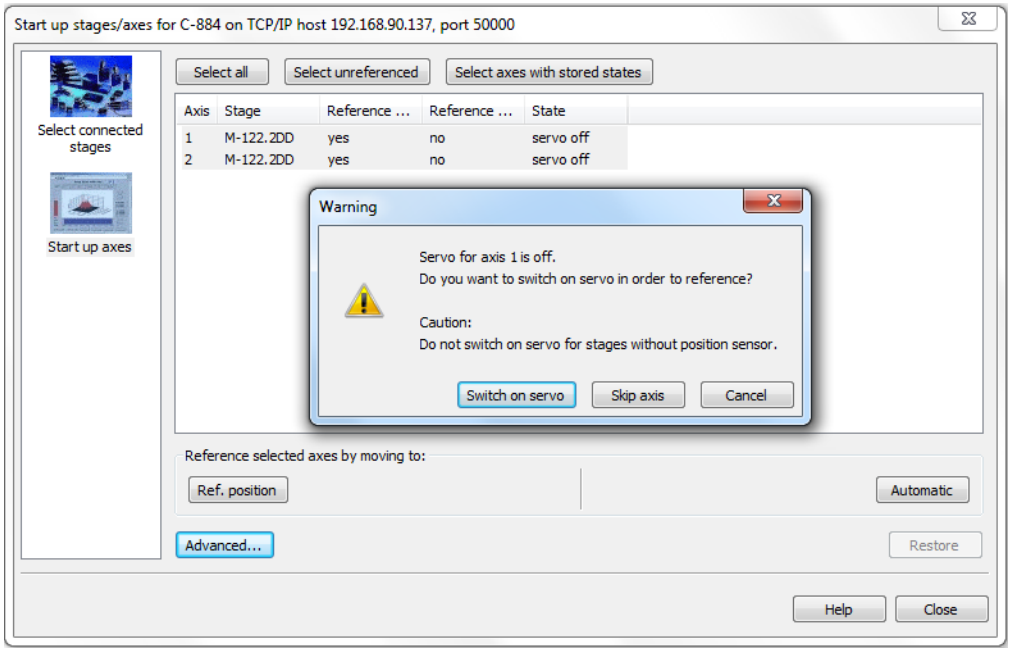
This dialog is available if a system requires that its axes be initialized before normal use. You can return to this window by selecting ***Start up axes...*** in the controller menu of the corresponding controller or in the ***Axis*** menu opened from the ***Axes*** tab card.

5.4.1 Referencing

For systems with incremental position sensors, the initialization includes determining the absolute position of each axis (referencing). Referencing is usually done by driving the motor slowly until a reference or limit switch with a known position is reached.

Clicking on one of the buttons ***Ref. position*** or ***Neg. limit***, ***Pos. limit*** (available for some controllers) starts the reference move of the selected axis or axes to the corresponding switch (reference switch or negative, positive limit switch). Use ***Automatic*** to have the reference move performed in automatic mode: The axes are moved to the reference switch, or - if no reference switch exists - to one of the limit switches.

If a warning message appears indicating that the servo mode is switched off, switch on the servo mode with the ***Switch on servo*** button.



The button ***Restore*** is enabled if the selected axis is unreferenced and positions were saved on the host PC the last time the connection to the controller was closed.

NOTICE

Damage due to crashes!

With ***Restore***, the saved position will be set as the current absolute position and will be the basis of all future calculations and range checks. If the stage has moved and the value is no longer correct, you could easily crash the stage.

➢ Be sure that the affected axes have not moved in any way since the positions were saved.

For more information see "Why do I need to reference an axis?" (p. 185).

If you need more control or you cannot move the stage to one of the known positions, call the ***Advanced startup*** (p. 176) dialog by clicking ***Advanced…***.

5.4.2 AutoZero

For systems with linear piezo actuators, both the range of sensor position values and the range of the output drive voltages are limited. If mechanical drift of the piezo actuator causes too great a shift in the relation between these ranges, then the usable closed-loop travel range will be reduced. Such an offset can be compensated by the AutoZero function.

Click ***Auto Zero*** to start the AutoZero procedure.

INFORMATION

During the AutoZero procedure, the axis will move, and the motion can cover the whole travel range.

When the piezo stage is first installed in the application, AutoZero must be run. Afterwards, AutoZero should only be executed in the following cases:

- The system is subjected to temperature changes.
- The load applied to the axis has changed (note that the effect of load changes depends on the stiffness of the stage axis).
- The mounting or environment of the stage has changed (e.g. its orientation).

Especially if absolute moves are needed, AutoZero should **not** be executed during normal operation because AutoZero changes the mechanical zero position of the axis.

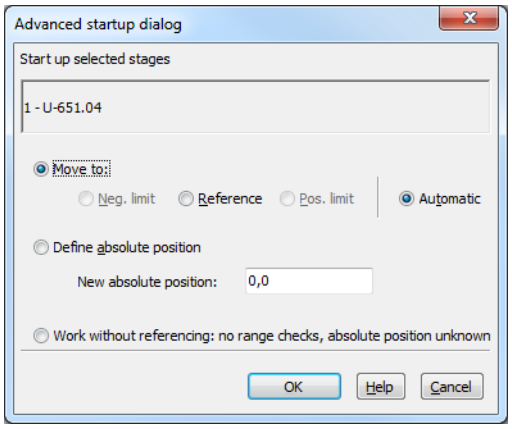
INFORMATION

AutoZero is not effective on non-linear axes.

See the user manual of the piezo controller for more information.

5.4.3 Advanced Startup

Systems with incremental position sensors must be referenced. For such systems, the ***Advanced startup dialog*** can be opened:



This dialog gives you more control over startup of the positioner axes. You have the following options:

- ***Move to***: Select how to perform the reference move. It can be performed via the reference switch (***Reference***), or via the negative or positive limit switches (***Neg. limit***, ***Pos. limit***; not available for all controllers). Use ***Automatic*** to have the reference move performed in automatic mode: The axes are moved to the reference switch, or - if no reference switch exists - to the negative limit switch.
- ***Define absolute position***: Inform PIMikroMove (and thus the motion controller) manually where the axes currently are. This makes absolute positioning possible and the status will be set to on target. Enter the position in the field ***New absolute position***. Note that the numeric value entered is interpreted as (scaled) physical units.
- ***Work without referencing: no range checks, absolute position unknown***: Abandon absolute positioning. When this is done, all range checks are switched off

NOTICE

Damage due to crashes!

The ranges for axis motion (e.g. minimum and maximum commandable position of an axis or soft limits) are not adjusted when you define the absolute position manually. This may result in target positions which are allowed by the controller but cannot be approached by the positioner. In this case, the positioner can hit the hard stop or objects in the environment and be damaged.

➢ Make sure to specify the correct value when defining the absolute position manually.

NOTICE

Damage due to crashes!

If you choose to enable relative moves only, no more range checks are performed. You could easily crash the stage.

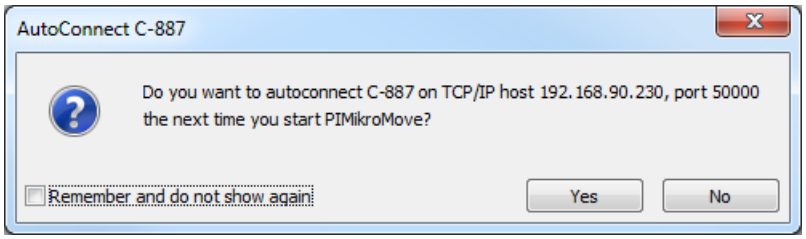
➢ Carefully command the distances for relative moves.

For more information see "Why do I need to reference an axis?" (p. 185).

5.5 AutoConnect

If the hardware configuration (controller, communication interface, connected stages) remains unchanged, you can use the AutoConnect feature. AutoConnect makes it possible to let all the startup steps described in "Connect Controller" (p. 170), "Select Connected Stages" (p. 172) and "Start Up Axes" (p. 175) run automatically in a predefined way at the start of PIMikroMove.

The ***AutoConnect Control Wizard*** assists you when defining the AutoConnect configuration. The wizard is started when you close the connection to a controller and click ***Yes*** in the ***AutoConnect*** selection dialog which opens automatically:

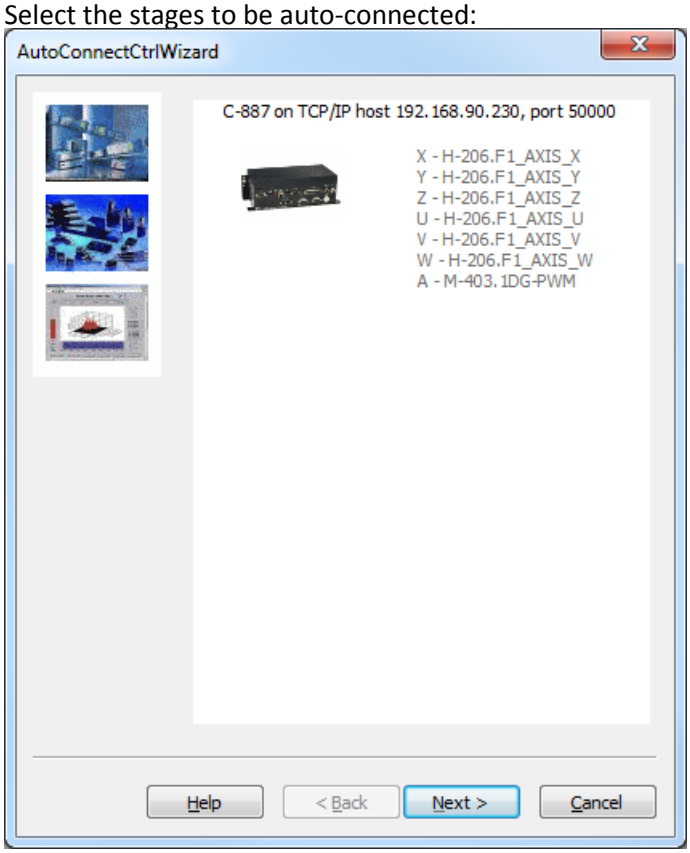


If you close all connections and click ***Yes*** in the ***AutoConnect*** selection dialog, separate dialogs and wizards will appear for the individual controllers. Depending on the controller, AutoConnect configuration can be done for each axis separately or for the controller as a whole.

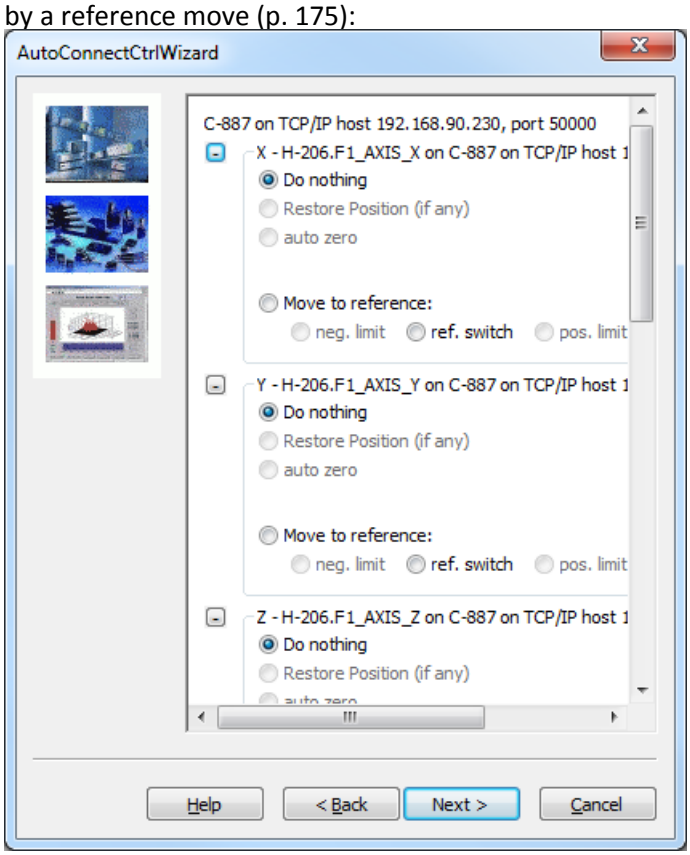
The ***AutoConnect Control Wizard*** guides you through the setup of the AutoConnect functionality. The setup may comprise the following steps:

1. Optional: Select the axes to be auto-connected (if axis selection is supported by the controller):

2. Select the stages to be auto-connected:

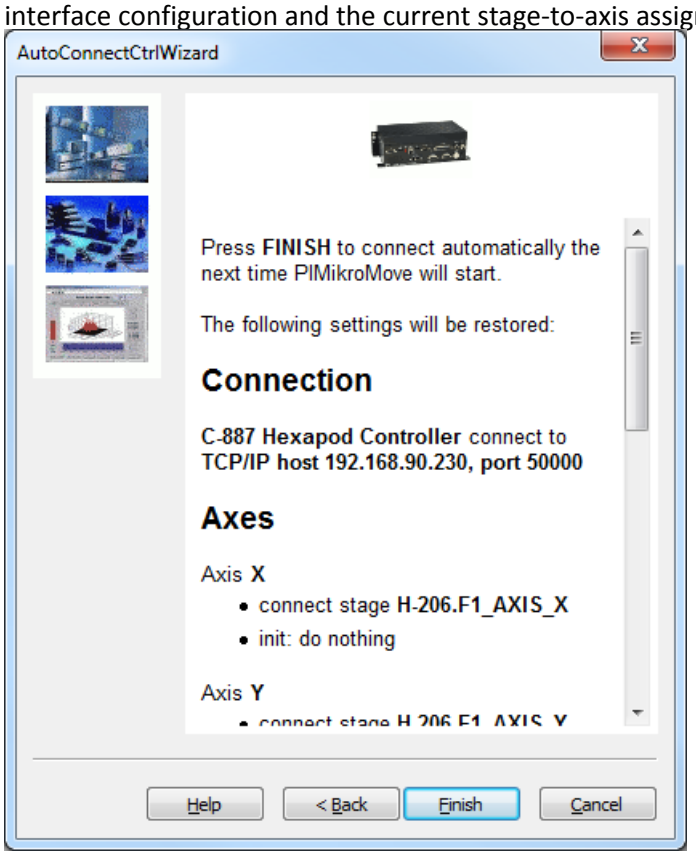


3. Define how to perform the axis initialization, e.g. by an AutoZero procedure (p. 176) or by a reference move (p. 175):



Options not supported by the controller are disabled and cannot be selected.

4. Save the AutoConnect settings by clicking ***Finish***. Information saved includes the current interface configuration and the current stage-to-axis assignment.

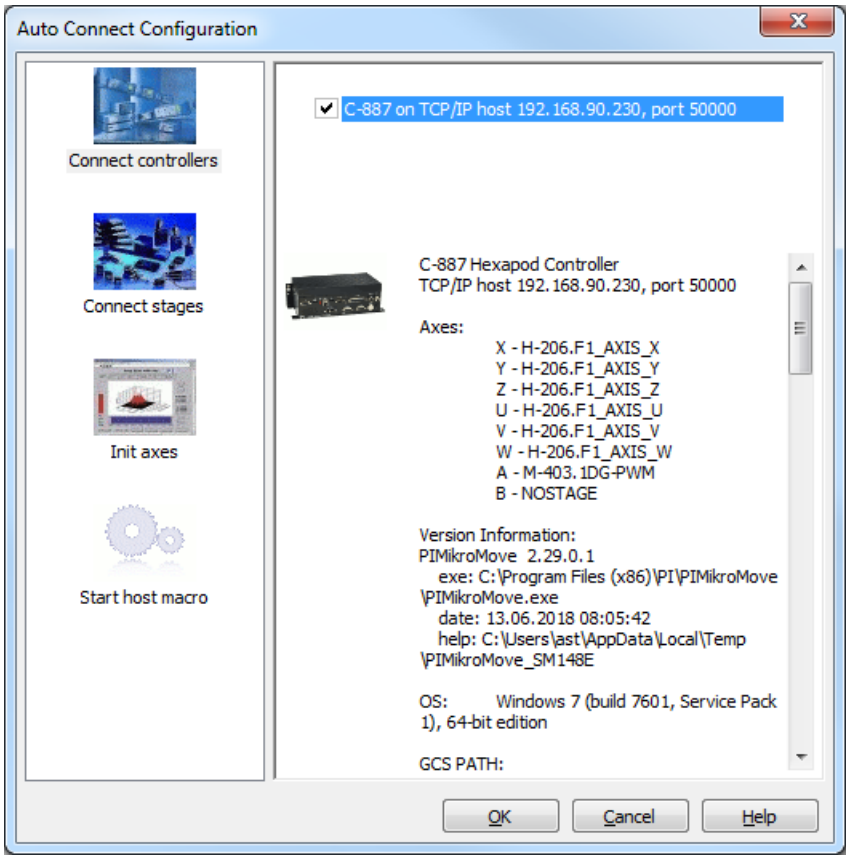


Changing the AutoConnect Configuration

The current AutoConnect settings can be changed in the ***Auto Connect Configuration*** window, which is accessible via the ***Connections > AutoConnect...*** menu function. Click on one of the icons in the leftmost pane of the window to select the settings you want to change.

Selecting ***Connect controllers*** not only deactivates/activates AutoConnect for the individual controllers, but also shows information about the hardware and software versions.

Via ***Start host macro*** you can specify if the controller is to launch a host macro. This macro will be executed when AutoConnect has finished successfully for all controllers affected by the macro (see "Host Macros" (p. 38) for details).



For controllers which are currently not connected, you can only deactivate settings in the ***Auto Connect Configuration*** window. To save changes, click ***OK***.

The AutoConnect procedure at the start of PIMikroMove can be skipped as follows:

- If you start PIMikroMove from the command line, you can use the "-na" or "--noautoconnect" switch (see "Command Line Options" (p. 182))
- The ***Wait Dialog*** at program start provides a ***Stop*** button with which you can abort AutoConnect.

5.6 Command Line Options

If you start PIMikroMove from a command line or via batch processing, you can choose the following options:

- -h or --help
  Open a window with a summary of the available command line options.
- --verbose
  Log detailed messages to the program's log file.
- -v or --version
  Show version information about PIMikroMove and the installed controller software.

- -na or --noautoconnect
  Start PIMikroMove without the AutoConnect (p. 178) functionality.
- --nowaitautoconnect
  Suppress the count-down window for the AutoConnect functionality. AutoConnect is then executed immediately upon the start of PIMikroMove.
- -c=<configuration file> or --config=<configuration file>
  Start PIMikroMove using a customized configuration file instead of the standard file. Specify the configuration file (.ini) either fully-qualified or relative to the directory of the program's executable.
- -m=<macro> or --macro=<macro>
  Start a host macro after PIMikroMove has been started. This works only if AutoConnect has been performed successfully, and if all controllers affected by this host macro are connected.
- -d=<milliseconds> or --delay=<milliseconds>
  Set the interval for controller status updates (minimum: 10).
- --usergeometryfile
  Always ask for geometry file for the visualization of Hexapod motion in the tab card "Positioner 3D View" (p. 31). The default behavior is, that the geometry data of the controller is used.
- --commandslog
  Support GCS commands log. This function is reserved for developers.
- --TCPIPDaisyChain
  Always display the "TCP/IP Daisy Chain" tab in the "Start Up Controller" (p. 169) window, even for controllers which do not support this option. This option is required for the access of controller modules with older versions of the C-885 PIMotionMaster.
- --HexapodBaseType=<Hexapod type>
  Adjust the Hexapod 3D view for the specified Hexapod type (for Hexapods with non-standard geometry).

6 Tutorials - Frequently Asked Questions

This chapter contains some general information useful in understanding the background of PIMikroMove and answers some frequently asked questions. It also has a collection of short recipes for accomplishing common tasks.

In this Chapter

6.1 Why do I need to reference an axis?

To move a stage to an absolute position the motion controller needs to know the current position. Many stages, however, have only relative position sensors (encoders). The controller must somehow determine where the stage is. This can be done by moving to a known fixed position. PIMikroMove uses a limit or reference switch to define such a known point (the reference position is the point where the reference signal changes its state).

If a stage has no limit switch or reference signal, or the stage cannot move to any of these positions without damaging the application, you can either enter the current absolute position directly, or—if the current application does not need it—you can operate using relative moves only, i.e. without absolute positions.

To reference an axis by moving to a known point, to enter the current absolute position or to start the axis in relative move mode use the Start up axes (p. 175) dialog.

Top (p. 185)

6.2 Why is everything disabled when Command entry is locked?

When the ***Command entry*** window is in the locked state, all other controls containing data regarding the selected controller are disabled (***Single-axis*** windows, ***Position pad***, …).

This is done to avoid confusion between the commands entered directly and those sent in the background by PIMikroMove to update its displays (PIMikroMove must continuously check the state of the axes and the controller).

For example, if you want to find out how a specific command works and the command you entered contains a typo or invalid data, then the controller does nothing. To find out what went

wrong, you can check the error state with ERR?. If background activity had been taking place, however, the error state may have been cleared in the meantime.

The locked state of the ***Command entry*** window is designed to give you complete control over what is sent to and received from the controller.

Top (p. 185)

6.3 How do I connect another controller to PIMikroMove?

To connect another controller or controller network to PIMikroMove you do not need to close any existing connections. You can connect as many controllers in software as you have physically connected to your PC. Simply call the Start up controller (p. 169) window with the ***Connections > New...*** menu sequence or the ***New connection*** button on the toolbar.

Top (p. 185)

6.4 How can I create a new stage type in the PI stages database?

The easiest way to add a new stage type to the user-stages dat file is to modify the parameters of an existing stage type and save them under a new name. Thereafter you can select this newly defined stage in PIMikroMove or in other PI software as well.

Proceed as follows:

1. Assign the stage type that comes closest to your stage to the appropriate axis. See "Select Connected Stages" (p. 172) for how to do this. Afterwards the ***Start Up Axes*** dialog may open—you can close this dialog because at this point it is not necessary to reference the axis.
2. Open the ***Single-Axis*** window for the axis (“Single-Axis Window" (p. 97)). To do this, either use the ***Single Axis*** Window item in the ***View*** menu or call the corresponding Axis menu from the Axes tab card.
3. Expand the ***Single-Axis*** window via the rightmost > button.
4. In the rightmost pane of the expanded ***Single-Axis*** window, display the columns for the parameters you want to modify. To do this, click the ***Select parameters…*** button. See "Select Columns to be Displayed" (p. 26) for further details.
5. Type new values in the parameter fields. As long as a value is shown in blue, it is only present in PIMikroMove but not yet sent to the controller. Press Enter on your keyboard to send the value to the controller’s volatile memory.
6. Right-click in the center pane of the expanded ***Single-Axis*** window and select ***Save parameters as User Stage type...*** from the menu that appears.
7. To save the modified settings as a new stage, enter a new name for your stage in the as stage type field and click ***OK***.
8. The new stage is now displayed on the Axes tab card, and you can work with it (e.g. reference the stage: right-click on the axis row and select the ***Start up axes...*** item, see "Start Up Axes" (p. 175)). If you want to further modify the stage parameters, use the ***Save parameters as User Stage type*** menu function again to save the changes.

Top (p. 185)

6.5 How can I modify default settings of a stage type in the PI stages database?

You can modify the default parameters of a stage type loaded from the PIStages2.dat database. Afterwards, add the modified parameters as a new stage type to the user-stages dat file.

For modifying the parameters of user stages in the PIStages3.db database see the PIStages3Editor manual.

For stage types loaded from the PIStages2.dat database proceed as follows:

1. Assign the stage type you want to modify to the appropriate axis. See "Select Connected Stages" (p. 172) for how to do this. Afterwards the ***Start Up Axes*** dialog may open—you can close this dialog because at this point it is not necessary to reference the axis.
2. Open the ***Single-Axis*** window for the axis (“Single-Axis Window" (p. 97)). To do this, either use the ***Single Axis*** Window item in the View menu or call the corresponding ***Axis*** menu from the ***Axes*** tab card.
3. Expand the ***Single-Axis*** window via the rightmost > button.
4. In the rightmost pane of the expanded ***Single-Axis*** window, display the columns for the parameters you want to modify. To do this, click the ***Select parameters…*** button. See "Select Columns to be Displayed" (p. 26) for further details.
5. Type new values in the parameter fields. As long as a value is shown in blue, it is only present in PIMikroMove but not yet sent to the controller. Press Enter on your keyboard to send the value to the controller’s volatile memory.
6. Right-click in the center pane of the expanded ***Single-Axis*** window and select ***Save parameters as User Stage type...*** from the menu that appears.
7. To save the modified settings as a new stage, enter a new name for your stage in the ***as stage type*** field and click ***OK***.

   For the stage type entry, do not use stage names which already exist in the PIStages2.dat database. If a stage of the same name exists in PIStages2.dat and the user-stages dat file, the parameter settings from PIStages2.dat will be preferred when assigning that stage to an axis (e.g. in the Select connected stages dialog (p. 172)), and the settings from the user-stages dat file will never be used.
8. The new stage is now displayed on the ***Axes*** tab card, and you can work with it (e.g. reference the stage: right-click on the axis row and select the ***Start up axes...*** item, see "Start Up Axes" (p. 175)). If you want to further modify the stage parameters, use the ***Save parameters as User Stage type*** menu function again to save the changes.

Top (p. 185)

6.6 I cannot find parameter xyz in the GUI

If you want to view or modify a certain parameter of the axes, or you are missing a function it maybe only hidden from display. If you want to change the set of displayed parameters and functions see "Select Columns to be Displayed" (p. 26).

Top (p. 185)

6.7 What is the difference between controller macros and host macros?

Controller macros are stored on the controller. They can be started by the host PC but the controller itself parses and executes the commands in the macro.

Host macros are stored on the host PC. PIMikroMove will parse each line and execute some commands (start macro, wait for condition, …) and send the other commands to the controller.

So controller macros are faster than host macros, since the timing behavior of PIMikroMove depends on the operating system and each command must be transferred to the controller.

Since host macros are simple text files they may be used for different controllers—as long as the axis identifiers are still valid.

Host macros can even add macro functionality to controllers which would otherwise have no possibility to do repeated tasks.

Top (p. 185)

6.8 Are there any shortcuts with PIMikroMove?

There are the following general shortcuts:

F1 Opens help window

F2 Opens ***Connect Controller*** dialog

F3 Opens ***PIStageEditor***

F4 Opens ***Command entry*** window

F5 Opens ***Position Pad*** window (provided if only one ***Position Pad*** is available)

On the ***Host macros*** tab card:

- Ctrl+S saves the current macro
- Ctrl+R starts the current macro (without saving)

On the ***Controller macros*** tab card:

- Ctrl+S saves the current macro
- Ctrl+D sends the current macro to controller

- Ctrl+R starts the current macro; if the macro content has changed, the macro is sent to controller before it is started

Top (p. 185)