Nanonis SPM Control Software Documentation

The documentation is divided into 2 sections:

Concepts	General information about the system architecture.
Reference	Documentation for all modules and windows of the graphical user interface.

Version history / Changelog

For a short visual introduction of some frequently used operations check the Video Tutorials on the Nanonis homepage.

For general questions not answered in this documentation or any other requests don't hesitate to send an email to info@specs-zurich.com.

Back Home

Version history

Nanonis SPM Control Software V5 changes log.

2017-06-01 (R7249)

- Performance optimizations in the software. When operating the software for a long time software operation could become sluggish. This update should improve the performance.
- Bias Spectroscopy: added option to use an alternate Z-Controller setpoint before starting the spectroscopy measurement and added option to switch the lock-in on/off per segment in multi-line segment measurements. Also added an initial delay per segment to let the lock-in signals settle.
- Z Spectroscopy: added 2nd condition to the Auto-Retract functionality
- Script module: added new commands (if, break)
- Scripts can now be called from the Scan module in pattern or subgrid modes. Several fixes in the Script module, e.g. related to LUT.
- Fixed bug in Z-Spectroscopy which added one RT cycle per point with a specific timing configuration
- Fixed bug in Scan Multipass where list of channels available for recording was wrong
- For users of R7232 we strongly recommend this update due to a hrDAC issue for AO1 in R7232 which could lead to glitches at the output, and due to a bug using the Lock-In in Bias Specroscopy

2016-08-26 (R6772)

- Support for up to 3 SC5's, providing up to additional 16 output and 16 input channels
- Script module: improved user interface with easier access to commands and drag&drop capability
- AtomTracking controller: Added speed limit for tip movement
- Fixes a bug that could lead to a "Memory is full" error when using Bias Spectroscopy
- Fixes a bug in the Lock-in low-pass filter configuration
- Several other fixes and improvements

2016-05-20 (R6504)

- Lock-In: second module with one additional modulator and two more demodulators
- Programming Interface functions for full Lock-in functionality added (including filter settings and other parameters)
- Data Logger: new module to continuously save data to disk at RT speed
- Script module: new features like the option to wait for a digital line or edge, and to set voltages based on a lookup table
- History graph: new auto-saving routine
- Support for new generation of RC5 controller (NI PXIe-8840 RT)
- Note: after installing the RT-Upgrade required for this version, no previous versions can be installed
- Various fixes and minor improvements

2015-12-07 (R6151)

- Lock-In: all new Lock-in module with configurable filters (high-pass, low-pass, sync) and delay compensation
- Scripting Tool: new module allowing the execution of user-defined scripts directly on the Real-time engine
- Programming Interface: some new functions added
- Various minor improvements

2015-07-14 (R5702)

- Frequency Sweep: new amplitude and phase fit options for resonance frequency and Q-factor determination
- Generic PI Controller can now be assigned to any User Output
- Programming Interface: added access to advanced options in Bias- and Z-Spectroscopy functions
- Non-working mouse behavior on the Scan Control image on some systems fixed
- Move button failure in Motor Controls PMD, Omicron and Newport fixed
- Various other fixes and minor improvements

2015-04-24 (R5531)

- Scan Control: new Histogram
- Maintenance release, fixes various issues in the initial V5 release

2015-02-15 (First release of software V5, R5411)

Changes in V5 compared to V4.5:

- All new user interface

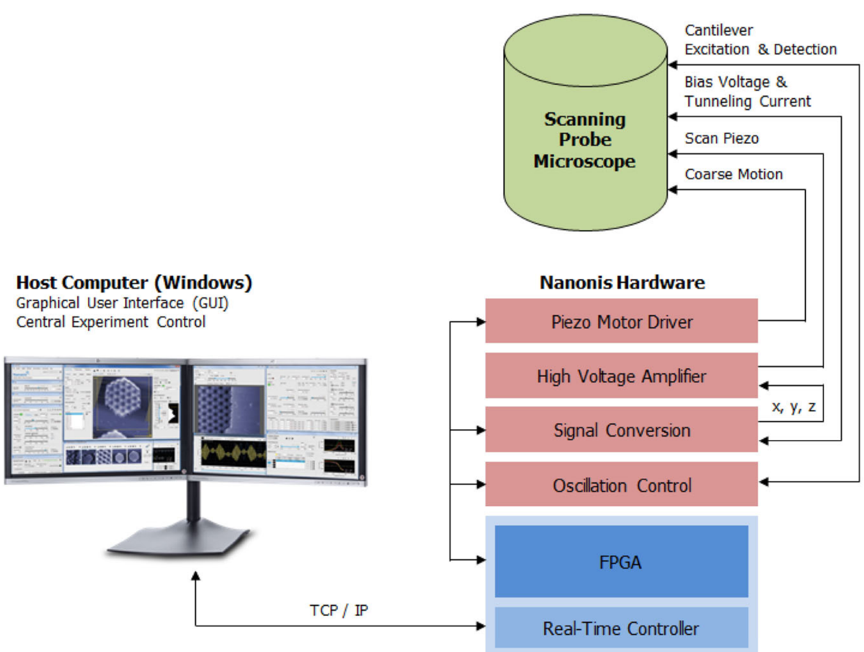
- New concept of Oscillation Control Module for multiple frequencies
- Various changes and new features throughout the whole software

Back Home

General Concepts

Back Home

System Architecture



The Nanonis SPM Control System is divided into three parts:

1. Host Computer

The Host Computer is a personal computer running under Windows where the user has access to the graphical user interface of the Nanonis SPM Control Software. All acquired data is stored on the local hard disk.
The connection to the real time engine is established via Ethernet (TCP/IP / UDP).

2. Real Time Controller

The Real Time Controller is a computer where a real-time operating system (RTOS) is installed for deterministic execution, and where all the time critical control loops and the data acquisition and communication processes are running.

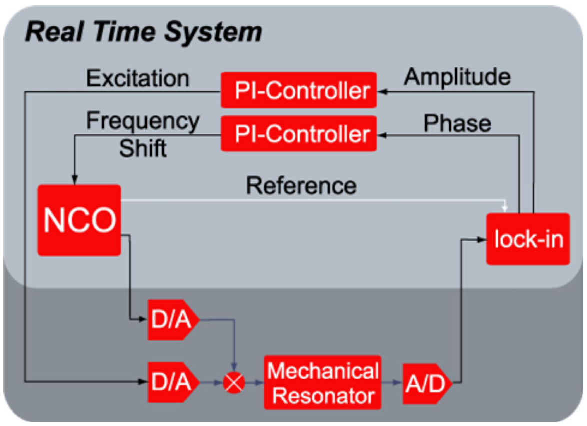
The Nanonis application software is installed on the hard disk and is loaded automatically when the real time engine is booted.
A National Instruments FPGA card provides multiple inputs and outputs to acquire and generate signals at high sampling rate.

3. Nanonis hardware

The RC5 module (Real-Time Controller) runs the real-time system and contains the FPGA card for the fast processing. It provides the digital connectors for the SC5 and OC4 modules and has 4 digital ports with 8 lines each, 4 high-speed digital inputs and 4 high-speed digital outputs, and clock input/output connectors.
The SC5 module converts the digital signals to analog and vice versa (DA and AD conversion), and it filters the analog signals. It provides 8 analog inputs, 8 analog outputs and 1 fast analog output.
The OC4 is the hardware device for the oscillation control module. It is used to excite and detect the oscillation of a mechanical oscillator.
The High Voltage Amplifier (HVA) takes the low voltage position signals from the SC5 and amplifies them for the scan piezo.
Please refer to the manuals of the corresponding devices for more information.

Back Home

Phase Locked Loop



The Phase Locked Loop (PLL) is implemented on the FPGA. Two proportional-integral controllers control the frequency shift and the excitation amplitude.
See the reference of the user interface of the Oscillation Control module.

Back Home

Signals

The system gives access to up to 80 signals, although some of them are just reserved for internal use. The user must select 24 signals out of the 80 to assign them to the 24 slots available in the software. This is done in the Signals Manager.
The list of 24 assigned signals is then available to work with in the different software modules (i.e. when selecting a channel to display in a graph).
The user has the flexibility to select the most convenient signals (not necessarily the signals physically available in the BNC inputs and outputs of the system).

This page shows the 80 signals typically available, although this might change depending on the hardware and software modules available in the system.

Physical Inputs

Current	Current flowing through the tip
Input 2-24	Auxiliary inputs (user definable)

Outputs

Bias	Bias voltage between tip and sample
Output 2	
Output 3	Auxiliary outputs (user definable)
Output 4	
X	Lateral scan signals
Y	
Z	Vertical signal controlled by the z-controller
Output 8-24	Auxiliary outputs (user definable)

PLL 1

Phase	Signal from the lock-in of the PLL
Amplitude	Signal from the lock-in of the PLL

Frequency Shift	Frequency modulation signal for the PLL
Excitation	Amplitude modulation signal for the PLL
PLL X 1	X- or Y-component from the lock-in of the PLL
PLL Y 1	

PLL 2

Phase 2	Signal from the lock-in of the PLL 2
Amplitude 2	Signal from the lock-in of the PLL 2
Frequency Shift 2	Frequency modulation signal for the PLL 2
Excitation 2	Amplitude modulation signal for the PLL 2
PLL X 2	X- or Y-component from the lock-in of the PLL 2
PLL Y 2	

Lock-in

LI Demod 1-8 X/R	X-Y or R-phi components of the Lock-in demodulators. For each demodulator one can select between X/Y or R/phi in the Lock-in module.
LI Demod 1-8 Y/phi	

Pulse Counters

Counter 1-4	Signal Frequency at Pulse Counter 1-4

Internals

Internal 23	Signals containing the RT additional signals assigned in the Signals Manager
Internal 24	

Reserved

All signals marked as **Reserved** are internally used by the system.

Back Home

Real time Engine

The control loops on the real time engine run in a time critical thread of the real time operating system. The loop rate can be set in tcp-receiver. This is also the rate at which the input signals are sampled and the output signals are output. Note that the input signals are actually sampled at a much higher rate (1 MS/s for the SC5) and then digitally filtered and oversampled to increase the resolution.
Some modules run directly on the FPGA, processing that data at much higher rates than the real time system.

In addition to the time critical loops there are communication processes running on the real time system. These run at lower priority and handle the communication over the TCP and UDP protocols which are not real time capable.

One of this communication processes provides a stream of data of the twenty four channels to the user interface. Since it is hard to transfer the signals at the full rate of the control loops, you can specify an oversampling for this data stream in tcp receiver. This is called the Signals Period and is usually set to 1-2 kHz (500 us - 1 ms). However, for special purposes you can set this Signals Period to 5 kHz (for example for enhanced spectrum analysis) or to 100 Hz (for example on slow machines or slow connections). This data stream is buffered and allows typically the host computer to be busy for a few seconds (this is not a real time machine).

The data stream of the 24 channels is then further oversampled on the host computer and provided in two different update rates: animations period and indicators period to be displayed in the various modules as numeric indicators or as animated graphical indicators.

Back Home

Measurements

"Measurements" is a concept built in the Nanonis control system that allows sweeping one variable while recording a set of other variables. Examples for measurements are the Generic Sweeper, Bias-sweep, Phase-sweep, Frequency-sweep and approach-retract.

Each module that exports a channel for measurements makes an entry in the list of available observables. You can select a set of observables with CTRL-clicking (clicking while holding down the CTRL key) the entries.
The speed of the measurements is usually controlled directly in the sweep module, but it is also available in the tcp-receiver. One data point is acquired per two measurements periods. The first period after the variable has been set to a new value is to let the observables settle to the new values, the second period is then used to take the measurement.

There are usually four data buffers to store the sweeps. They are displayed in different colors in a graph and you can select which curves to display or hide. Two graphs are available to display two different observables simultaneously.
The data buffer is stored in a raw ASCII file named nameXXX.dat in the current session directory, where XXX is a consecutive number starting at 000. The data is prepended by a header containing information about the type of data and when it was recorded. The user can add additional header attributes in the save dialog.

Back Home

Starting the Software

When you start the Nanonis software, the last session directory is retrieved from the registry. This is user specific, i.e. different users may use separate session directories. The software then loads and starts the modules. Each module loads its settings from the "Nanonis-Session.ini" file in the session directory (session settings file). If a module can't find its stored settings in this file, it will load the settings from the default settings file. Then, the layout is loaded from the session settings file. If no layout can be found in the session settings file, it is loaded from the default settings file.

When starting the software for the first time, it will ask for a session directory. Browse into the desired directory (just highlighting it won't work) and click "Select Current Directory". If there is no session settings file is in the selected directory, the modules will load their settings from the default settings file. Also, the layout will be loaded from the default settings file. Save layout and settings once you have set up the software to your personal preferences.

To see how to change the default settings file, please refer to the save and load settings chapter.

Back Home

Opening and closing modules

After starting the Nanonis SPM Control application, the modules are running, but their window may be hidden (depending on the saved layout).
The modules can be displayed through the menu of the main window.

The modules configure the respective input and/or output channel as you can see for example in the Signals Manager.
You can close the window but the module will continue to run.

Back Home

Working with Sessions

The concept of sessions in the Nanonis context is a way to store settings and data in one common place. So for different experimental setups or conditions or even for different users, you might want to have separate sessions.
You could, for instance, have one session for room temperature measurements before cooling down your microscope while you have another session for low temperature measurements. Similarly different users of the microscope might want to have different sessions to separate their data files and to customize the screen layout to their personal preferences.

The very first time you start the software, you will be prompted to select your first session directory. We suggest that you choose something like *C:\sessions\My Session*. To do this, you have to create the directory, move into it with the file browser and then press the *Select Cur Dir* button.

To change your session, you can change the session path in the Main window.

Note: when you browse for a session directory, you have to go into the directory and press "Select current directory". Just selecting the directory does not work.

The session directory is stored in the windows registry database for each user in order to remember the last session when you restart the Nanonis SPM control software.
All scan data and measurement data is stored in the session directory by default.

In each session directory there is a file called *Nanonis-Session.ini* where the settings are saved to.
The layout data might also be stored in the *Nanonis-Session.ini* file.

Note: you should have write access to the specified session directory as the settings and data are stored to this directory.

Tip: You can create a Sessions folder for example in your home directory or directly under C:\.
For each Session you are measuring a sample, create a directory for example containing the date like "2004-08-26". The session directory would then look like *C:\Sessions\2004-08-26*.

Back Home

Layouts and Settings

Settings is a generic term that regroup the modules controls value and configuration parameters.
Layout refers to the windows arrangement on the screen (windows positions and open/closed status).

Layout and settings might be loaded from and saved to various destinations.

1) **Presets**: the presets files are located in the *AppData\Nanonis\Certificate* folder*. They are independent of the session folder and thus considered as **global**. For better management, layout and settings presets files are kept separated.
2) **Session**: refers to the *Nanonis-session.ini* file located in the current session folder. Since every session folder has its own session file, it is considered as **local**. Both layout and settings values are stored in the same file.
3) **Other file**: allows you to select any other ini file where you want the layout and settings to be loaded from or saved to.

The idea behind the presets is to be able to load layouts or settings from a known starting point that the user has configured once.
From then, the user can configure his experiment, acquire and save data and be interested to save the current layout and settings to the *Session*. This way, the settings are stored along with the data file and can be retrieved for further investigation.

Settings

The settings files are ASCII files with the ini extension. Each module has its own section within the file.
When a module starts, it automatically loads the settings from the *Startup* settings file defined in the options window. If it can't find its section, it will load the settings from the default settings file.
You can save the settings for a single module through its file menu or save the settings of all modules via the main window file menu.
The modules menus look like this:

Load session	Load the module settings from the *Nanonis-session.ini* file located in the current session folder.
Save session	Save the module settings to the *Nanonis-session.ini* file located in the current session folder.
Load...	Load the module settings from a preset, the session file or any other file.
Save....	Save the module settings to a preset, the session file or any other file.

Layouts

The layout files are ASCII files with the lyt extension for the presets and ini extension for the others.
The file stores the position and the open/close status of the running modules and windows on the screen.

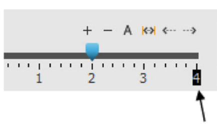
Back Home

Slider QuickScale Buttons

This documentation explains the Quickscale buttons that adjust the scales of slider controls and indicators. (Note: Graph Zoom and Pan buttons are explained in the generic graph handling.)

The QuickScale Buttons do not have any effect on the values displayed.

Remember that the scale can also be changed directly by altering the numeric values at the beginning and end of the scale. You can edit them by selecting them with the mouse.



	Function	Button / Icon	Explanation
Zoom	In	·	
	Out	·	
	Auto Fit once		Uses the currently "best" scale setting.
	Show Full Range	·	Adjusts the scale to the range used by the signal. Note that this button sometimes has 2 states (stays down when clicked). In this case, the range is automatically adjusted upon changes.
Pan	Left	·	Moves the scale to the left by a quarter of its range.
	Right	·	Moves the scale to the right by a quarter of its range.
	To Center	·	Centers the scale around the current value.
	To Zero	·	Places "Zero" in the middle of the scale.

Back Home

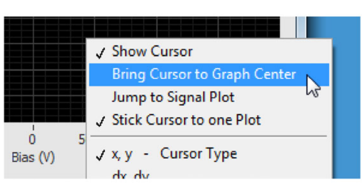
Graphs Handling

This Nanonis graphs and charts documentation explains the export of graph data, cursor handling and background functionality, zoom and pan functions, right click menus and more.

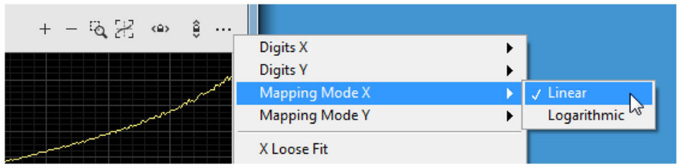
Some less frequently used graphs have the generic LabVIEW graph handling implementation.

Graph right-click Menu and More Options Menus

When right-clicking a graph, most available graph functions appear listed in a menu. Some items like e.g. Zoom and Pan include their related keyboard shortcuts and mouse actions.

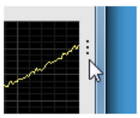


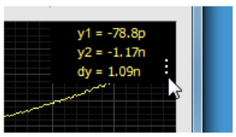
Additionally, More Options icons indicate a shortcut menu, click or right-click them to see the options. If two More Options icons are shown next to Zoom or Pan buttons, some Y-scale items are split up for the specified scale.

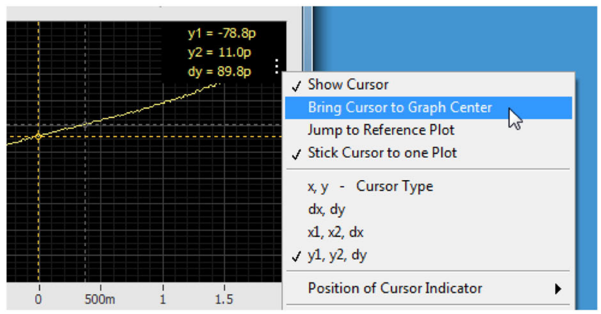


Cursor Handling and Cursor Types

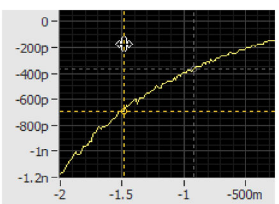
Cursor functions are available in the More Options menu of the cursor indicator and also in the right-click menu of the graph itself. The cursor type determines the cursor indicator information. Try them out and find the cursor type that suits the measurement task. The cursor indicator shows values for the selected plot. Plot selection can be changed by jumping to another plot or by free drag after deselecting “Stick Cursor to one Plot”.







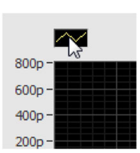
The cursor position can simply be changed by dragging the cursor line.

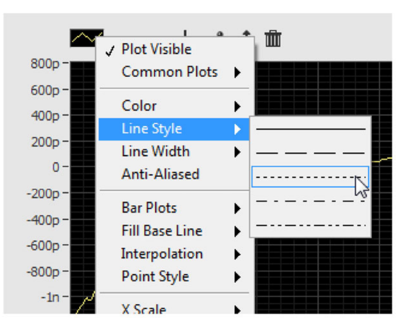


The position of the cursor indicator (for graphs where the cursor indicator is inside the plot area) can be re-positioned by dragging the indicator to the selected corner or by selecting the desired position in the menu.

Plot Properties, Colors and Styles

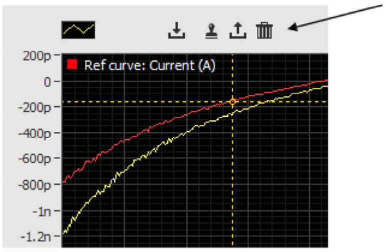
To change plot colors, styles and other plot properties, click on the plot legend icon to let the plot legend menu appear.





Background Functionality

Background functionality is available through buttons and graph right-click menu. Once a reference curve is pasted or loaded to the background, you can change its color by the reference color selector. The name of the background reference curve is shown next to the color selector.



Function	Button / Icon	Description
Paste Plot Data to Background		Paste the present signal curve to the graph's background.
Load Data and paste to Background		Load a saved curve to the graph's background to have a reference curve.
Clear Background Data		Delete the background curve.

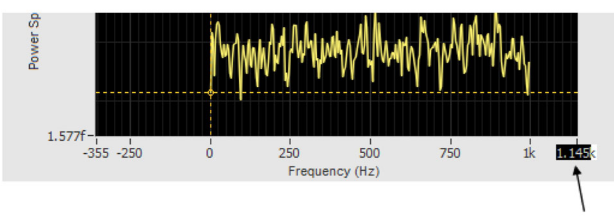
Export Functionality

Function	Button / Icon	Description
Save Plot Data to ASCII File		Save the signal curve to an ASCII file (.dat format).
Copy Data	-	Copy an image of the graph including plot color and graph background color to the clipboard.
Export Data to Clipboard	-	Copy the plot (and reference plot) data to the clipboard.
Export Data to Excel	-	Copy the plot (and reference plot) data directly to Microsoft Excel spreadsheet. Functionality is only available if Microsoft Excel is installed.
Export simplified Image...	-	Open the Export dialog with various file format options (.bmp, .eps, .emf) to copy a simplified image to clipboard or save to a file.

Zoom and Pan functions

The following Zoom and Pan functions can be found over the whole software, not all functions are available in every graph module.

Remember that the graph scales can also be changed directly by altering the numeric values at the beginning and end of the scale. You can edit them by selecting them with the mouse.



For keyboard shortcuts the window needs to be selected and the mouse positioned over the graph area.

	Function	Button / Icon	Key Shortcut a	Mouse Action	Note
Zoom	In	+	+ (Plus)	Mouse Scroll in / Mouse double Click	
	Out	–	- (Minus)	Mouse Scroll out / Ctrl + Mouse double Click	
	X Auto Fit once X Auto Fit on/off		- Ctrl + Shift + A	- Click&Hold / Double Click on X Auto Fit button	Only where Button visible, no Menu control. (New behavior replacing previous X Auto Fit Lock button.)
	Y Auto Fit once Y Auto Fit on/off		- (A) Ctrl + A	- (Double Click on plot area with middle mouse button) Click&Hold / Double Click on Y Auto Fit	Only for Spectra-type Graphs e.g. Spectrum Analyzer. (New behavior replacing previous Y Auto Fit

				button	Lock button.)
	(X + Y) Auto Fit once		A	Double Click on plot area with middle mouse button	For Spectra-type Graphs functionality is Y Auto Fit once
	Marquee Selection		S	S + Mouse Selection	
	Horizontal Marquee		H	H + Mouse Selection	Only available through Menu
	Vertical Marquee		V	V + Mouse Selection	Only available through Menu
	Show Full Channel Range		F	-	Full Range Y available for Charts e.g. Signal Chart, History X and Y for e.g. Beam Deflection
Pan	Y up		Up	Pan with Mouse	
	Y down		Down	Pan with Mouse	
	X left		Left	Pan with Mouse	
	X right		Right	Pan with Mouse	
	(X or Y or both) To Center	C	C, "." (Point)	-	
	(X or Y or both) To Zero	0	Z, "0" (Zero)	-	

Back Home

Numeric Control Behavior

All signals in the software are represented in physical units. The Controls and Indicators usually have a label specifying the SI unit of the signal (e.g. Bias (V)). The numeric field (control/indicator) contains the value, including the unit prefix (m (milli), u (micro), n (nano), p (pico)). The prefix changes automatically so that the number has in maximum three digits before the comma.

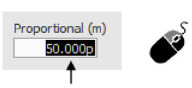
Nanonis Numeric Controls contain some additional usability features and safety measures. One major safety measure is the exclusion of the prefix when making a first selection of all digits, this to prevent huge unintended value changes when typing new digits and deleting the prefix by overwriting it. Another safety measure is the prevention of unexpected jumps to zero while decrementing (which was the case in older versions).

Selecting Numeric Digits with or without Prefix

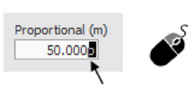
When selecting digits of a Numeric Control the first time, the prefix is excluded from the selection. This enables direct overwriting of digits without losing the actual prefix.

A selection can be done be mouse double clicking, by holding the mouse button down and move until the desired digit or by tabbing to a numeric control.

To select all digits including the prefix, select all digits including the prefix a second time.



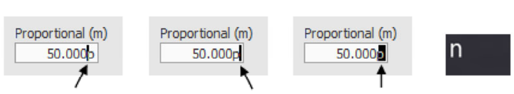
Selecting only a prefix can be done by mouse double clicking on the prefix letter or by holding the mouse button down and moving only over the prefix of the numeric control.



Deleting or changing a Prefix

To delete a prefix, position the cursor next to it or select the prefix and use the keyboard keys Delete respectively Backspace.

To change a prefix, position the cursor next to it (left or right) or select the entire prefix and type the desired new prefix. The old prefix is directly overwritten with both methods.



Reject an unwanted Entry

To reject an unwanted entry simply hit the “Escape” key.

Incrementing single Digits

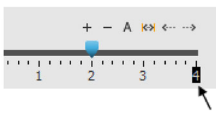
To increment a single digit, position the cursor on the right of the desired digit and use the mouse wheel or the Up key to increment respectively the Down key to decrement.

While incrementing or decrementing, most numeric controls in the Nanonis software show trailing zeros, this to make the changing of digits consistent and less nervous.

When decrementing and a digit arrives at the number one without having digits on the left, the cursor jumps to the right and continues. Therefore - while having a prefix - a Numeric control will never reach zero through incrementing or decrementing.

Numeric controls without decimal point or without the option of a prefix allow crossing zero at their lowest possible value before zero.

Incrementing single Digits is also available for graph or slide scale digits, but only with keys, not with mouse wheel yet.



Back Home

Nanonis Scan File Format (sxm) Version 2

A Nanonis scan data file consists of an ASCII header followed by the binary image data.

LabVIEW routines to load and save scan files (sxm) can be found in the Programming Interface.
A reference implementation for importing sxm data into Matlab or Octave can be found in loadsxm.m.
A reference implementation for reading and postprocessing the sxm data with Python can be found in loadsxm.py.

There are several 3rd party image processing applications that support the .sxm format, e.g. SPIP, Mountains, WSxM, Gwyddion, SPIW.

Header

The header consists of tags surrounded by colons (':') followed by one or more lines of values. The file has to start with the tag NANONIS_VERSION. The following tags are recognized as of *version 1* of the file format.

SCANIT_TYPE	Describes the data format for the binary data. Usually this is FLOAT MSBFIRST, i.e., the binary data is stored as 4-byte floating point data with the most significant bit first (big-endian, network order).
REC_DATE	Date at which the image was taken. The format is dd.mm.yyyy.
REC_TIME	Time at which the image was taken. The format is hh:mm:ss.
REC_TEMP	Temperature during the scan. Unit is Kelvin [K].
ACQ_TIME	Time it took to take the image. Unit is seconds [s].
SCAN_PIXELS	2x1 array, number of pixels in x- (pixels per line) and y-direction (number of lines).
SCAN_FILE	File name under which the file was originally stored.
SCAN_TIME	Time per line for forward and backward scan. Unit is seconds [s].
SCAN_RANGE	Scan range in x and y. Units are meters [m].
SCAN_OFFSET	Offset in x and y for the scan. Unit is meters [m].
SCAN_ANGLE	Rotation angle of the image. Unit is degrees [deg].
SCAN_DIR	Direction of the scan. Can be 'up' or 'down'.
BIAS	Bias voltage at the time when file was saved. Unit is volts [V].
Z-CONTROLLER	Contains information about the selected controller at the time when file was saved. The first line shows the parameter names (tab-separated), the second line gives the values. The parameters are: Controller name, on/off (0/1), setpoint, proportional gain, integral gain.
DATA_INFO	This field contains information about the channels in the file. The first line after the header contains a tab-separated list with the field description of the lines following. Usually this is Channel, Name, Unit, Direction, Calibration and Offset. The data in the fields Calibration and Offset can be used to reverse calculate the voltage read at the AD converters. It is not needed for the calibration of the data.
SCANIT_END	End of header.

Binary Data

The binary data begins after the header and is introduced by the (hex) code \1A\04. According to SCANIT_TYPE the data is encoded in 4 byte big endian floats. The channels are stored one after the other, forward scan followed by backward scan.
The data is stored chronologically as it is recorded. On an up-scan, the first point corresponds to the lower left corner of the scanfield (forward scan). On a down-scan, it is the upper left corner of the scanfield. Hence, backward scan data start on the right side of the scanfield.

Changes from version 1 to 2:

In version 2, the SCAN_PIXEL are in the indicated order, while in version 1 the pixels/line and # pixels were in the wrong order. In V2, the DATA_INFO items are tab-separated while in V1 it was space-separated.

Back Home

Nanonis ASCII Data File Format (dat)

A Nanonis .dat ASCII data file consists of a header followed by the experiment data in columns.

LabVIEW routines to load and save ASCII data files (.dat) can be found in the Programming Interface.
A reference implementation for importing .dat data into Matlab or Octave can be found in loaddat.m.

Header

The header consists of key-value pairs separated by a tab character (\t or ASCII code 9). There can be any number of key-value pairs, including zero. An example of a key-value pair is (without the single quotes): 'Integration time (s)\t100E-6' where \t stands for a tab character.

Data

The data part starts with a line containing the text '[DATA]'. On the next line the channel names are listed (tab separated). This is followed by the data in engineering or scientific notation (e.g. -74.2584E-12), one channel per column (the columns are also tab separated).

Example

An ASCII data file could look like follows:

```
Experiment Sweep

Saved Date 04.07.2013 14:51:54

User

X (m) 0.000000E+0

Y (m) 0.000000E+0

Start time 04.07.2013 14:51:13


[DATA]

Output 3 (V)    Input 3 (V)

-1.00000E-3     -380.673E-6

-980.000E-6     -370.535E-6

-960.000E-6     -359.088E-6

-940.000E-6     -332.361E-6

-920.000E-6     -315.218E-6

-900.000E-6     -294.795E-6

-880.000E-6     -275.864E-6

-860.000E-6     -252.678E-6
```

…

Back Home

Nanonis Binary File Format (3ds) Version 1

A Nanonis binary file consists of an ASCII header followed by the binary experiment data.

LabVIEW routines to load and save binary data files (3ds) can be found in the Programming Interface.
A reference implementation for importing 3ds data into Matlab or Octave can be found in load3ds.m.

Header

The header consists of key-value pairs separated by an equal sign ('='). Some keys are mandatory (see list below), but arbitrary items can be added at will. An example of a key-value pair is (without the single quotes): 'Grid dim="24 x 24"'. The value is enclosed by double quotes if it contains spaces.
The header is finalized by a line containing the text ':HEADER_END:' followed by the windows end of line characters CR/LF (carriage return/line feed; ASCII codes 0D and 0A). The following keys are defined (required) in version 1 of the binary file format.

Grid dim	Dimension of the grid in number of points. Format is 'Nx x Ny' with Nx = Number of points in x, Ny = Number of points in y.
Grid settings	Grid position and dimension in physical units (m). Format is 'Cx;Cy;W,H,A' with Cx = grid center x, Cy = grid center

	y, W = grid width, H = grid height, A = angle. Cx, Cy, W, H are in meters (m), A in degrees (deg).
Sweep Signal	Name of the swept parameter. This is a string, usually a signal name followed by its units in brackets. Example: 'Bias (V)'.
Fixed parameters	List of required parameters. These are stored at the beginning of each experiment. Usually the fixed parameters are 'Sweep Start' and 'Sweep End', i.e. the limits of the sweep signal.
Experiment parameters	Additional parameters stored for each experiment. These can contain the position where the experiment is taken ('X (m)', 'Y (m)', 'Z (m)') and other parameters. It's recommended to store at least the Z position as a parameter as this can be used to reconstruct the topography afterwards.
# Parameters (4 byte)	Total number of parameters stored with each experiment (= number of fixed parameters + number of experiment parameters). Each parameter is stored as a single precision floating point number using 4 bytes (big-endian).
Experiment size (bytes)	Size of experiment data in bytes. Each floating point number uses 4 bytes. When acquiring 1 channel forward and backward, 256 points, this will be 2 x 256 x 4 bytes = 2048 bytes.
Points	Number of acquired points in the experiment (e.g. bias spectroscopy).
Channels	Channels acquired in the experiment, separated by semicolons (';'). When acquiring data forward and backward 2 channels will be listed. Example: 'Current (A);Current [bwd] (A)'.
:HEADER_END:	End of header.

Binary Data

The binary data begins after the header. All data is stored in 4 byte big endian floats with the most significant bit (MSB) first.
The experiments aren't separated, all data is written into the file continuously. Each experiment starts with the fixed parameters, followed by the experiment parameters and the experiment data (Channels one after the other). The size of the experiment data is defined in the header so it's easy to read a specific experiment. From the start of the binary data an experiment including the fixed and experiment parameters always takes (# Parameters) * 4 + (Experiment size) bytes.

Example

The header of a binary file could look like follows:

```
Grid dim="24 x 24"
Grid settings=0.000000E+0;0.000000E+0;6.880776E-9;6.880776E-9;-2.244028E+1
Sweep Signal="Bias (V)"
Fixed parameters="Sweep Start;Sweep End"
Experiment parameters="X (m);Y (m);Z (m);Z Offset (m);Settling time (s);Integration time (s)"
# Parameters (4 byte)=8
Experiment size (bytes)=2048
Points=256
Channels="Current (A);Current [bwd] (A)"
Experiment="Grid Spectroscopy"
Date="30.08.2006 14:52:45"
User=user1
:HEADER_END:
```

continued by, in hexadecimal representation:

```
0d 0a c0 00 00 00 40 00 00 00 b0 f6 18 55 b1 94 18 42 b0 84 7c 66 00 00 00 00 39 51 b7 17 38 d1 b7 17 ae ca c3 f0 ae
c6 60 e3 ae c2 31 bf ae bf 5a a3 ae c5 af f1 ae be e8 94 ae ba 8e 31 ae b7 d9 79 ...
```

where 0d 0a is the end of line after the :HEADER_END:. Then, c0 00 00 00 (= -2) is the first fixed parameter (Sweep Start) and so on. 38 d1 b7 17 (=1E-4) corresponds to the last parameter (Integration time (s)). After that 256 values of the first data channel (Current (A)) will follow, starting with ae ca c3 f0 (= -9.2207E-11). Next will be 256 values of the second data channel (Current [bwd] (A)). As no more channels were acquired this will finalize the first experiment and the data will continue with the parameters of the 2nd experiment.

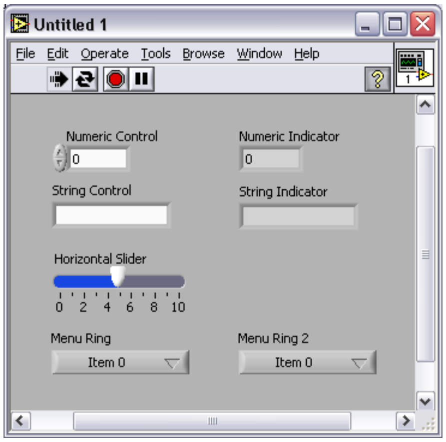
Back Home

LabVIEW Controls and Indicators

This chapter is a short introduction in LabVIEW front panel elements available for user interaction. In general two types are distinguished by LabVIEW, the controls and the indicators.

Note: Nanonis specific Numeric control handling is explained in the user interface section.

The figure below shows a basic set of LabVIEW controls and indicators. The controls on the left allow a user input. These can be Numeric Controls for number input (top left), String Controls for string input or sliders like the one shown below the String Control.



The controls can have certain constraints such as the number of decimals, the range in which an input is valid, the length of a string, the type of a number (integer or floating point) and many more. It should be mentioned that the scale of a slider does not necessarily have to be linear, sometimes logarithmic scales are used. Often, a slider is combined with a digital indicator or control. The user can choose to either use the mouse or cursor keys to manipulate the value continuously or to use the control field to type in the number directly.

Another control type is the Menu Ring as shown in the bottom row. This control lets the user pick an element from a predefined list.

On the right of the figure a set of indicators are shown. In LabVIEW nearly every control has its corresponding indicator and vice versa. The indicators distinguish themselves from the controls in not allowing the user to change the displayed value. As the name suggests, they are merely used to convey information to the user.

Back Home

Working with LabVIEW Graphs

The following figure shows a generic LabVIEW graph, implemented in some less used modules of the software. More frequently used graphs and charts have the Nanonis graph handling implemented.

One or more data traces are displayed in the graph. As an example a sine function with a red trace is desplayed. The remaining elements are explained in the following.

Legend
: The legend shows the color code and name of a data set as displayed in the graph. You can change the color, line type and point type by clicking on the legend and selecting the appropriate option from the context sensitive menu.

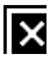
Cursor Legend
: At times a cursor is displayed in the graph. This is usually the case when the user should select a point in the graph, e.g. the working point of the interferometer as described.
The first field shows the name of the cursor. The second and third fields indicate the x-and y- coordinates, respectively, of the current position. As the two fields are controls, you can directly set the cursor position by entering the x- and y- values of the desired position instead of moving the cross hair with the mouse.
With the three icons to the far right you can customize the behavior of the cursor like locking it to the data in the graph or modify color and appearance.

Scale Legend
: The scale legend describes the names and properties of the axes in the graph. When the lock to the right of the name is open, it is possible to change the scale properties. The icon to the right of the lock is used to auto-scale the axis. So in case you have totally lost yourself, clicking on this icon will auto-scale back to the

extent of the data.
The icon to the very right opens a context menu from which you can select the format of the labels (decimal, scientific, ...), the number of decimals displayed, the mapping mode (linear, logarithmic), whether or not the scale and the labels should be visible and the color of the grid.

Graph Palette

The three icons in the graph palette allow you to choose between different reactions of the graph to movements of the mouse. Highlighting the first icon will bring the graph into cursor mode where it is possible to move the cursor (if there is one) with the mouse. To do this, grab the cursor with your mouse and move it to a new position.
The second icon is used for zooming. Clicking onto the icon will open a small window with six possibilities to choose from. The first and fourth will allow you to zoom into the desired region by selecting the window with the mouse. The second will only zoom the x- axis, the third zooms only the y- axis. The fifth and sixth icon allow you to zoom interactively in and out by holding down the right mouse button while pointing the mouse inside the graph window.
Instead of zooming with the mouse, it is also possible to enter the new scale directly by clicking on the first or last number and entering a new value.
Clicking on the third icon with the hand will switch to panning mode where you can move the whole graph within the window by grabbing it with your mouse.



Six zoom mode options.

Please beware that graphs do not necessarily display data in the form y = f(x), but can also be used to display a scanned image, which in that case would be c(x,y) = f(x,y) where c is the color code at the position (x, y) or a parametric plot in the form of x = x(t), y = y(t), t in [0;1].

Back Home

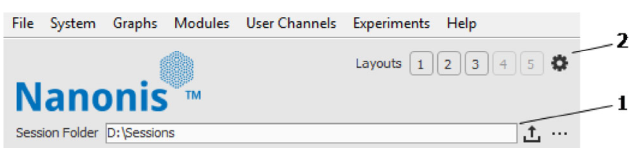
Nanonis SPM Control Software Reference

This is the reference manual for all windows and panels of the graphical user interface.

Back Home

Nanonis SPM Controller Main Window

From this panel all the modules of the Nanonis Controller can be opened via the menu bar.



1	Session Folder	Manage the session folder. - **Session Folder**: display the current session folder. Opens a browser window to select the session directory. Important note: go **into** the desired directory and then press "Select current directory". More options related to the Session folder: - *Open session directory in Windows Explorer* - *Create new session directory with current date as name* - *Open Help Reference (F1)*
2	Load Layout & Main Options	Use these buttons to load layouts and change windows arrangement. Notes: - It is possible to configure which layout is assigned to a button in the *System>Options* window. - The layout assigned to the button is shown in the button tip-strip. Opens the Main Options window.

File menu

Layouts and settings might be loaded from and saved to various destinations:

1) **Presets**: the presets files are located in the *<AppData>\Nanonis\Certificate <Certificate no>* folder*. They are independent of the session folder and thus considered as **global**. For better management, layout and settings presets files are kept separated.
2) **Session**: refers to the *Nanonis-session.ini* file located in the current session folder. Since every session folder has its own session file, it is considered as **local**. Both layout and settings values are stored in the same file.
3) **Browse**: allows you to select any other ini file where you want the layout and settings to be loaded or saved.

(*) <AppData> folder:
- In Windows XP: *C:\Documents and Settings\All Users\Application Data*
- In Windows Vista/7/8/10: *C:\ProgramData*

Layouts and settings can be easily managed from the Main Options window.

Load Layout	Select from the submenu which layout to load. The button assigned to every layout (See “2”) is next to its name. - **Presets**: load the layout from a preset file. - **Session**: load the layout from the *Nanonis-session.ini* file located in the current session folder. - **Browse**: select an ini file that contains a layout.
Save Layout	Select from the submenu to which layout file the current windows arrangement is to be saved. The button assigned to every layout (See “2”) is next to its name. - **Presets**: save the current windows arrangement to a preset file. - **Session**: save the current windows arrangement to the *Nanonis-session.ini* file located in the current session folder. - **Browse**: select an ini file where to save the current windows arrangement.
Load Settings	Select from the submenu which settings to load. - **Presets**: load the settings from a preset file. - **Session**: load the settings from the *Nanonis-session.ini* file located in the current session folder. - **Browse**: select an ini file that contains settings.
Save Settings	Select from the submenu to which settings file the controls value are to be saved. - **Presets**: save the current controls value to a preset file. - **Session**: save the current controls value to the *Nanonis-session.ini* file located in the current session folder. - **Browse**: select an ini file where to save the current controls value.
Quit	Stops the controller and closes the Nanonis application.

System menu

TCP receiver	Opens the TCP receiver window. There you can view and adjust settings concerning the communication with the real time engine.
Sent Commands	For debugging purposes only. Lists the last commands sent to the real time engine.
Peek / Poke	CAUTION! To be used by experts for debugging purposes only! View (peek) or modify (poke) various settings in the real time engine.
Signals Manager	Displays current values and signal calibrations and offsets, and gives access to more advanced signals.
Options	Opens the options window.
Status Window	Opens the status window.
Plugins	Direct access to the quick plugins and the Plugins Manager.

Graphs menu

Signal Monitor	Opens the Signal Monitor which display 4 selectable signals as numeric indicators and as graphical animation.
Signal Chart	Opens the Signal Chart which displays 2 selectable signals in graphs over a short period of time.
History	Opens the History Chart which displays 2 selectable signals in graphs over a medium period of time.
Long Term Chart	Shows a Long Term Chart of one selectable signal over a long period of time.
Oscilloscope	Opens the Oscilloscope for a further signal analysis.
Oscilloscope – Two Channels	Opens the Two Channel Oscilloscope with two selectable channels and pre-trigger capabilities.
Oscilloscope – High Resolution	Opens the High Resolution Oscilloscope with up to 1 MS/s and up to 1 million points.
Spectrum Analyzer	Opens the Spectrum Analyzer to analyze a signal in the frequency domain.
Long Term Spectrum	Opens the Long Term Spectrum to analyze the spectrum of a signal over a long time.
Signal to Sound	Opens the Signal to Sound module which converts the DC value of any signal to a sound.

Modules menu

Bias	Opens the Bias panel where the bias voltage can be set.
Current	Opens the Current window.
Z-Controller	Opens the Z-Controller to set the control signal and adjust control parameters for the tip-sample distance controller.
Oscillation Control *	Opens the Oscillation Control panel where the control parameters can be adjusted.
Beam Deflection *	Opens the Beam Deflection module to calibrate and display the deflection signals.
Interferometer Control *	Opens the Interferometer Controller (W-Controller) module to calibrate and control the interferometer.
Scan Control	Opens the Scan control window to control the scanning process.
Piezo Calibration	Opens the Piezo Calibration window where the piezo characteristics (nm/V) can be set.
Lock In *	Opens the Lock In module to modulate any signal and observe amplitude and phase on another signal.
Digital Lines	Opens the Digital Line module to configure or control the digital lines.
Motor Control	Opens the Motor Control module to control the coarse positioning.
Function Generator *	Opens the Function Generator to generate waveforms on an output.
Kelvin Controller *	Opens the Kelvin controller module.
Generic PI controller *	Opens the Generic PI controller.
F-out Scala *	Opens the output module for combination with Scala controllers.
Pulse Counter 1/2 *	Opens the Pulse Counter module 1 or 2, respectively.
Atom Tracking *	Opens the Atom tracking module.
Laser control *	Opens the Laser control module.

OC4 Sync *	Opens the Digital I/O module of the OC4. Used to synchronize external devices with the Oscillation Controller.
Scripting Tool *	Opens the Script Tool which allows creating scripts with lists of commands to deploy to the real time controller. They will run at real time speed.
Data Logger	Opens the Data Logger module which allows to select a set of channels, start their acquisition at a specific averaging rate, and save them into file for as long as the user sets the acquisition duration.

*) optional modules

User Channels menu

User Input x	Opens the Input Configuration module for input channel x (x is the channel number).
User Output x	Opens the Output Configuration module for output channel x (x is the channel number).

Experiments menu

Generic Sweep Module	Starts the module to sweep an arbitrary signal.
Bias Spectroscopy	Perform bias spectroscopy measurements like I/V curves.
Bias Sweep	Sweep the bias voltage and observe other channels.
Z Spectroscopy	Perform Z spectroscopy measurements like I/Z curves.
Approach/Retract	Approaches/retracts the tip to/from the sample and lets you observe different signals.
Spectrum	Perform a frequency sweep and observe different signals.
Phase Variation	Perform a phase sweep and observe different signals.
Bode Plot	Acquire a bode plot using the oscillation control module.
Transfer function	Opens a frequency sweeper (logarithmic scale) for the lock in to record a transfer function.

The availability of some Experiments depends on the licensed modules.

Help menu

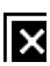
From this menu you can:

- open the documentation of the Nanonis controller software through the **Reference** item
- check the version history of new features and improvements in **What's new**
- open the **Info** window to check on current software versions and certificate
- open the **License Agreement**
- launch the **Lock** window to prevent other users play with the Nanonis user interface while an experiment is running
- check if a new software release is available to download in **Check for Updates**

Status Window

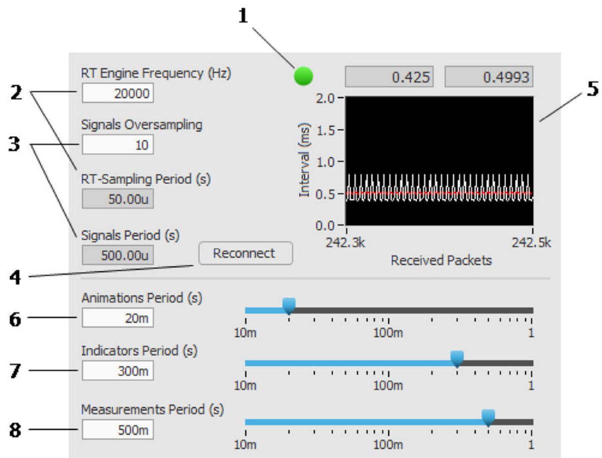
Display the status of several modules and features (i.e. status of scanning, Z-controller on/off, tilted plane, etc…).

The transparency of the window can be changed with keys ("+" to make it more opaque, "-" to make it more transparent). *Pin* button on the top right corner switches the behavior of the window between floating or not floating, which means that the window can be set to be always on top of the others.



TCP receiver

The TCP Receiver panel is used to define the loop rates of the real time machine, the data transfer rate to the host computer and the update rates of the numerical indicators and animated graphics.
The state of the TCP/IP connection between the front end and the real time engine can be inspected. The graph displays the time between two TCP packets that arrive from the RT-Engine.
It is a good practice to open this panel first before starting other modules. Initially it takes about five seconds until the connection is established. During normal operation of the Nanonis Software this panel can be closed.



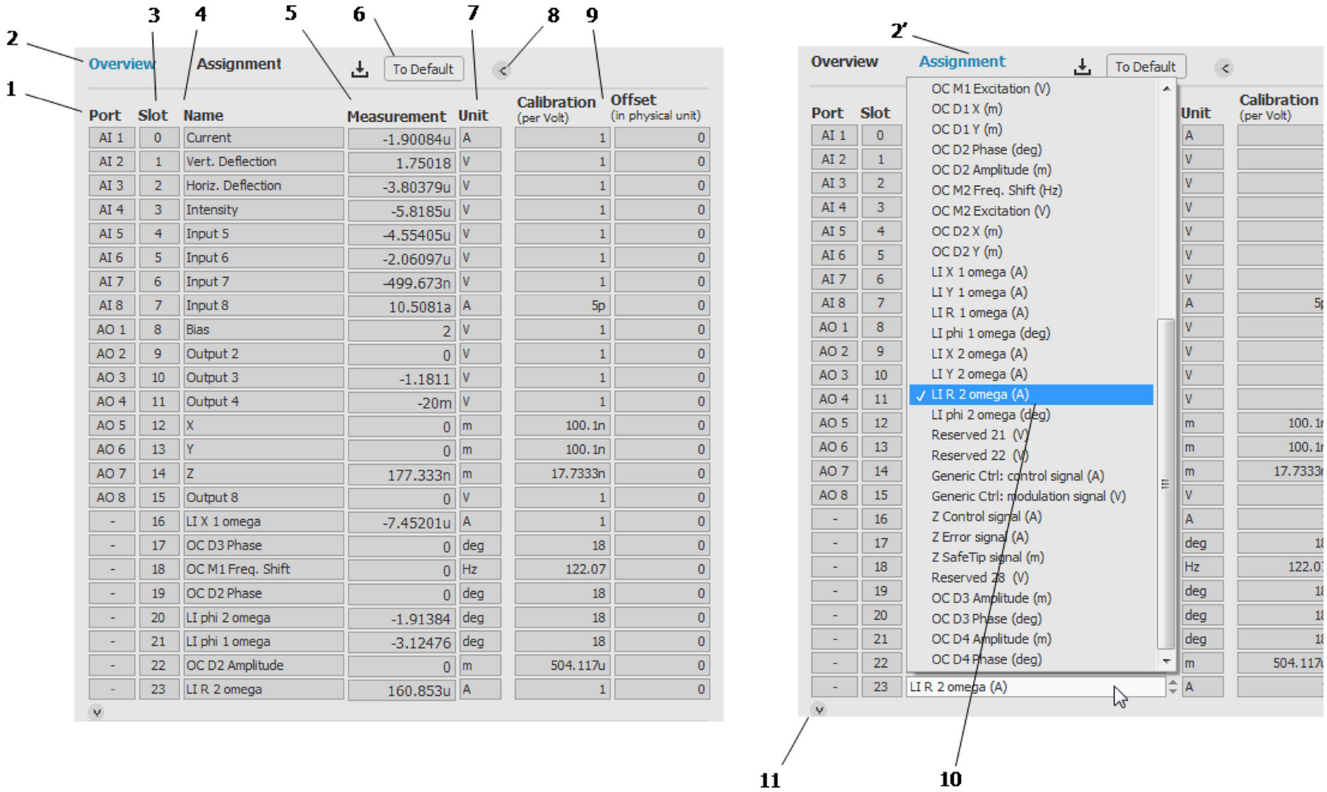
1	Connection status	The indicator turns green as soon as the connection between real time computer and user interface is established. If the connection breaks for some reason (e.g. cable removed from slot), pressing the button makes the software try to reestablish the link.
2	RT Engine Frequency (Hz) / RT-Sampling Period (s)	The **RT-Sampling Period** indicator displays the time between two successive iterations of the control loops on the real time engine. It can be changed in **RT Engine Frequency**. Values here are in Hz, i.e. the inverse of the sampling period. All controllers (e.g. Generic PI Controller, z-controller, w-controller) and spectroscopy measurements (e.g. 3D Sweeper, Bias Spectroscopy) run at this rate. The value should usually be 20 kHz. It is not advisable to change it. NOTE: This parameter is loaded from the license file. If you want to change its default value, edit the key "RT-Engine Freq (Hz)" in the section "System". Create a backup copy of this file before altering it since any other changes to it will result in an invalid file and you won't be able to start the software anymore!
3	Signals Oversampling / Signals Period (s)	The **Signals Period** is the rate at which the signals are transferred to the front panel. This is usually lower by a factor of 10 than the sampling rate, because an internal oversampling of the signal is done on the real time engine. You can reduce the oversampling down to 1 in order to resolve higher frequencies in the Spectrum Analyzer.
4	Reconnect	Tries to re-establish a broken connection to the real time engine.
5	Received packets	The graph shows the current interval between the received message packages from the real time engine in white and a moving average in red. The interval should correspond to the **Signals Period** from above. Periodic fluctuations of the white curve are perfectly normal due to the packet size, but the red line should be more or less flat.
6	Animations period	Update rate of animated graphical indicators. These are e.g. some graphs & sliders. A reasonable value is 20 ms (50 updates per second). Increase this period to reduce the processor load for the graphical user interface, especially on slow computers.
7	Indicators period	Update rate of digital indicators, e.g. the numbers displayed besides each slider. Here, 3 updates per second, or 300 ms is enough.
8	Measurements period	The **Measurements period** is the integration time for precise measurements (averaging over specified period), like recording of a force-distance curve or a resonance of a cantilever. For fast measurements with small steps, a value of 50 ms may be reasonable. For normal use, 300-500 ms is a good value, but for recording a resonance of a high-Q cantilever, values of several seconds might be necessary. Usually this parameter doesn't need to be set from this module, the sweep modules will set this value according to the sweep timings.

Back Home

Signals Manager

The Signals Manager has 2 separate tasks. The **Assignment tab** (" 2' ") allows the user to assign any of the up to 80 signals available in the Nanonis Control System to any of the 24 slots available in the host, and the **Overview tab** (" 2 ") *summarizes* the information of the 24 slots. The assigned signals define the list of available signals in the Nanonis software.

The 80 signals are separated in 3 groups: inputs, outputs and internal channels.
The input and output channels correspond to the signals physically present on the BNC ports of the Nanonis Signal Conversion unit. The internal channels can't be accessed directly; although you can use a User Output in Monitor mode to have it available physically.
The internal channels are of 3 different types. These can be signals used by the PLL 1, PLL2, the Lock-in module and the channels used by the Assignment Tool (Internal 23, Internal 24), and the rest are channels reserved by the system.

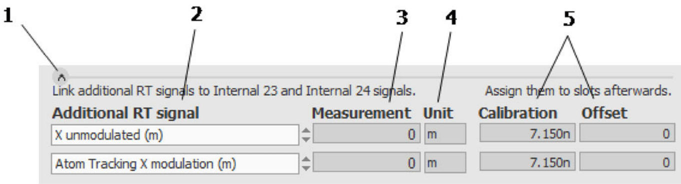


1	Port	Indicates on which BNC port of the Nanonis Signal Conditioning unit the signal is present. This is only valid for the physical inputs and outputs.
2, 2'	Overview/Assignment tabs	The Overview tab just displays the complete list of assigned signals, whereas the Assignment tab allows assigning one of the 48 signals available in the Nanonis Control System to one of the 24 slots available in the host.
3	Slot	Indicates the slot index (0-23). This information is helpful when using the Nanonis Programming Interface since the channels are identified by its slot index.
4	Name	Signal name which cannot be edited in general, except for the user inputs and user outputs whose name can be modified in the corresponding module.
5	Measurement	Indicates the signal value oversampled during **Measurement period**. The Measurement period can be changed in the TCP receiver.
6	Save / To Default	Store the current signal configuration to be loaded the next time the software starts. **To Default**: load the original signal configuration (installed originally with the software).
7	Unit	Signal physical unit.
8	Expand/Contract button	Show/hide the **calibration & offset** information.
9	Calibration & Offset	The signals voltages are converted to real world physical values according to the calibration & offset parameters: Physical signal = (Voltage * calibration) + offset. Notes: - The voltage corresponds to the one presents on the BNC of the Signal Conversion unit (*i.e.*, +/-10 V range). - The **calibration** is in [physical unit/V]. - The **offset** is in [physical unit].
10	RT Signal to Slot assignment	In the Assignment tab, when clicking on a RT signal, it displays the complete list of 48 signals available in the Nanonis Control System. Any of them can be selected and automatically assigned to one of the 24 slots available in the host.
11	Show Assignment Tool	Display the Assignment Tool configuration

Assignment Tool

The Assignment Tool is an advanced section used to assign RT signals to the Internal 23 and Internal 24 channels.
These RT signals are not usually accessible in the software. But if one of them is required by the user, this section gives the option to link the required signal to one of the 2 internal channels intended for this.

When a signal is assigned to an internal channel AND the internal channel is assigned to a slot in the Signals Manager, the signal can be measured, displayed in the graphs, and/or saved into a file as a normal signal does through the Nanonis software.
You could even use a User Output in Monitor mode to have it physically available.



1	Hide Assignment Tool	Hide the Assignment Tool configuration
2	Additional RT signals	List of RT signals which can be assigned to channels. This list depends on the Nanonis modules that are licensed.
3	Measurement	Indicates the signal value oversampled during **Measurement period**. The Measurement period can be changed in the TCP receiver.
4	Unit	Displays the signal physical unit.
5	Calibration & offset	The signals voltages are converted to real world physical values according to the calibration & offset parameters: Physical signal = (Voltage * calibration) + offset. Notes: - The voltage corresponds to the one presents on the BNC of the Signals Conversion unit. (*i.e.*, +/-10 V range). - The **calibration** is in [physical unit/V]. - The **offset** is in [physical unit].

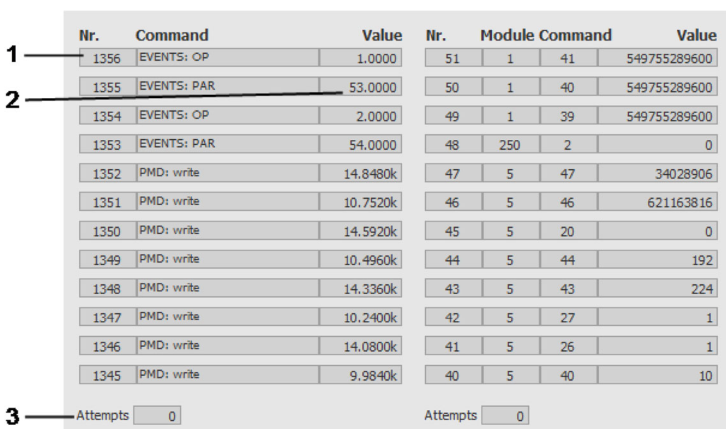
Back Home

Sent Commands

The "Sent Commands" window is for debugging purposes only.

It shows the last commands ("1") sent to the real time engine via TCP/IP. The **attempts** indicator ("3") shows the number of attempts for sending the last command. It is usually 0, but when the TCP/IP connection is bad, commands are repeated up to 10 times which will be indicated here.

The left side shows the commands sent to the real-time engine (parameters), the right side shows commands going to the FPGA directly. Commands sent from the real-time engine to the FPGA are not visible here.



1	Command Number	Continuous index number of the commands sent
2	Value	This is the data represented as a floating point number.
3	Attempts of last send	Number of attempts made to send the last command.

Back Home

Peek / Poke

CAUTION! The "Peek/Poke" window is for debugging purposes only. You can read and write values from and to the real time engine. DON'T send any commands to the real time machine unless you really know what you do.

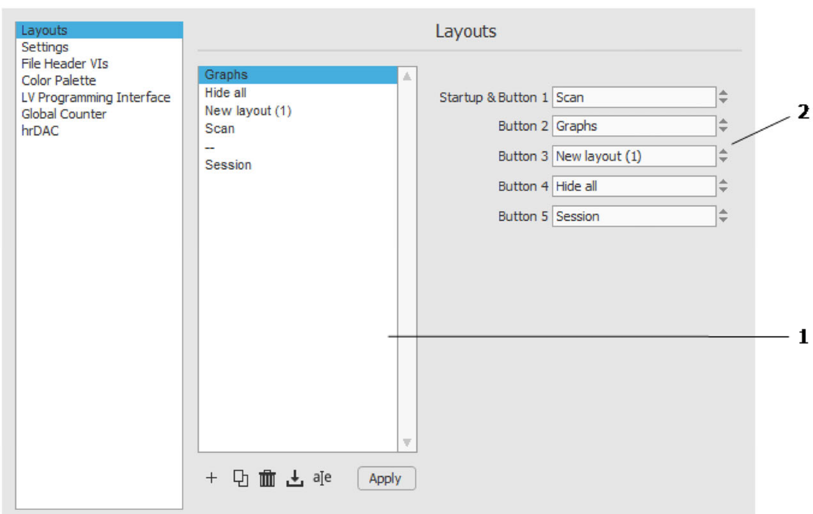
1	Signal selector	When choosing a signal, the actual value is read from the real time machine and displayed in "2".
2	Current Value	The current value of the signal selected in the **Signal Selector** ("1") is displayed here. By modifying the value it is written to the real time machine. Be careful when altering values!

Back Home

Main Options

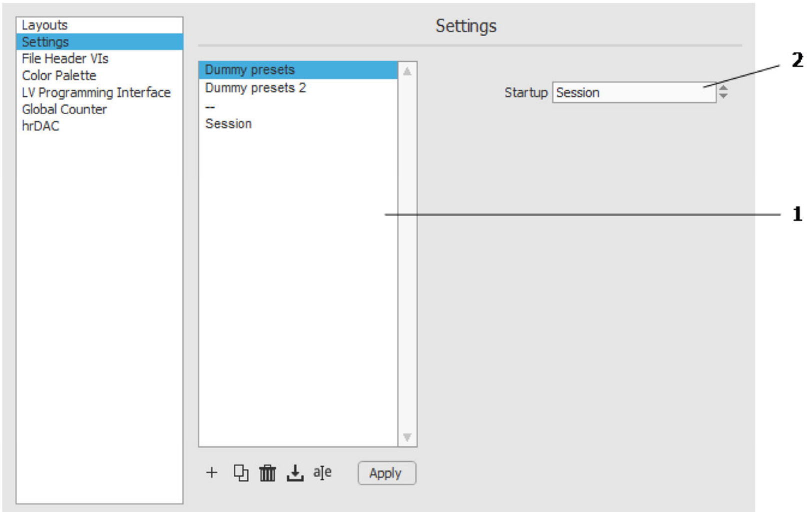
This window contains options for the software that are not specific to a single module.

I) Layouts



1	Layouts list	Display the list of layouts. - **Presets**: the presets files are located in the *AppData\Nanonis\Certificate\Layouts* folder*. They are independent of the session folder and thus considered as **global**. - **Session** layout refers to the windows arrangement saved in the *Nanonis-session.ini* file located in the current session folder. You cannot duplicate, delete or rename this item. Apply an action to the selected layout or to the list: + create a new preset layout. duplicate the selected layout. delete the selected preset layout. save the current windows arrangement to the selected layout. a]e make editable the selected layout name. The **Apply** button applies (load) the windows arrangement stored in the selected layout. Note: it is possible to rename the presets by clicking directly on the list item. *) *AppData* is located by default in these locations: - with XP: C:\Documents and Settings\All Users\Application Data\ - with Vista/7/8/10: C:\ProgramData\
	Layouts buttons	Select from the controls which layout is assigned to which buttons of the Main Window. The assignment of **Button 1** is also used to define the **Startup** layout (i.e. which layout is loaded when starting the Nanonis software).

II) Settings



1	Settings list	Display the list of settings. - **Presets**: the presets files are located in the *AppData\Nanonis\Certificate\Settings* folder*. They are independent of the session folder and thus considered as **global**. - **Session** settings refers to the controls settings saved in the *Nanonis-session.ini* file located in the current session folder. You cannot delete or rename this item. Apply an action to the selected settings or to the list: create a new preset settings file. duplicate the selected settings file. delete the selected preset settings file. save the current controls settings to the selected settings file. make editable the selected settings file name. The **Apply** button applies (load) the controls settings from the selected settings file. Note: it is possible to rename the presets by clicking directly on the list item. *) *AppData* is located by default in these locations: - with XP: C:\Documents and Settings\All Users\Application Data\ - with Vista/7/8/10: C:\ProgramData\
2	Startup	Select from the control which settings file is loaded when starting the Nanonis software.

III) File Header VIs

This page lets you select VIs that will return key-value-pairs to be stored in saved files. In most places where data can be saved throughout the software you can select what parameters to store in the header of the saved file. The VIs specified in the list are named as "Ext. VI 1/2/3".

Note: LabVIEW must be installed and running on the host PC if you want to use this feature. VI Server must be enabled in LabVIEW on TCP/IP port 3363. To do this, open LabVIEW and go to 'Tools>Options… >VI Server: Configuration' and activate TCP/IP protocol on port 3363.

<table>
<tr><td>1</td><td>External VIs</td><td>Specify the path to the VI (you can also click on the browse folder button or you can drag a file from windows explorer).
The VI must have exactly this connector:

where Array is a 2D-array of strings (dimension n x 2), the first column containing key names and the second column containing the value.

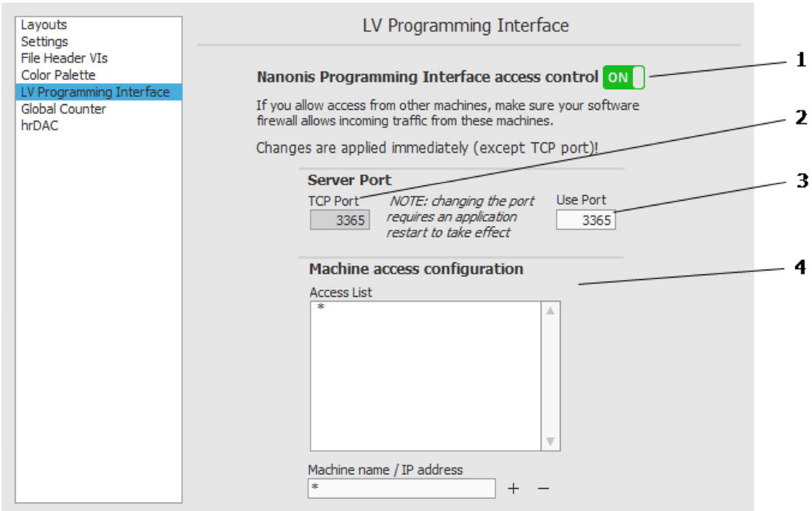
Example Array:<table><tr><td>Pressure (mbar)</td><td>1.437E-10</td></tr><tr><td>Temperature (K)</td><td>295.7</td></tr><tr><td>Sample</td><td>Si-111</td></tr><tr><td>…</td><td>…</td></tr></table>
An example VI can be found here.

If no path is specified you can still select "Ext. VI 1/2/3" in the list of available parameters, but no data will be added to the header of the saved file.</td></tr>
<tr><td>2</td><td>Test</td><td>Click on this button to test the specified external VI. If it works, you should see the returned parameters in the output table ("4"). If it doesn't work it should display an error dialog.</td></tr>
<tr><td>3</td><td>Clear</td><td>Deletes the path in the corresponding field.</td></tr>
<tr><td>4</td><td>Test output</td><td>This table displays the output of test calls when clicking one of the test buttons ("2").</td></tr>
</table>

IV) Color Palette

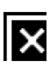
1	Palettes list	Display the list of color palettes. The palettes are used to colorize the data in several modules (Scan control, Scan monitors, Scan inspector…) The palettes files are located in the *AppData\Nanonis\Color Palettes* folder*. They are independent of the session folder and thus considered as global. Apply an action to the selected palette or to the list: create a new color palette. duplicate the selected color palette. delete the selected color palette. import a color palette from WSxM or SPIP make editable the selected color palette name. Note: it is possible to rename the palettes by clicking directly on the list item. *) *AppData* is located by default in these locations: - with XP: C:\Documents and Settings\All Users\Application Data\ - with Vista/7/8/10: C:\ProgramData\
2	Color ramp	Define the color ramp of the color palette. You can add more markers by right-clicking the ramp and selecting **Add Markers**. You can delete and select the color of the markers by right-clicking on them. The top and bottom color boxes correspond to the under and over range data.
3	Items colors	Define the color of items that appear in the visualization panel of the Scan modules. The **Save Palette** button saves the changes of the selected color palette.

V) Programming Interface



1	Enable access	Use this switch to enable/disable access using the Nanonis Programming Interface.
2	TCP port	This is the TCP port the application currently listens on (if access is enabled).
3	Use port	The TCP port can be changed here, but it requires an application restart to take effect. You will see that if you change the Port, the current TCP Port (“2”) doesn’t update because it still listens to the previous port (until application restarts).
4	Machine access	Control which machines are allowed to access the Nanonis application through the Programming Interface. - **Access List**: list of IP addresses, address ranges or host names that are allowed to access the application. If you don’t need to restrict access you can use the * wildcard. If you want to restrict access, enter the IP addresses or host names of the machines you want to grant access. - **Machine name / IP address**: to add a machine to the **Access list**, enter the name here and click on the + button. You can enter an IP range by using wildcards, e.g. 192.168.1.* allows access from that subnet. To remove an entry from the **Access list**, select an item and then click on the − button.

VI) Global Counter



The Global Counter can be used to automatically write an index in the saved data filename. To use it you only have to write a %N placeholder in the basename controls.
Every time a file is saved with the %N placeholder, the global counter value is incremented.
This option is useful if you want to keep track of the order in which you acquired measurements.

Example:
Let’s say that we are alternatively acquiring STM images and performing I(V) spectroscopy on some molecules.
In Scan Control, we configure the basename to “MySample_%N”.
In the Bias spectroscopy module, we configure the basename to “MySample I(V)_%N”.
The saved filenames might then look like:
MySample_00001.sxm
MySample I(V)_00002.dat
MySample_00003.sxm
MySample I(V)_00004.dat
MySample I(V)_00005.dat
MySample I(V)_00006.dat

MySample_00007.sxm
MySample_00008.sxm
…

VII) hrDAC

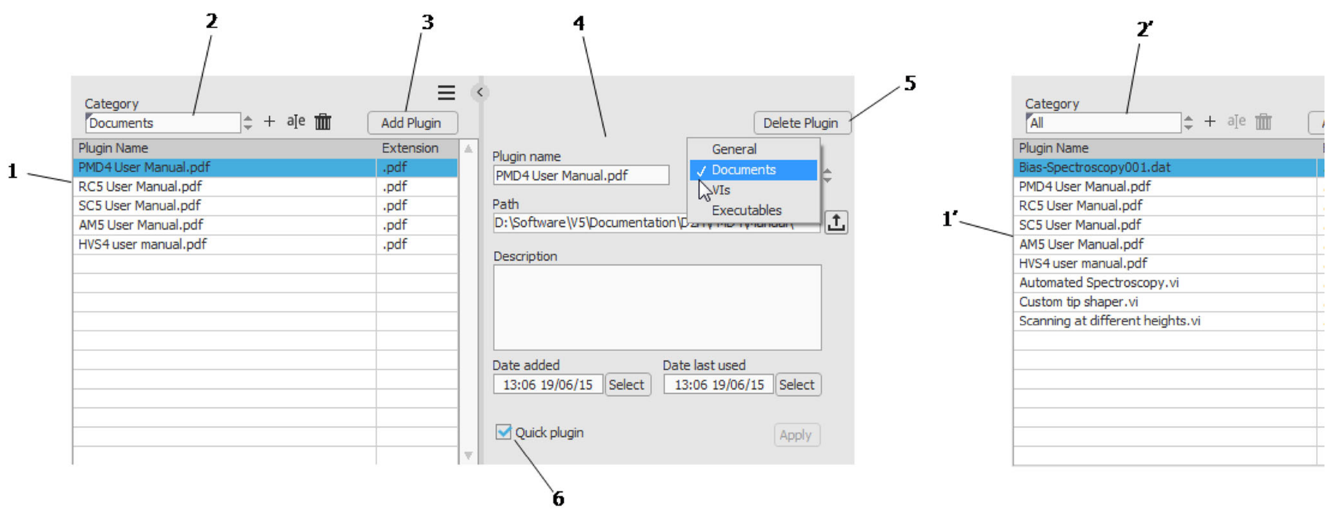
If hrDAC (high resolution digital-analog conversion) is available for your system, you can enable and disable it from this page.
This is only available after the hrDAC tuning module has calibrated the DACs. To do so, start the software in hrDAC tuning mode (if hrDAC is available this is an option at the license selection dialog on software start).

Back Home

Plugins

As part of the System menu, the Plugins item offers the possibility to integrate external files within the Nanonis software.
This can be very convenient e.g. when creating routines outside of the Nanonis environment (like automated procedures or custom experiments).

The external files can be any kind of files like documents, executables, LabVIEW VIs, etc. The file type determines which application takes care of opening the file when selecting it from the Nanonis software.
A plugin can be spontaneously opened through System>Plugins>Open plugin and/or it can be stored and opened through the Plugins Manager.
The Plugins Manager module allows the management, organization, and documentation of all the plugins.

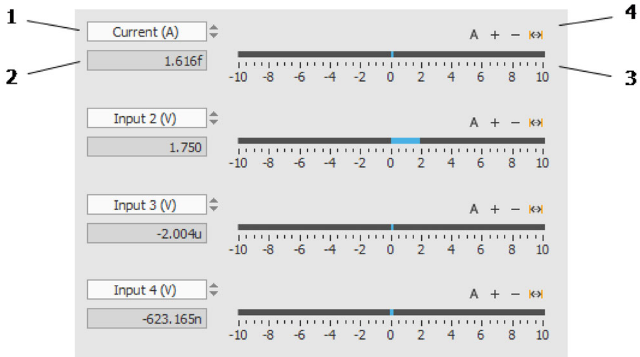


1, 1'	List of plugins	Display of plugins assigned to the selected **Category**.
2, 2'	Category	The selected category defines the plugins displayed right below. All and General categories exist always. The user can create more categories as needed. When a plugin is created and no specific category is used, it is categorized as General by default. When All is selected, all plugins (all categories) are displayed in the list. · Create a new category. · Delete selected category. · Edits selected category name. This can also be performed by right-clicking the Category name and selecting **Edit Category Name**.
3	Add plugin	Add a new plugin to the Plugins Manager. A file dialog window immediately opens to select the file (plugin).
4	Plugin information	Information related to the selected plugin. The user can set a name, assign the plugin to an existing category, write a description, and even set the plugin as a **Quick Plugin** (6).
5	Delete plugin	Remove the selected plugin from the Plugins Manager.
6	Quick plugin	When setting this flag, the selected plugin will appear as an item within the System>Plugins menu, so that the plugin can be quickly opened from the Nanonis Software menu without opening the Plugins Manager.
7	Expand/Retract button	Show/hide the **Plugin information** section.

Back Home

Signal Monitor

The Signal Monitor can be used to observe up to four different signals simultaneously.



1	Signal	Select the signal you want to observe.
2	Numerical indicator	The numeric indicator shows the current value of the signal. Tip: the update speed can be adjusted from the Indicators period in TCP receiver.
3	Graphical animation	Graphical indication of the signal. Tip: the update speed can be adjusted from the Animation period in TCP receiver.
3	QuickScale	Adjust the scale with those buttons.

Back Home

Signal Chart

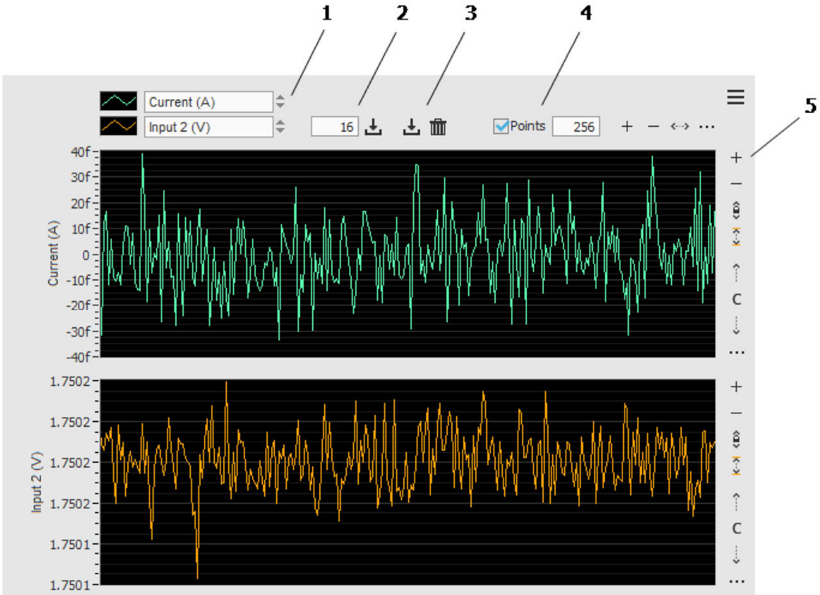
The Signal Chart displays two signal channels. An additional averaging can be applied to the data received from the FPGA card.

1	Signals	Select the signals you want to display in the charts.
2	Averaging	Set how many data samples (received from the FPGA card) are averaged for one data point displayed in the chart. By increasing this value, the signal to noise ratio decreases (as well as the charts rolling speed).
3	Zoom and Pan	Generic graph handling like Zoom and Pan and right-click menu options are explained in the user interface section.

Back Home

History

The History graph is similar to the Signal Chart except that it holds the data over a longer period of time. Moreover, the data points of all the 24 channels can be saved to file. The History graph operates with a circular buffer of 25000 points per channel. On every refresh cycle, all the 24 channels are recorded simultaneously into the buffer. Once the buffer is full the oldest data are overwritten by the new ones.
The buffer refresh rate is defined by the Animations period from the TCP receiver window.



1	Signals	Select the signals you want to display in the two graphs.
2	Autosave	Set the maximum number of files to save in series and activate the autosave button to automatically save the specified number of traces to ASCII files. A new file will be saved each time the buffer of 25000 points is full.
3	Save, Clear	Save the data buffer of all 24 channels to an ASCII file (.dat format). Clear the data buffer.
4	Data **Points**	Define the number of points to be shown. The maximal amount to be shown is 25000 points, corresponding to the size of the circular buffer of the History graph.
5	Zoom and Pan	Generic graph handling like Zoom and Pan and right-click menu options are explained in the user interface section.

Back Home

Long Term Chart

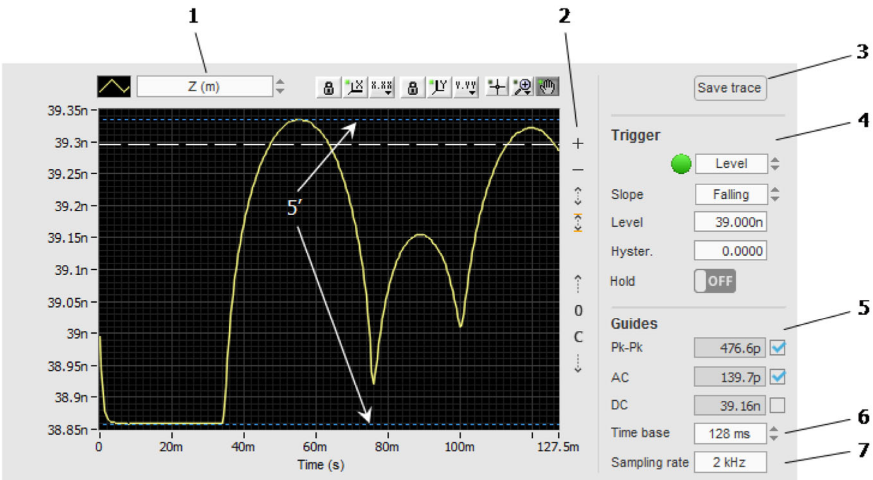
The Long Term Chart stores and displays the select signal over a long period of time.
The chart operates with a circular buffer of 10000 data points. Once the buffer is full the oldest data are overwritten by the new ones.
This chart is very useful to measure the creep of the scan piezo or record the thermal drift of the system over days.

1	Signal	Select the signal channel you want to record.
2	Zoom and Pan	Generic graph handling like Zoom and Pan and right-click menu options are explained in the user interface section.
3	Delay	Time delay between two consecutive data points. It can be set from 0.5 to 100 s.
4	Time remaining	Time remaining before the buffer is full. Once the buffer is full it starts rolling over (the oldest data are overwritten).
5	Save, Clear	- Save the data buffer to an ASCII file (.dat format). - Clear the data buffer.

Back Home

Oscilloscope

The Oscilloscope replaces an external oscilloscope for the inspection of the signals available in the Nanonis software.
You can monitor signals with exact timing information. The trigger lets you steadily display periodic signals.
Note that, depending on the **Time base**, the signal is averaged and may not show high frequencies.

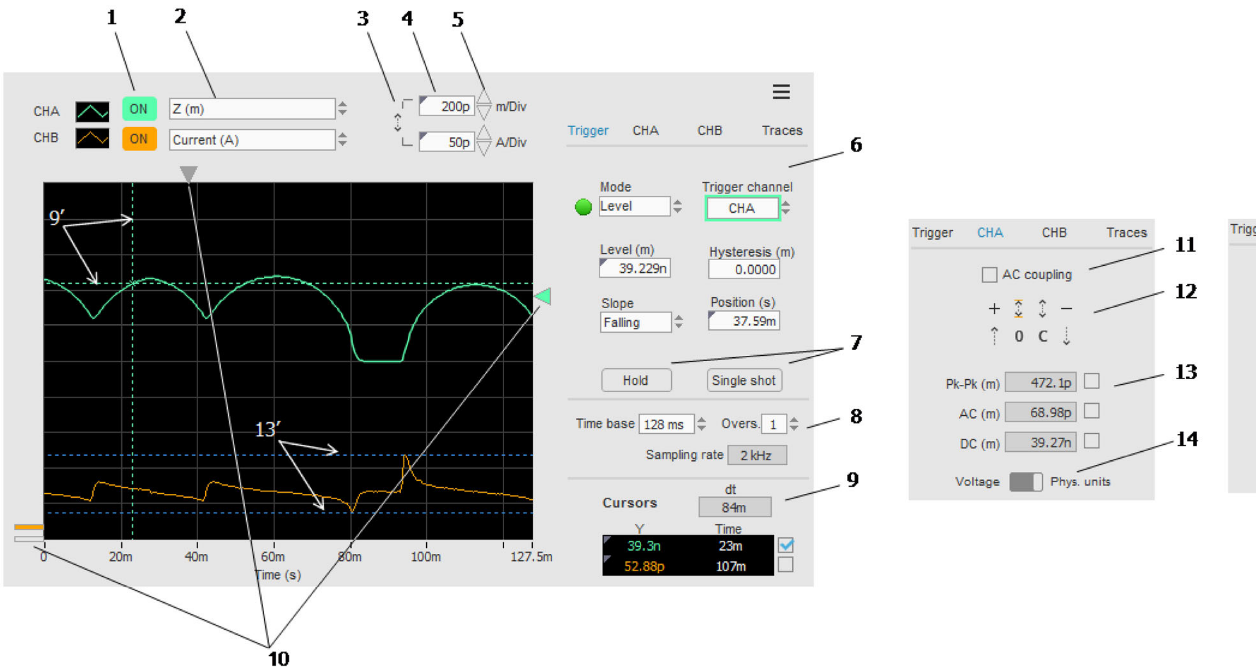


1	Signal	Select the signal channel you want to display.
2	QuickScale	Use those buttons to adjust the vertical **scale** and **position** of the trace.
3	Save trace	Save the current trace to an ASCII file (.dat)
4	Trigger	Configure the oscilloscope trigger: **Green LED**: when the trigger occurs, the LED lights up. When the oscilloscope is waiting for the next trigger event, the LED is off. **Mode**: - "Auto": the trigger threshold is automatically found. - "Level": the trigger threshold is user-defined in the **Level** field. - "Immediate": the trigger is off. **Slope**: triggering direction when the signal crosses the **Level**. **Level**, **hysteresis**: trigger level and hysteresis. **Hold**: freeze the oscilloscope so the current trace remains on the graph.
5,5'	Guides	Display some useful information related to the current signal trace: **Peak-Peak**: amplitude between the maximum and the minimum value of the signal displayed. **AC (rms)**: estimated Root Mean Square value of the AC component around the DC signal component. **DC**: estimated mean value of the signal displayed. The checkboxes are used to display the corresponding guide line (5') in the graph. Note: **AC** and **DC** values are estimated and their accuracy relies on the appropriate choice of the **Time base** so that the signal's bandwidth is much lower than the **Sampling rate**.
6	Time base	Select the oscilloscope's time base. Each graph's data point is an average over an integer number of samples, N, acquired at the **Signals period** rate (defined in the TCP receiver window). Since the number of data points displayed on the graph is fixed to 256, the **Time base** sets the value of N. For instance, if **Signals period**=1 ms and **Time base**=1.28 s then N=5. Note: by decreasing the **Signals oversampling** in the **TCP receiver** window, you can access lower **Time base** values that can be required for higher bandwidth signals. Note: whereas the Oscilloscope always displays 256 points (averaging the data over an integer number of samples given by the **Time base**), the Two Channel Oscilloscope adjusts the number of displayed points acquired at the **Signals period** rate when changing its **Time base**.
7	Sampling rate	Actual sampling rate of the signal displayed in the graph. It is equal to **256/Time base**. Note: for good results, this value must be at least 4 times higher than the signal's bandwidth. Otherwise, aliasing will appear and the signal won't be displayed properly.

Back Home

Two Channel Oscilloscope

The Two Channel Oscilloscope replaces an external oscilloscope with two channels for the inspection of the signals available in the Nanonis software.
You can simultaneously monitor signals with exact timing information. The trigger on any of the 2 channels lets you steadily display periodic signals. The pre-trigger capabilities offer valuable information over the acquired data.



1	Display Channel buttons	Show/Hide on the graph the signals selected in channel A and channel B. The color of the button shows the color of the corresponding signal in the graph. These colors and other display settings can be changed clicking on the small icon on the left of the **Display Channel** buttons.
2	Signals	Select the signals you want to have in channel A and channel B.
3	Autoscale	Autoscale both signals to fit in the graph. Signal on channel A will be in the top half and signal on channel B will be in the bottom half of the graph.
4	Units per division	Show the scale of the Y axis for both channels. The Y scale is divided into 10 divisions.
5	Increase/Decrease	Increase/Decrease the units per division of the Y axis, rescaling the corresponding signal.
6	Trigger configuration	Configure the Two Channel Oscilloscope trigger: **Green LED**: when the trigger occurs, the LED lights up. When the oscilloscope is waiting for the next trigger event, the LED is off. **Mode**: - "Auto": the trigger threshold is automatically found. - "Level": the trigger threshold is user-defined in the Level field. - "Immediate": the trigger is off. **Trigger channel**: channel used to check the trigger. **Level**, **hysteresis**: trigger level and hysteresis. The Level has a right click menu (small grey triangle) to bring the trigger level to the last DC value. When changing the trigger level, the trigger level indicator (see Objects on graph) changes accordingly. **Position**: trigger position in the X axis. The Position has a right click menu (small grey triangle) to bring it to zero, or to the center of the X axis (thus having 50% of pre-trigger samples). When changing the trigger position, the trigger position indicator (see Objects on graph) changes accordingly. **Slope**: triggering direction when the signal crosses the Level.
7	Freeze buttons	**Hold**: Freeze/release the traces currently displayed. **Single shot**: Acquires a single trace and then it holds it on the graph until using the Hold button to release it or until single shooting again.
8	Time base	Select the time base of the Two Channel Oscilloscope. Unlike the normal Oscilloscope where the number of points is fixed to 256, here the Time base sets the number of points to 256, 512, 1280, 2560, 5120, and 12800. They are acquired at the Signals period rate (defined in the TCP receiver window), and each graph's data point is averaged over the integer number of samples defined by **Oversampling**. The resulting **Sampling rate** is displayed below the **Oversampling** control. Hovering the mouse over the **Sampling rate** indicator displays a tip strip showing the final oversampling in the Two Channel Oscilloscope data: the oversampling set to all the signals in the TCP Receiver plus the oversampling set in the Two Channel Oscilloscope. Note: by decreasing the Signals oversampling in the TCP receiver window, you can access lower Time base values that can be required for higher bandwidth signals. Note: for good results, the **Sampling rate** must be at least 4 times higher than the signal's bandwidth. Otherwise, aliasing will appear and the signal won't be displayed properly.
9, 9'	Cursors	Display the X (time),Y value of the cursors. The cursors can be individually shown or hidden with the checkboxes on the right hand side of the cursors section. If both channels are displayed in the graph, the cursors can be individually assigned to any channel just using the right-click menu (a small

		grey triangle shows where a right-click menu is available). **dt** shows the time difference between both cursors.
10	Objects on graph	**Zero reference**: 2 small bars on the Y axis show where the zero value for both channels is on the graph, and they can be moved along the Y axis dragging the signal along. They have the color of their corresponding channel when the Zero value is inside the visible range of the Y axis, else they don't have color. The zero value for each channel can be brought to the graph (meaning that the units/div doesn't change, but the signal moves along the Y axis until the zero value falls inside the graph) with the corresponding QuickScale Zero button in the CHA/CHB tabs. When the signal is not inside the visible range of the graph, a small arrow with the color of the corresponding graph appears at the top or bottom of the axis, giving a hint of where the signal is. **Trigger position**: A small round shaped triangle on top of the graph shows the trigger position in the X axis. When dragging it along, it changes the trigger position indicator in the Trigger Configuration tab. A tip strip while dragging displays the position in percentage of the X axis. **Trigger level**: A small round shaped triangle on the right side of the graph shows the trigger level. Its color corresponds to the color of the trigger channel. It can be dragged up and down to change the trigger level. It only appears if the **Trigger Mode** is set to Level in the Trigger Configuration tab and if the trigger level is inside the visible range of the Y axis. When the trigger level is not inside the visible range of the Y axis, a small arrow with the corresponding channel color appears at the top or bottom corners of the Y axis.
11	AC coupling	When checked, the DC estimated value is subtracted from the signal at the selected channel.
12	QuickScale	Use these buttons to adjust the vertical **scale** and **position** on the selected channel.
13, 13'		Display some useful information related to the current signal trace: **Peak-Peak**: amplitude between the maximum and the minimum value of the signal displayed. **AC (rms)*:*** estimated Root Mean Square value of the AC component around the DC signal component. **DC**: estimated mean value of the signal displayed. The checkboxes are used to display the corresponding guide line (13') in the graph. Note: AC and DC values are estimated and their accuracy relies on the appropriate choice of the Time base so that the signal's bandwidth is much lower than the Sampling rate.
14	Units switch	When switching to **Voltage**, it displays the data of the selected channel in volts (i.e. without calibration).
15	Traces buttons	**Load**: Load traces from an ASCII file (.dat) and paste them into the background of the graph. **Save**: Save the current traces (both channels) to an ASCII file (.dat) **Paste CHA**: Paste a snapshot of the current trace on channel A to the background of the graph. **Paste CHB**: Paste a snapshot of the current trace on channel B to the background of the graph. **Clear CHA**: Clears the channel A pasted snapshot from the background. **Clear CHB**: Clears the channel B pasted snapshot from the background.

Back Home

High Resolution Oscilloscope

The High Resolution Oscilloscope includes an oscilloscope and a spectrum analyzer that replace external devices to perform time and frequency-domain analysis.
The frequency spectrum is displayed as the square root of the Power Spectral Density (PSD).
The signals acquired and monitored are those present on the FPGA target of the Nanonis controller. When recording signals which are modulated within the system, the unmodulated signal is recorded.
*Signal simulation options in simulation mode (Demo mode) are listed under IV) Simulation mode Settings.

I) Oscilloscope

1	Plot Color	Set the color for the main plot on the oscilloscope display
2	Calibration mode	Select between “Raw Values” (Voltage IO to / from SC5) or “Calibrated Values” as defined in the Nanonis software. This setting affects the data shown in the curve and trigger level and hysteresis values.
3	Signal	Select the signal channel you want to display. Channels available for acquisition include all 8 inputs and outputs of the attached SC5 device.
4	Oversampling	Choosing to acquire data at a lower rate than he maximum 1MS/s allows for an improved S/N ratio and also increases the time window for the acquisition for a given number of samples. Both the total time window for acquisition and the associated sampling frequency are shown for convenience.
5	Nr. Samples	Select how many data points are to be acquired. This number is coerced to a multiple of 32 due to internal requirements.
6	Autosave	Set the maximum number of files to save in series and activate the autosave button to save all data received to ASCII files
7	Background	save the curve to an ASCII file (.dat format). paste the present curve to the graph's background. load a saved curve to the graph's background (reference curve). delete the background curve.
8	Zoom, Pan and Drop-Down menu	Use those buttons to zoom and pan the trace. The drop-down menu prevents further Oscilloscope display options. Note: Generic graph handling is explained in the user interface section.
9	Trigger/Measure	Switches between trigger setup and cursor measure modes (See below)
10	Pre-Trigger	Position of recorded trigger based on current pre-trigger setting. Can also be used to change the pre-trigger setting
11	PSD button	Show/hide the lower panel that contains the spectrum analyzer.
12	Zoom active?	Shows whether the X-scale zoom functionality detailed in 16 below is active or not
13	X-Scale zoom	Allows for fluid zooming on the x-axis of the oscilloscope data for users wanting to display only a small portion of the acquired data

II) Oscilloscope – Trigger tab

1	Immediate Trigger	The trigger is always active (no synchronization) The oscilloscope is trigger immediately whenever the current data set is received by the host software. There is no synchronization with any real-

		world signal.
2	Level Trigger	An analog channel must be chosen on which to synchronize the trigger. This channel is not required to be the same channel as the acquisition channel. Settings are as detailed below. **Source :** select the signal (8x input) on which the trigger applies. Triggering is performed on the non-averaged raw channel data (1MS/s). **Slope** : select the triggering direction (Rising edge or Falling edge) **Trig Level** : define the trigger level. In "Raw" mode the unit of this setting is Volts. In "Calibrated" mode the units of the trigger level are according to the currently active calibration of the trigger channel. **Hysteresis** : Hysteresis allows for more reliable triggering on noisy signals. In "Calibrated" mode the units of the trigger level are according to the currently active calibration of the trigger channel. **Trig Time** : Setting / display of the current time window (in seconds) for samples recorded prior to the current trigger.
3	Digital Trigger	A digital channel must be chosen on which to synchronize the trigger. Settings are as detailed below **Source***:* select the signal (32 LS-DIO and 4 HS-DIO) on which the trigger applies. Trigger detection on the LS-DIO channels is performed at 500kS/s. Trigger detection on the HS-DIO channels is performed at 10MS/s **Slope**: select the triggering direction (Rising edge or Falling edge) **Trig Time**: Setting / display of the current time window (in seconds) for samples recorded prior to the current trigger.
4	Arming	Choice of operation mode: **Single Shot** : Records the next available data set and then stops acquisition **Continuous** : Records every available data set and automatically re-triggers acquisition
5	Re-arm	For "Single Shot" mode, pressing this button will re-arm the oscilloscope to record the next available data set
6	Savename	This is the base name used for saving oscilloscope data (manual and autosave) i.e. OSCI_Scope_001.dat results from a savename "OSCI_Scope"

II) Oscilloscope – Measure tab

1	Cursors	The cursors are used to measure time difference and its equivalent frequency. **Delta T** : time difference between the two cursors. Freq. = 1/Delta T Tip: to bring the cursors to the visible portion of the graph, right-click on the graph and select "Cursors>Bring to Center".
2	Data	Display some useful information related to the current signal trace: **Peak-Peak** : amplitude between the maximum and the minimum value of the signal displayed. **AC (rms)** : estimated Root Mean Square value of the AC component around the DC signal component. **DC** : estimated mean value of the signal displayed. Note: *AC* and *DC* values are estimated and their accuracy relies on the appropriate choice of the **Time base** so that the signal's bandwidth is much lower than the Sampling rate.
3	Additional Cursor indicator	Displays the measurement values associated with the selected cursor types. Click on the drop-down menu to expose more cursor options. Note: Generic graph handling including cursor handling is explained in the user interface section.

III) Spectrum panel

1	Autosave	Set the maximum number of files to save in series and activate the autosave button to save all data received to ASCII files
2	Background	• save the curve to an ASCII file (.dat format). • paste the present curve to the graph's background. • load a saved curve to the graph's background (reference curve). • delete the background curve.
3	Zoom, Pan and Drop-Down menu	Use those buttons to *zoom* and *pan* the trace. The drop-down menu prevents further Oscilloscope display options. Note: Generic graph handling is explained in the user interface section.
4	FFT Window	Select the window function applied to the timed signal before calculating the Power Spectral Density. Possibilities are "No windowing", "Hanning", "Hamming", "Blackman-Harris", "Exact Blackman", "Blackman", "Flat Top", "4 Term B-Harris", "7 Term B-Harris" and "Low Sidelobe".
5	FFT Averaging Mode	"No averaging", "Vector averaging", "RMS averaging" or "Peak hold".
6	FFT Averaging Weighting	"Exponential" or "Linear".
7	FFT Averaging Count	Specifies the number of averages used for "RMS" and "Vector" averaging. If **weighting** mode is "exponential", the averaging process is continuous and new spectral data have a higher weighting than older ones. If **weighting** mode is "linear", the averaging combines **count** spectral records with equal weighting and then stops.
8	FFT Averaging Restart	Restart the averaging process.
9	Cursor control	Display the cursors coordinates (PSD *vs.* Frequency). The cursors can be hidden/shown by right-clicking on the graph and selecting *Cursor>Show*. The second cursor is only shown when the averaging type "RMS" is selected. The color indicates on which curve the cursor is snapped to. Note: Generic graph handling including cursor handling is explained in the user interface section. Tips: the cursor can be moved from one curve to the other by dragging its crosshair, if *Cursor>Stick to one curve* is deselected in the right-click menu. It is also possible to make it jump to the other curve by right-clicking on the graph and selecting *Cursor>Jump to Other Curve.* If you drag the cursor using its vertical line, it won't jump to the other curve.
10	Spectrum curve	Spectrum of the selected signal.

IV) Simulation mode Settings

1	Oscilloscope Simulation Controls	The controls of interest for the Oscilloscope FPGA are shown in the image above (image shows part of the panel of the program "Sim-Engine.exe"). These signals defined for demonstration purposes affect the Oscilloscope FPGA module only. Changes made here do not affect the rest of the system and vice versa. **Signal Type** : Chooses between simulated DC, sine wave, square wave or triangle wave signals **Amplitude (V)** : Amplitude (half peak-peak) of the signal to simulate. **Frequency** : Frequency of the signal to simulate **Offset (V)** : Offset of the simulated signal **Noise Type** : Choose between uniform and gaussian distributed noise

		Noise (V) : Amplitude of noise added to the simulated signal

Back Home

Spectrum Analyzer

The Spectrum Analyzer replaces an external device to perform frequency analysis of the signals available on the Nanonis SPM Control system.
The frequency spectrum is displayed as the square root of the Power Spectral Density (PSD).

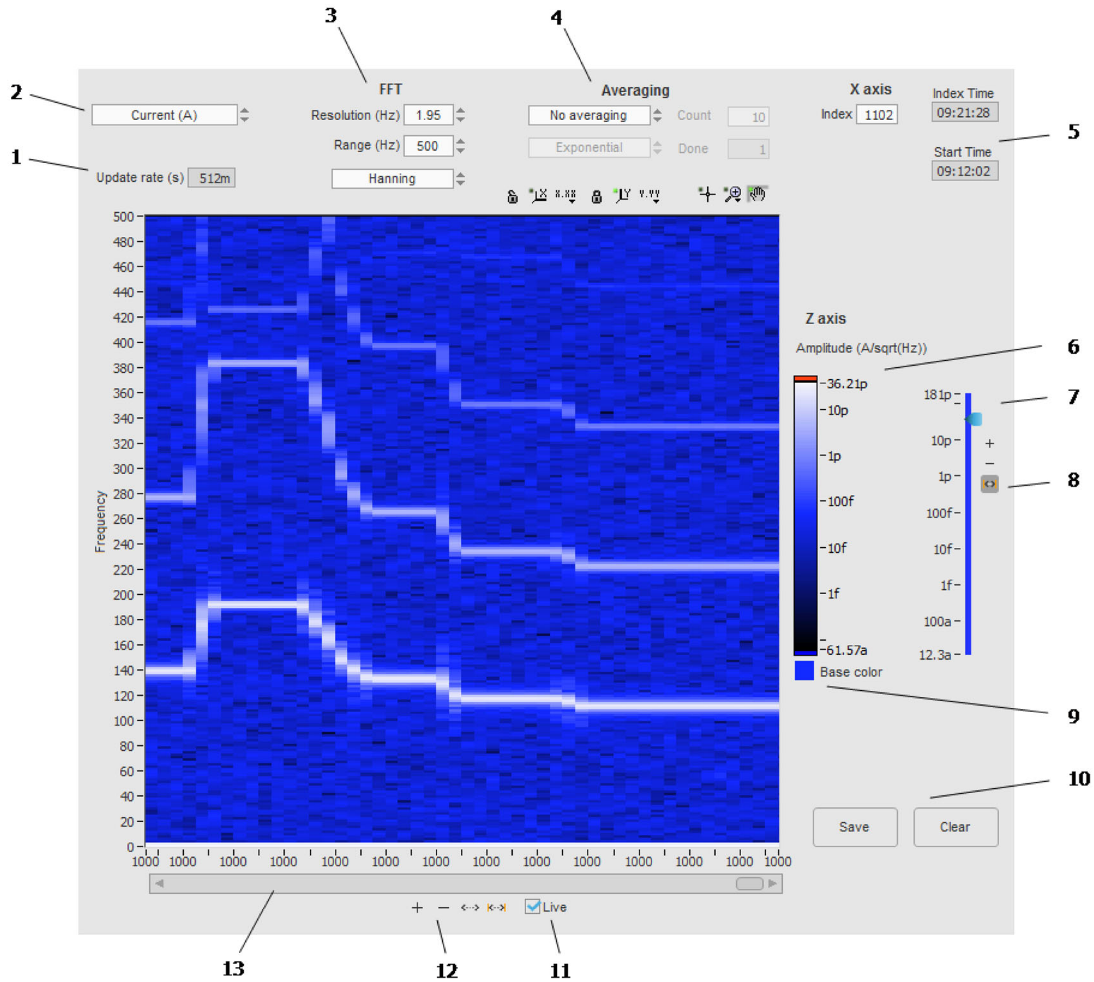
1	Signal	Select the signal channel to analyze
2	Spectrum curve	Spectrum of the selected signal
3	Reference curve	Spectrum of the loaded or pasted curve in the background. This curve can be used as a reference to compare the present spectrum to a previously acquired one. This is useful to compare noise levels and peaks amplitude. Tip: the color of this curve can be changed by clicking on the small box next to the curve legend (top left corner of the graph).
4	Save and Background functions	- Save: save the spectrum signal curve to an ASCII file (.dat format). - Load: load a saved curve to the graph's background (reference curve). - Paste: paste the present spectrum signal curve to the graph's background. - Clear: delete the background curve.
5	Zoom and Pan	Use those buttons to zoom and pan the trace. The drop-down menu prevents further display options. Note: Generic graph handling is explained in the user interface section.
6	**FFT** functions	**Range**: Specifies the frequency range (i.e. the x-axis span, 0 Hz being the minimum). The frequency range depends on the Signals period defined in the TCP receiver window. To access higher frequency range you can decrease the Signals oversampling in the TCP_receiver. Note: smaller frequency range takes more time to update the spectrum. **Resolution:** Specifies the frequency resolution. Note: higher resolution requires more points to be acquired and thus takes more time to update the spectrum. **Window** type: Window function applied to the timed signal before calculating the Power Spectral Density. Possibilities are "No windowing", "Hanning", "Hamming", "Blackman-Harris", "Exact Blackman", "Blackman", "Flat Top", "4 Term B-Harris", "7 Term B-Harris" and "Low Sidelobe".
7	**Averaging**	**- Type**: "No averaging", "Vector averaging", "RMS " or "Peak hold". **- Weighting**: "Exponential" or "Linear". **- Count**: specifies the number of averages used for "RMS" and "Vector" averaging. If weighting mode is "exponential", the averaging process is continuous and new spectral data have a higher weighting than older ones. If weighting mode is "linear", the averaging combines count spectral records with equal weighting and then stops. **- Restart** the averaging process.
8	**Cursors**	Display the cursor coordinates. The cursors can be hidden/shown by right-clicking on the graph and selecting Cursor>Show. The color indicates on which curve the cursor is snapped to. Tips: the cursor can be moved from one curve to the other by dragging its crosshair, if Cursor>Stick to one curve is deselected in the right-click menu. It is also possible to make it jump to the other curve by right-clicking on the graph and selecting Cursor>Jump to Other Curve. If you drag the cursor using its vertical line, it won't jump to the other curve. Band RMS shows the rms value in the frequency band defined between both cursors. Note: Generic graph handling including cursor handling is explained in the user interface section.

Back Home

Long Term Spectrum Analyzer

Acquires frequency spectra of the selected channel and display them as function of time. This can be use to see whether some disturbances appear or disappear over time.

The Long Term Spectrum has a circular buffer that can hold 5000 spectra. Each spectrum is measured like in the Spectrum Analyzer module.
The Power Spectral Density (PSD) data are displayed in an intensity chart:
- **X Axis** is the spectrum number (i.e. the time).
- Y Axis is the frequency.
- **Z Axis** (intensity) is the PSD amplitude of the selected signal at the corresponding time & frequency.



1	**Update rate**	Indicates the time it takes to measure one spectrum.
2	Signal	Select the signal channel to analyze.
3	**FFT**	- **Resolution**: Specifies the frequency resolution. Note that higher resolution requires more points to be acquired and thus takes more time to update the spectrum. - **Range**: Specifies the frequency range (i.e. the y-axis span, 0 Hz being the minimum). The frequency range depends on the Signal Period defined in the TCP receiver window. To access higher frequency range you can decrease the Signals Oversampling in the TCP receiver. Note that smaller frequency range takes more time to update the spectrum. - Window type: Window function applied to the timed signal before calculating the Power Spectral Density.
4	**Averaging**	- Averaging mode: "No averaging", "Vector averaging", "RMS averaging" or "Peak hold". - Weighting mode: "Exponential" or "Linear". - **Count**: specifies the number of averages used for "RMS" and "vector averaging". - **Done**: displays how many spectra have been averaged. If weighting mode is exponential, new spectral data have a higher weighting than older ones. If weighting mode is linear, the averaging combines count spectral records with equal weighting.
5	History	- **Index**: select a particular spectrum - **Time** stamp: time at which the spectrum was acquired
6	Color ramp	Show the correspondence between the color and the PSD amplitude.
7	Color range	Define the range limits of the color ramp.
8	Quick scale	Change the color range scale.
9	Color box	Define the color of the intensity chart.
10	Save, Clear	- **Save**: save the spectrum data to file. - **Clear**: clear the graph.

11	**Live** mode	Toggles the chart to automatically display new spectra or not.
12	Quick scale	Change the displayed samples range scale.
13	Displayed samples range	Defines the portion of the buffer to display in the graph.

Back Home

Signal to Sound

The "Signal to Sound" module converts the DC value of any signal to a sound and puts it on the computer's speaker or headphones. You can define a signal range on which to map a frequency range. This module isn't intended to detect fast oscillations (noise) of a signal, but to reflect changes in the DC value (averaged / low-pass-filtered) of a signal.

Example application for SPM users: You can convert the Z signal to a sound while using the Follow Me mode in the Scan Control module. If the signal range is set up appropriately, you will hear when moving over objects or step edges.

1	Switch sound on/off	Switch sound output on or off
2	Channel to convert	Select the channel to convert to sound
3	Frequency mapping	Switch between linear of exponential frequency mapping. With linear mapping, the signal range ("4") is mapped linear to the frequency range ("5"). With exponential mapping, the frequency increases only slowly in the lower part of the signal range and faster in the upper part.
4	Signal range & value	The gray slider shows the current value of the signal selected in "2". The displayed range is the signal range that is mapped on the specified frequency range ("5"). You can zoom in to the interesting range of the signal so the full frequency range is mapped on the interesting signal range. To change the "range of interest", use the QuickScale buttons or edit the values at the lower/upper limit manually (click the limit and edit it).
5	Frequency range	Lower and upper frequency limit, together they define the frequency range the signal range ("4") will be mapped on.
6	Current value	Current value of the signal selected as Channel to convert ("2").
7	Frequency	Displays the current frequency that corresponds to the current signal value ("6").
8	Volume	Use this slider to adjust the volume of the sound output.

Back Home

Bias

The Bias module is used to adjust the bias voltage applied between tip and sample.
It also offers a pulse generator to pulse the bias voltage for a specified amount of time. During the pulse, the Z-Controller can be set on hold.

1	Bias	Applied bias voltage. To modify the bias you can either enter a value in the digital display or move the slider.
2	Expand/Collapse	Shows/hides the bias pulse and calibration sections.
3	Bias pulse	Configure and trigger bias pulses. This can be useful to prepare the scan tip. - **Voltage**: define the pulse height. - **Abs/rel switch**: select whether the Voltage value is absolute or relative to the current applied bias. - **Width**: define the pulse duration. - **Z-Ctrl hold checkmark**: when selected, the Z-Controller is set to hold during the pulse. This means that the controller doesn't control the Z position during the pulse. - **Pulse** button: trigger one pulse.

4	Range & Calibration	- **Range**: Switch the bias range. Note: the range selector is available only if your system supports bias range switching. - **Calibration**: Calibration of the Bias output. If you have a **Range** switch the calibration is stored per range setting. - **Offset**: Allows compensating for an offset in Bias.
5	QuickScale	Use those buttons to adjust the scale.

Tools

·	Bias Sweep	Opens the Bias Sweep module.
·	Bias Spectroscopy	Opens the Bias Spectroscopy module to perform fast bias sweeps with deterministic timings.
·	Contact Potential Difference	Opens the Contact Potential Difference (CPD) compensation module.

Note: Numeric control handling is explained in the user interface section.

Back Home

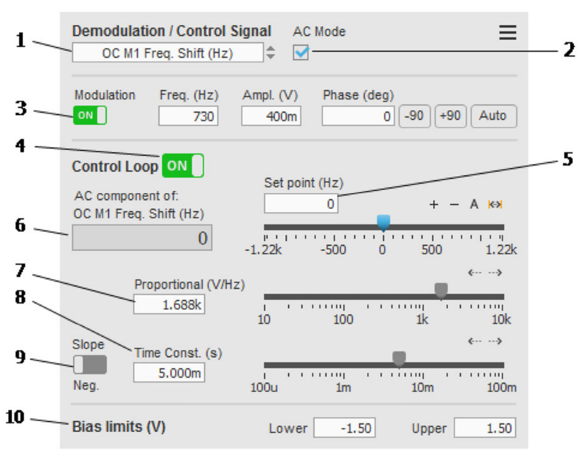
Kelvin Controller (optional module)

Use this module for Kelvin probe measurements.

It modulates the Bias voltage (upper section of the window) and demodulates the selected signal (frequency shift for Kelvin Probe measurements) with a lock-in detection and displays the AC component of the demodulated signal. This is the amplitude of the modulation of the frequency shift signal (or whatever signal you selected) induced by the bias modulation. The bias voltage is controlled (using the PI-controller parameters in the lower section of the window) to minimize this modulation of the frequency shift (AC component).

Alternatively you can also use the module as a generic PI controller. Set AC mode off, so bias is not modulated. Define a setpoint for the selected control signal and set up the PI parameters. The module will now try to keep the control signal at the setpoint by changing the bias voltage.

A separate Generic PI Controller module is available which will use a different output (instead of bias) to control the specified signal. This is even more generic and can be used as PI control module for any external device. For example, you could use the generic PI control module to control a temperature.



1	Demodulation signal / Control signal	In **AC mode** (see "2"), this is the signal which will be demodulated and the induced amplitude will be controlled. In DC mode (**AC mode** off), this is the control signal which the PI controller will try to keep at the **set point** ("8"). Select Frequency Shift for Kelvin Probe measurements.
2	AC mode	In AC mode, the bias voltage will be modulated according to the settings in the modulation section. The PI controller will try to keep the modulation amplitude of the **demodulated signal** ("1") at the **set point** specified in "8". In DC mode (AC mode off), bias isn't modulated and the module works as a normal PI controller to keep the control signal ("1") at the set point.
3	Bias modulation settings	- **Modulation**: Switch the bias modulation on or off. Switching it off while the controller is running also switches off the controller. - **Frequency**: Frequency of the bias modulation - **Amplitude**: Amplitude of the bias modulation - **Phase**: Reference phase of the demodulation (the AC component is calculated for this phase with respect to the modulation) - **-90** / **+90**: change the reference phase by +/- 90 degrees - **Auto**: Adjusts the phase to maximize the demodulated signal (AC signal, “6”)
4	Kelvin controller on/off	Switch the Kelvin controller on or off. In **AC mode**, switching it on while no modulation is running also switches on the modulation.
5	Set point	Set point for the PI controller. In **AC mode** ("2") the set point is the AC component (amplitude) of the demodulated signal. Set it to 0 Hz for Kelvin Probe measurements. In DC mode, this is the set point for the value of the control signal.
6	Demodulation amplitude / Control signal	In **AC mode**, this is the demodulated amplitude of the signal selected in "1" (for Kelvin Probe this is the AC component (at modulation frequency) of the frequency shift signal). In DC mode this shows the value of the control signal.
7	Proportional	Proportional gain of the Kelvin controller.
8	Time Constant	Time constant of the Kelvin controller. A good value is at least 10 times the bias modulation period (period = 1/frq).
9	Slope	Depending on the controller configuration you might need to invert the controller slope.
10	Bias Limits (V)	**Upper** and **lower** limits of Bias (DC value, the modulation can exceed the limits) when the Kelvin controller is on.

Setup for a Scanning Kelvin Probe:

1. Set the demodulation signal to Frequency shift, AC mode = on, set point = 0 Hz.
2. Set the bias voltage to a negative value significantly off the contact potential difference.
3. Set the bias modulation frequency to a value that is about the demodulation bandwidth of the PLL (note that the z-controller should be set slow enough not to follow the modulation of the Kelvin controller).
4. Set the bias modulation amplitude to a value in order to get a good signal.
5. Press the auto phase button. The phase is adjusted to maximize the AC component of the Frequency Shift. Note that the lock-in is a one channel lock-in. To adjust the phase it first measures the amplitude at the given phase, then turns the phase by 90 degrees, measures the amplitude again and computes now the vector and its angle. You also can manually sweep the reference phase and maximize the demodulated signal.
6. Set the Kelvin controller time constant to at least 10 times the oscillation period of the modulation
7. Switch the Kelvin controller on.
8. Watch the bias signal with the signal chart and adjust the proportional gain of the Kelvin controller to reach the desired noise level. Note that the speed of the Kelvin controller can become very slow when you want a low noise level.
9. Adjust the scan speed by comparing forward and backward scan.

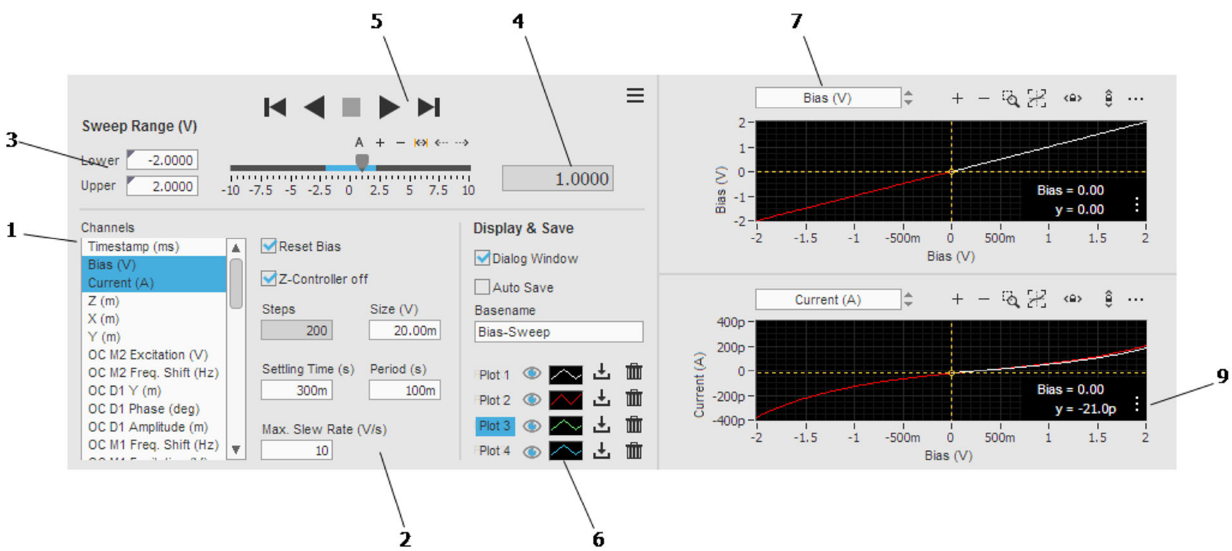
Back Home

Bias Sweep

The Bias Sweep is used to measure several channels while sweeping the bias voltage. The data recorded in a bias sweep are stored into one of four buffers which are displayed in the two graphs (Plot 1-4). You can save the buffers either manually or automatically after the sweep is done.

The sweep is controlled from the host computer (i.e. not from the real-time controller), therefore limiting the acquisition speed to about 50 pt/s.

Note: Generic graph handling and numeric control handling are explained in the user interface section.



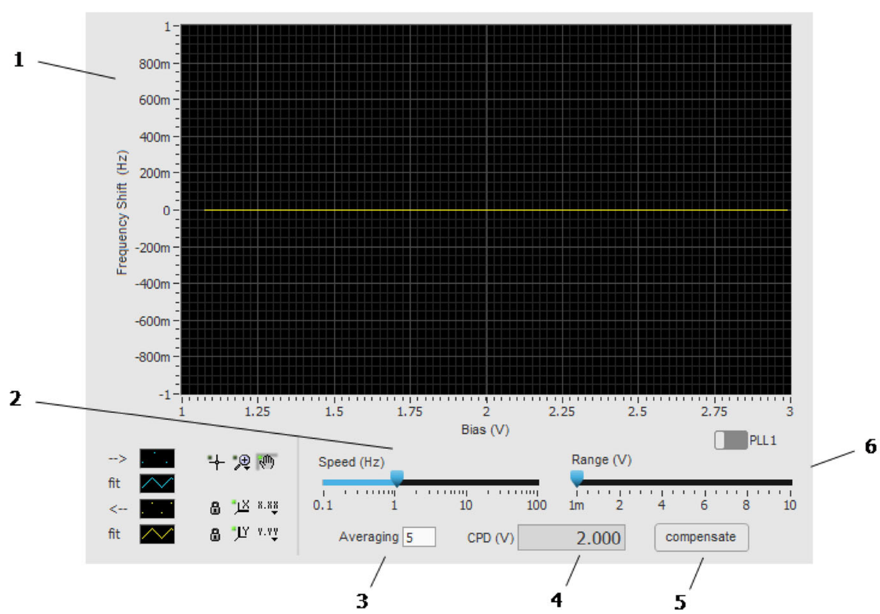
1	Channels	Select the channels you want to record. You can select more than one channel by pressing the Ctrl-key while selecting the channels with the mouse.
2	Acquisition parameters	**Reset Signal**: When selected, the signal is set to its initial value (*i.e.* value before starting the sweep) when the sweep is done. **Z-Controller off**: When selected, the Z-controller is switched off during the ramps and for the duration of the sweep. It is turned on afterward (if previously on). Before turning off the controller, the **Switch off Delay** and **TipLift** values (in the Z-controller module) are taken into account. Thus, to get a reproducible Z position, it is recommended to set a non-zero **Switch off Delay**. **Steps**: Number of measurement points to record **Size**: define the step size. **Settling Time (s)**: Time to wait just before starting the sweep. **Period (s)**: Defines the measurement speed. Higher values result in more accurate data because of the oversampling. This control is the same as the one in TCP receiver **Max. Slew Rate (X/s)**: Maximum rate at which the **sweep signal** changes when ramping to the starting point. If the **switch** is set to **Size** but the sweep range is such that the number of steps is not an integer, then the **Number** field will turn orange. This warns that the last step of the sweep will be smaller than the value specified by the user.
3	Limits	Define the Lower and Upper limits of the bias sweep. The Lower value must be smaller than the Upper one. Otherwise, the fields will blink and it won't be possible to start a sweep.
4	Bias	The digital control and the slider can be used to manually set a bias voltage. They are coupled with the controls on the Bias module. The yellow bar represents the sweep range as defined by the Limits.
5	Control	▶ Start the sweep, from **Lower** limit to **Upper** limit.

		Start the sweep, from **Upper** limit to **Lower** limit. When starting a sweep, if the signal value is different from the starting value, the signal will first ramp to that point at the rate defined by **Max. slew rate**. Stop the sweep. Ramp to the **Lower** or **Upper** limit at the rate defined by **Max. slew rate**.
6	Save options & Plot	**Dialog**: when selected, a dialog window will open before saving data. **Auto**: when selected, the data are automatically saved after each sweep. **Basename**: defines the file basename when saving sweeps. The file names will be "*Basename*NNN.dat" where NNN is an automatically increasing number. **Plot**: select to which plot (buffer) the next sweep will be saved to. : show/hide the plot curve on the graphs. : save the data to ASCII file (.dat). : clear the plot.
7	Signal to display	Select the signal you want to display in the graph.
8	Cursor	Display the cursor coordinates. The cursor can be hidden/shown by right-clicking on the graph and selecting Cursor>Show. The cursor can be moved from one curve to the other by dragging its crosshair. Note: Generic graph handling including cursor handling is explained in the user interface section.

Back Home

CPD Compensation

This panel is used to compensate for a contact potential difference (CPD). The CPD is detected by performing bias sweeps and watching the frequency shift.
When this panel is opened the range is set to a very small value, so no modulation will be actually on the output. You can now carefully increase the modulation amplitude until you see a parabola in frequency response. By clicking compensate the apex of the parabola is brought to center and therefore the contact potential compensated.



1	Sweep graph	Shows the frequency shift against the bias voltage. The bias voltage causing the least frequency shift is assumed contact potential difference (CPD).
2	**Speed**	Sets the frequency of the bias modulation
3	**Averaging**	The CPD is calculated by averaging the last specified number of results.
4	**CPD**	Indicates the calculated contact potential difference (CPD).
5	**Compensate**	Sets the Bias Voltage to the calculated CPD.
6	**Range**	Sets the range for the bias sweep.

Back Home

Bias Spectroscopy

The Bias spectroscopy module lets you perform fast bias sweeps while measuring arbitrary channels. Usually, it is used to record I(V) curves. Unlike the Bias sweep, the spectroscopy curves are performed on the real-time engine so timings are deterministic.

For a short visual introduction how to use the Bias spectroscopy while recording I/V and dI/dV curves watch this Video Tutorial on the Nanonis homepage.

A sweep can be started at any time using the **Start** button. If you start a sweep while scanning, the scan is paused for the duration of the spectroscopy measurement. The Z-Controller is set to hold (deactivated) by default during this time, so you don't have to turn it off manually.

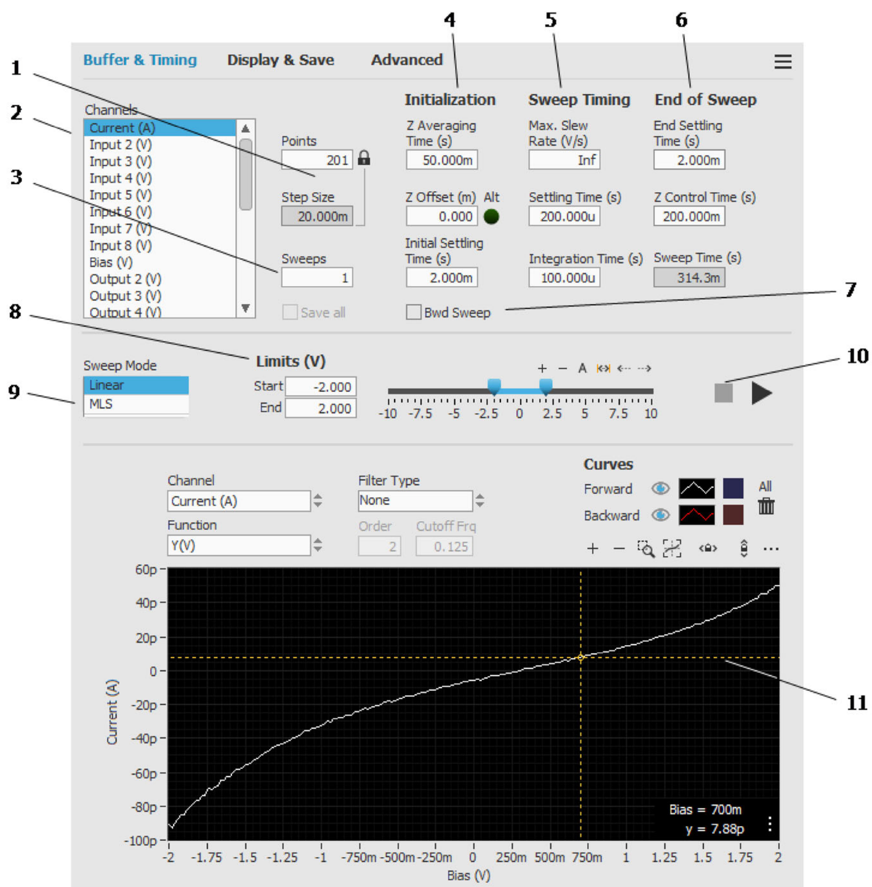
The module holds the data of 10 sweeps in its buffer, and lets you save the sweeps manually or automatically after they have been recorded. It displays all spectroscopy curves acquired, whether it is started from this panel or any other experiment that performs Bias spectroscopy measurements.

The Bias Spectroscopy module supports two modes of operation: Linear and Multi line segment modes. In Linear mode the sweep comprises of an equidistant set of points measured between a given start and stop value. In Multi line segment mode the user can (via a pop-up editor) define up to 16 individual line segments with definitions for **bias start**, **bias stop**, **integration time**, **settling time** and **number of steps**. A new Multi line editor has been included to allow user defined multi line segment operation.

At the bottom of this page, you will find a comprehensive timing diagram. It explains all the timing settings of the Bias spectroscopy module.

Note: Generic graph handling and numeric control handling are explained in the user interface section.

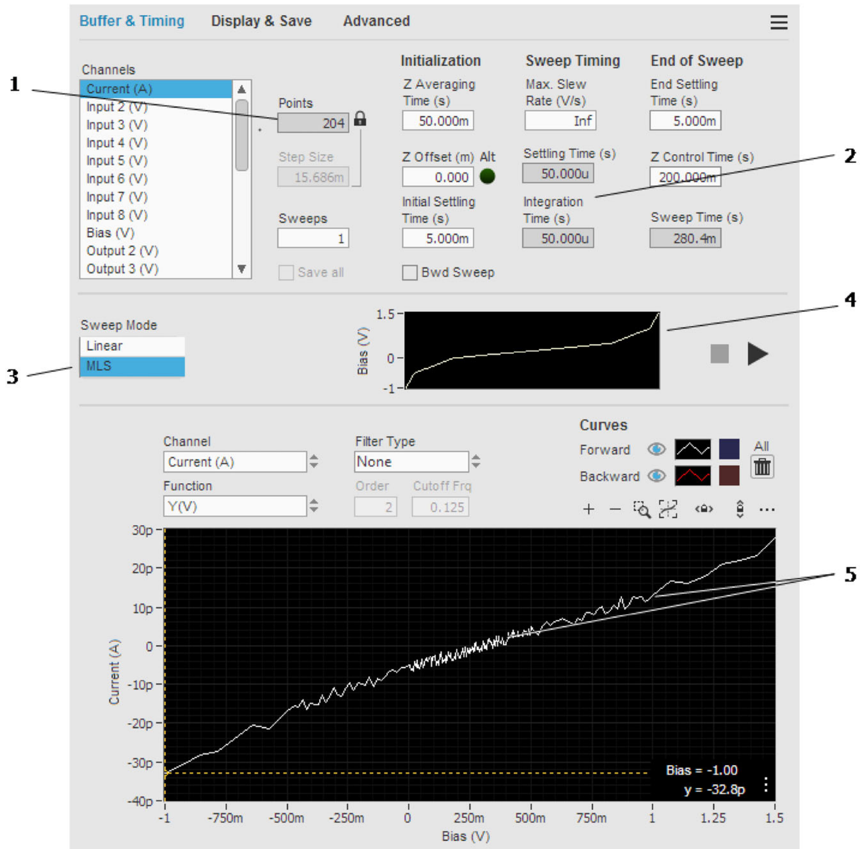
I) Buffer & Timing tab (Linear sweep mode)



1	Points Step Size Lock	When defining how many data points to record during a spectroscopy measurement, both **Points** and **Step Size** can be entered directly and the other is automatically calculated using the entered value (either **Points** or **Step Size**) and the **Start** and **End** values. In order to be able to reliably react to changes to either **Start** or **End** values, either **Points** or **Step Size** is declared as a "master" parameter.

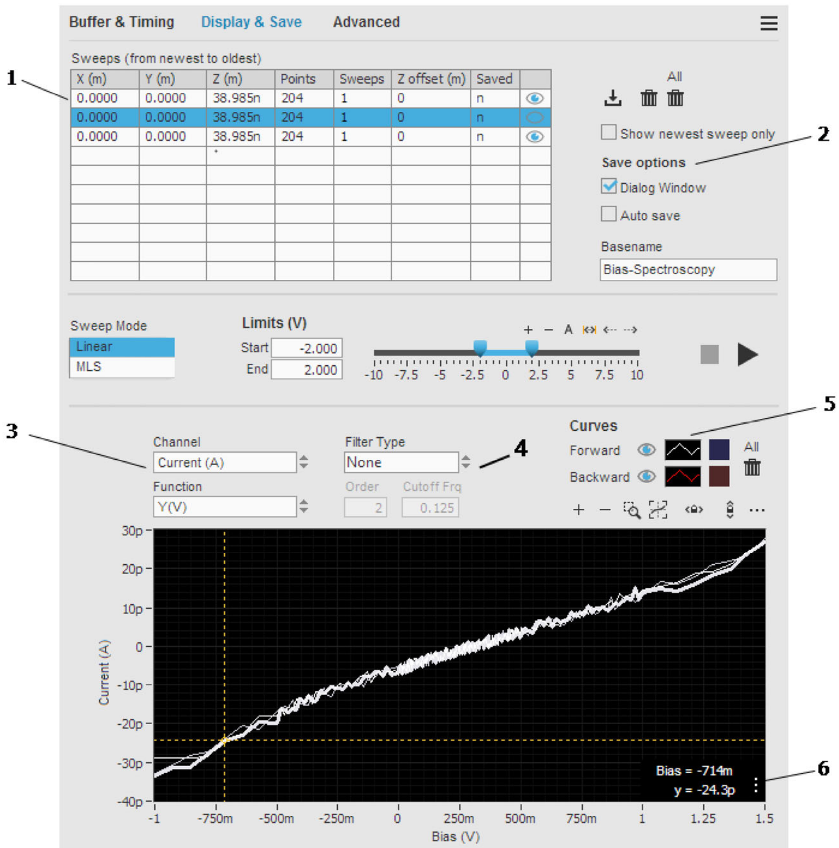
		Here the **Points** parameter was entered manually and is now defined as "master". Changes to **Start** or **End** in this state will automatically recalculate the **Step Size** parameter. Here the **Step Size** parameter was entered manually and is now defined as "master". Changes to **Start** or **End** in this state will automatically recalculate the **Points** parameter. If an entered value for **Step Size** results in a non-integer **Points** parameter, the **Step Size** parameter is highlighted with an associated tipstrip that with the current settings, the actual last Bias value for the spectroscopy measurement does not exactly match the specified **End**.
2	Channels	Select the channels to record.
3	Sweeps	Number of sweeps to measure and average.
4	Initialization	**Z averaging time**: duration over which the Z position is recorded and averaged before and after the sweep (the latter only if ***Record final Z position*** is selected in the Advanced section). After the initial Z averaging time, if **Z-Controller to Hold** is selected in the Advanced section, the Z-Controller is set to hold and the tip is placed at the previously averaged Z position (plus **Z offset**). **Z offset**: offset added to the initial averaged position Z_{aver} before starting to sweep. This parameter is disabled when **Z-Controller to Hold** is deselected in the Advanced section. The LED "Alt" next to the Z offset indicates if an alternate Z-controller setpoint is enabled. Note: the **TipLift** offset, defined in the Z-Controller, is not applied during a spectroscopy since the Z-Controller is not turned off but set to hold. **Initial settling time**: additional time to wait (in addition to the Settling Time) after the bias has ramped to the starting point of the sweep. Adjust this parameter to avoid transient effect induced by the bias change.
5	Sweep Timing	**Max. slew rate**: maximum rate at which the bias changes (before, during and after sweeping). Note: when **Max. slew rate** is high enough to change the bias within 1 real-time cycle (100us at 10kHz real-time frequency) changing the bias voltage doesn't require (extra) time. **Settling time**: time to wait after changing the bias to the next level and before starting to acquire data. Adjust this parameter to avoid transient effect induced by the bias change. **Integration time**: time during which the data are acquired and averaged.
6	End of Sweep	**End settling time**: time to wait after the sweep has finished and the bias is ramped to the initial value. **Z control time**: time during which the Z-Controller is enabled once a sweep has finished. When averaging multiple sweeps (i.e. **Sweeps**>1), the Z-Controller is enabled for this duration between each sweep. After the last sweep, it will wait the specified time before continuing a running scan. This ensures each sweep reliably starts from the same position. This parameter is disabled when **Z-Controller to Hold** is deselected in the Advanced section. **Sweep time**: time required to acquire one sweep.
7	Additional Flags	**Backward sweep**: select whether to measure the backward (return) sweep or the forward only. **Save all**: when selected, the data from the individual sweeps is saved along with the average data of all of them. This control is active only when **Sweeps** is greater than 1.
8	Limits (V)	Define the first and last bias values of the sweep.
9	Sweep operation mode	Switch between Linear (equidistant sweep) or Multi line segment sweep (User defined up to 16-different line segments)
10	Control buttons	Start/stop the spectroscopy measurement.
11	Sweep data	Data is uniformly spaced between the defined start and stop values

II) Buffer & Timing tab (Multi linear segment sweep mode)



1	Number of Points	This display shows the current number of measurement points for the current Multi line sweep configuration. It is not editable directly but reflects the values defined via the Multiline segment editor.
2	Timing	Due to the ability to define **Settling time** and **Integration time** independently for all segments of a multi-segment sweep, during multi line segment sweeps, these values are not editable and display the average values for these parameters.
3	Sweep operation mode	Switch between Linear (equidistant sweep) or Multi line segment sweep (user defined up to 16 different line segments)
4	Multi line segment preview	This graph shows the current bias change (Y Axis) against measurement point (X axis). Clicking the plot area of this graph will open the Multiline segment editor. It is intended as a preview regarding the expected measurement point density over the course of the sweep and does not correlate with the shape of the final data curve (e.g. Current vs Bias).
5	Sweep data	Sweep data is comprised of several linear regions of differing point density

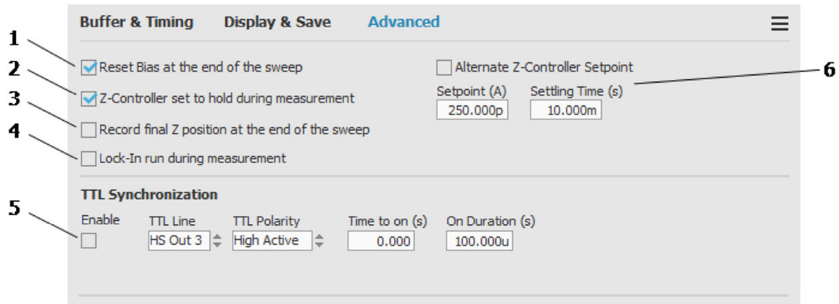
III) Display & Save



1	Sweeps list	List the last 10 acquired sweeps. The *X*, *Y* and *Z* columns indicate the tip position before starting the experiment. **Sweeps** initializes to 0 when starting the experiment and it shows the number of completed sweeps when the experiment finishes. If it is manually stopped and the total number of sweeps is not reached, it shows the number of completed sweeps in red. **Saved** indicates if the selected curve has already been saved into file (n=no, y=yes). Show newest sweep only activates display of the last measured curve. The checkmarks to the right of the table allow to show/hide the corresponding plots in the graph. · save the selected sweeps · delete the selected sweeps All · delete all sweeps *Tips:* *A menu appears when right-clicking on the list.* *Double-clicking on the title bar select all the sweeps.* *When the sweep selected in the Sweeps list has been saved,, the full file path appears when hovering the mouse over the list. If several are selected, only the path of the first selected sweep appears.*
2	Save options	**Dialog window**: when selected, a dialog window will open before saving data. **Auto save**: when selected, the data are automatically saved after each sweep. **Basename:** defines the file basename when saving sweeps. The file names will be "*Basename*NNN.dat" where NNN is an automatically increasing number. *Tip:* *When the sweep selected in the Sweeps list has been saved, the full file path appears when hovering the mouse over the list. If several are selected, only the path of the first selected sweep appears.*
3	Y-axis	**Channel**: select the signal you want to display in the graph. **Function**: select whether to display the signal or its derivative.
4	Data filtering	**Filter type**: select a filter to smooth the displayed data. Note: When saving, if any filter selected, filtered data are saved to file along with the unfiltered data. Some extra parameters are available for some filters: **Order**: filter order of a dynamic filter or width (in number of points) for the gaussian filter. **Cutoff frq**: cutoff frequency for dynamic filters. This assumes the acquired data have a sampling frequency of 1 Hz, regardless of the sweep timing. This parameter is very important when filtering as it defines how "strong" the filter is.
5	Curves	Define the curves display properties: **Checkmark**: show/hide the forward and/or backward plots. **Color box**: select the color of the plot #10. The colors of plots 2-9 are interpolated between the first and last plot colors.
6	Cursor	Display the cursor coordinates.

		The cursor can be hidden/shown by right-clicking on the graph and selecting Cursor>Show. The cursor can be moved from one curve to the other by dragging its crosshair. Note: Generic graph handling including cursor handling is explained in the user interface section.

IV) Advanced



1	Reset Bias	When selected, the Bias voltage returns to the initial value (before starting the sweep) at the end of the spectroscopy measurement (default). When not selected, Bias remains at the last value of the sweep. This is useful e.g. when combining several sweep segments. Example for an automated measurement (using Programming Interface): switch off Z-Controller, start spectroscopy sweep segments (only fwd sweep, no reset bias), set bias back to initial value, switch on Z-Controller.
2	Z-Controller Hold	Select whether to set the Z-Controller on hold (i.e. disabled) during the sweep. It is selected by default. When deselected, **Z-offset** and **Z control time** parameters are disabled.
3	Record final Z	Select whether to record the Z position during **Z averaging time** at the end of the sweep (after **Z control time**) and store the average value in the header of the file when saving. Using this option increases the sweep time by **Z averaging time**.
4	Lock-In run	Select whether to set the Lock-In to run during the measurement. When using this feature, make sure the Lock-In is configured correctly and settling times are set to twice the Lock-In period at least. This option is ignored when Lock-In is already running. This option is disabled if the **Sweep Mode** is MLS and the flag to configure the Lock-In per segment in the Multiline segment editor is set.
5	TTL Synchronization	TTL synchronization allows for controlling the high-speed digital outs according to the individual stages of the bias spectroscopy measurement. **Enable**: Selects whether the feature is active or not **TTL Line**: Chooses which digital port should be controlled **TTL Polarity**: Chooses the polarity of the switching action **Time to on (s)**: defines the time to wait once Settling Time has started before activating the TTL line **On Duration (s)**: Defines how long the TTL line should be activated before resetting
6	Alternate Z-Controller Setpoint	Configures the usage of an alternate Z-controller setpoint for the duration of the spectroscopy measurement. When switched on, the Z-controller setpoint is set to the specified value right after starting the measurement (after pausing the scan when scanning, according to step 1 of the timing diagram). After changing the setpoint the settling time will be waited for the Z-controller to adjust to the modified setpoint. Then the Z averaging will start (according to step 2 of the timing diagram). The original Z-controller setpoint is restored at the end of the measurement, before restoring the Z-controller state (according to step 13 of the timing diagram).

V) Acquisition Timing diagram

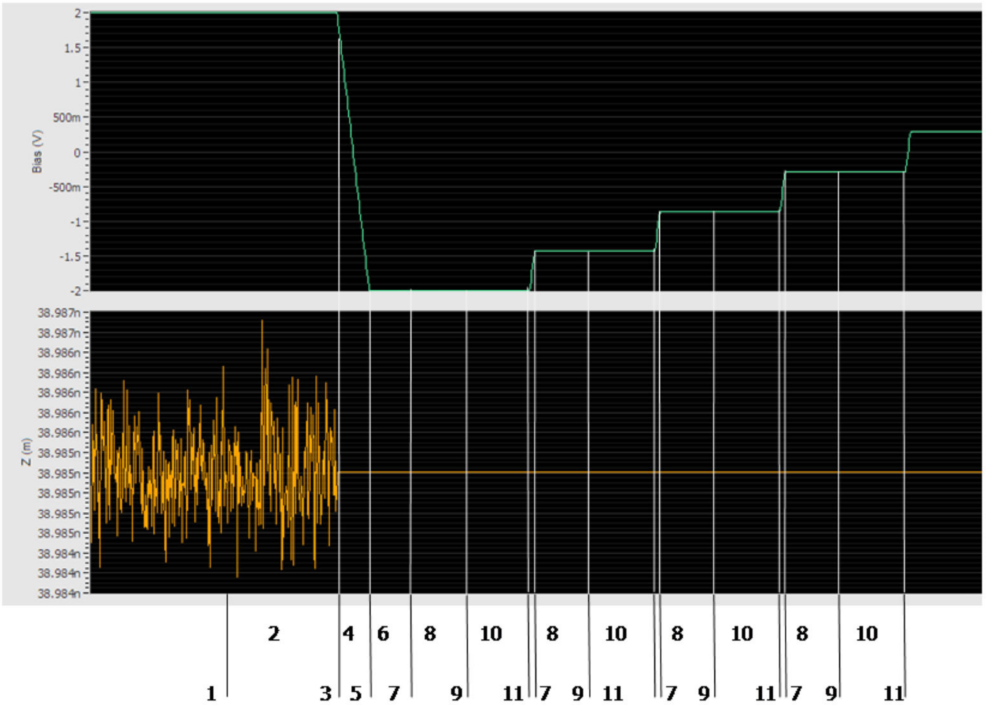


Figure: Bias spectroscopy start timing diagram

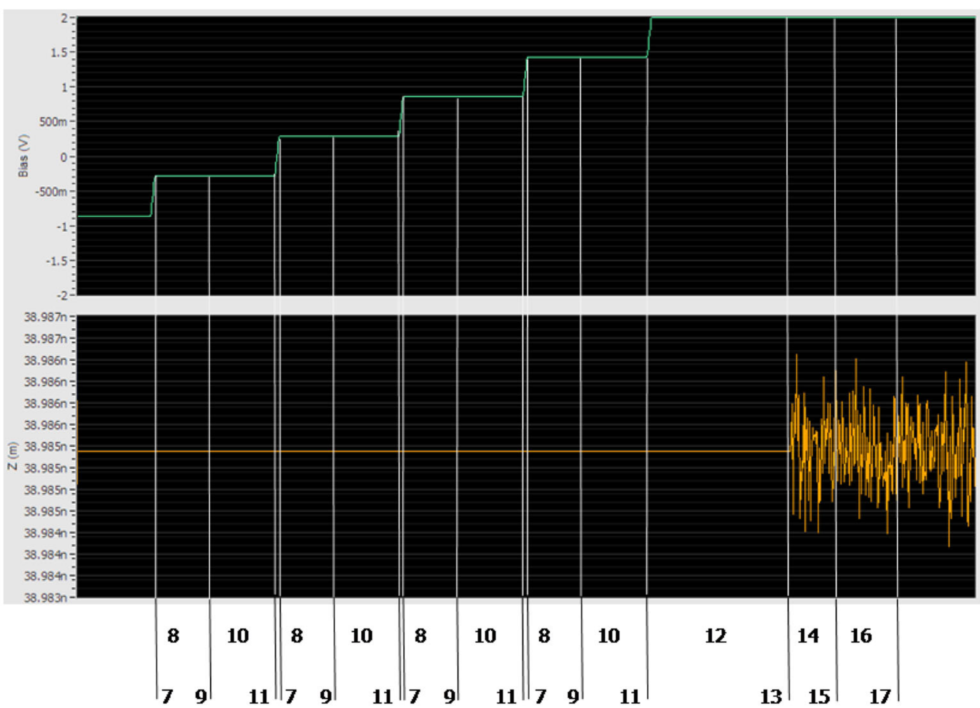


Figure: Bias spectroscopy end timing diagram

1. Start. If the tip is scanning, the scan is paused.
2. Measure the Z position during **Z averaging time** => Initial averaged position, Z_{avg}, is calculated.
3. Set Z-Controller to hold (i.e. disable it during the sweep); set Z position to "Z_{avg} + ***Initial Z-offset***" and start ramping to Start Bias
4. Ramp bias to next Bias value at ***Max. slew rate;***; set Lock-In to run (i.e. enable it during the sweep if the Lock-In run checkbox is selected).
5. Finished ramping, starting **Initial Settling Time**
6. Wait **Initial settling time** to let the system settle after having changed the bias in phase 4.
7. **Initial Settling Time** finished, starting **Settling Time**
8. Wait **Settling time** to let the system settle on the new bias level. This value is usually shorter than **Initial settling time** because the bias steps during the experiment are quite small.
9. **Settling Time** finished, start Acquiring

10. Acquire data of the selected channels during **Integration time**. The averaged measurements will be the recorded values for this bias.
11. Acquisition finished, move to next Point on the spectroscopy curve, ramp to next bias level at the **Max. slew rate.**
    (Repeat 7-11 for all remaining data points)
12. Reset Bias (if selected) and wait **End settling time**.
13. Switch Z-Controller to original on/off state.
14. Wait **Z control time** to let Z position settle.
15. **Z control time** finished, starting **Z averaging time**
16. Measure the Z position during **Z averaging time** => *Final averaged position is calculated.*
17. Resume scan, if the tip was scanning before starting the spectroscopy.

VI) TTL Synchronization Timing diagram

Figure: Bias spectroscopy TTL Synchronization timing diagram

1. Settling Time begins
2. Wait **Time to on (s)**
3. Switch TTL on
4. Wait **on Duration (s)**
5. Switch off TTL
    Repeat Steps 1 to 5 for each step

Note that the timing of the TTL switching is relative to the start of the **Settling Time** (After Slew rate limitations) but has otherwise no relation to other spectroscopy parameters. **Time to on (s)** and **on Duration (s)** are the only parameters which determine at which stage after the start of Settling Time the TTL signal switches on and off.

Back Home

Multi line segment editor

The Bias Spectroscopy module supports a Bias sweep using a sequence of linear segments, each with their own **Bias Start**, **Bias Stop**, **Integration Time**, **Settling Time**, **Points**, and **Lock-In** configuration.

This editor allows the user to define their own custom line segment data for performing flexible non-uniformly spaced bias sweeps. Up to 16 individual line segments can be defined.

I) Multi line segment editor

1	Segments List	This table shows the current set of data defined for the Multi line segment operation. A minimum of one and a maximum of 16 segments can be defined.
2	Data Entry	The data for the currently selected line segment are shown here and can be edited directly. Changes are updated in the Segments List and the preview graph. **Lock-In per Segment**: When selected, the Lock-In can be defined per segment. Otherwise, the Lock-In is set globally according to the flag in the Advanced section of Bias spectroscopy. **Bias Start** and **Bias Stop** define the start and end points of the current line segment. **Points** is the number of points to record for the selected segment. **Points Total** is the total number of points for the set of line segments defined. This is NOT equal to the sum of each individual line segment since duplicate measurements of overlapping elements are compensated. Configuration Per Point: **Settling Time (s)** is the time to wait after changing the bias to the next level and before starting to acquire data. Adjust this parameter to avoid transient effect induced by the bias change. **Integration Time (s)** is the time during which the data are acquired and averaged. Configuration Per Segment: **Initial Settling (s)** is applied at the beginning of the segment after the Lock-In setting is applied. **Lock-In run**: When selected, the Lock-In will run during the segment. This section appears only if the Lock-In module is licensed, and it is enabled only if the global Lock-In per Segment flag is enabled. Otherwise, the Lock-In is set globally according to the flag in the Advanced section of Bias spectroscopy.
3	Edit buttons	Start : add a copy of the currently selected line segment to the beginning of the data set. End : add a copy of the currently selected line segment to the end of the data set. If multiple segments are selected, the segment with the lowest index is chosen for replication. This deletes the currently selected segment(s). Selecting multiple segments allows simultaneous deletion of those segments. This operation will always ensure that at least one segment remains.
4	Apply	Applies current configuration to the Bias Spectroscopy module. Red color means that the configuration has changed but it is not applied yet.
5	Multi line segment preview	This graph shows the current bias change (Y Axis) against measurement point (X axis). It is intended as a preview regarding the expected measurement point density over the course of the sweep and does not correlate with the shape of the final data curve (Current vs Bias). The currently selected line segment is highlighted in red.

II) Multi line segment editor (Multiple segments selected)

1	Segments list	By selecting multiple line segments, certain data entry operations can be performed on each selected segment simultaneously.
2	Data entry	Following changes to the normal mode of operation can be observed: Disabled control means that this value cannot be set in multiple selection mode. Select each line segment individually to set the values. Values for the parameters are the same for all selected segments. Changes will overwrite the previous values. Values for the parameters are not the same for all line segments. Changes will overwrite the previous values.
3	Multi line segment preview	All selected segments are highlighted in red.

Back Home

Current

The current window displays the measured tunneling current. Also, some related values are calculated and displayed.
The related values and the calibration can be displayed or hidden using the button.

For supported preamplifiers with different gain options, the gain can be switched from this module. The calibration is adjusted automatically.

1	Current	The tunneling current is displayed here.
2	Related values	These indicators show some values related to the tunneling current and the bias voltage. The values calculated are: resistance, conductivity, (electrical) power and conductivity in units of $2e^2/h$.
3	Range switch	Switch the hardware gain via digital lines. The **Calibration** & **Offset** fields ("4") have different settings depending on the selected gain. This option only appears for certain systems.
4	Calibration & Offset	Calibration and offset of the current measurement. These are the same values as in the Signals Manager window.

5	Auto offset	Automatically corrects the offset such that the current is zero. Press when you know the current is zero.

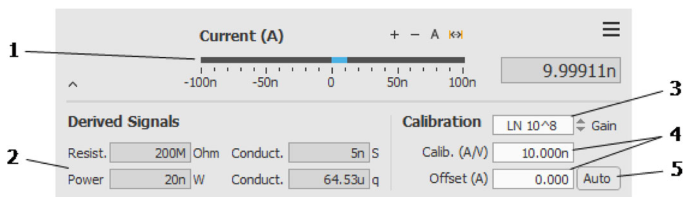
Back Home

Current Femto (DLPCA-200)

The current window displays the tunneling current measured using a Femto DLPCA-200 tunneling current preamplifier. Some other related values are calculated and displayed.
The related values and the calibration can be displayed or hidden using the button.

The current Femto module allows switching between different gains and cut-off frequencies. Digital lines are used to set the tunneling current preamp gain. Please refer to the Femto DLPCA-200 Datasheet for the cut-off frequencies of all available gain settings.

Important note: To enable external control (from the software) switch the gain selector to "Remote" and the H/L switch to "H". Coupling will always be DC, when set to AC this enables external control and the software will set it to DC.



1	Current	The tunneling current is displayed here.
2	Derived Signals	These indicators show some values related to the tunneling current and the bias voltage. The values calculated are: resistance, conductivity, (electrical) power and conductivity in units of $2e^2/h$.
3	Gain switch	The Femto DLPCA-200 allows 14 different settings, 7 low noise (LN, lower cut-off frequency) and 7 high speed (HS, higher cut-off frequency) settings. These are: LN 10^3, LN 10^4, …, LN 10^9, HS 10^5, HS 10^6, …, HS 10^{11}. The gain selector on the DLPCA-200 must be set to "Remote", the H/S selector to "H" to enable gain control from the software.
4	Calibration & Offset	Calibration and offset of the current measurement. These are the same values as in the Signals Manager window. Usually you shouldn't have to modify the calibration because this is defined by the gain setting ("3").
5	Auto offset	Automatically corrects the offset such that the current is zero. Press when you know the current is zero.

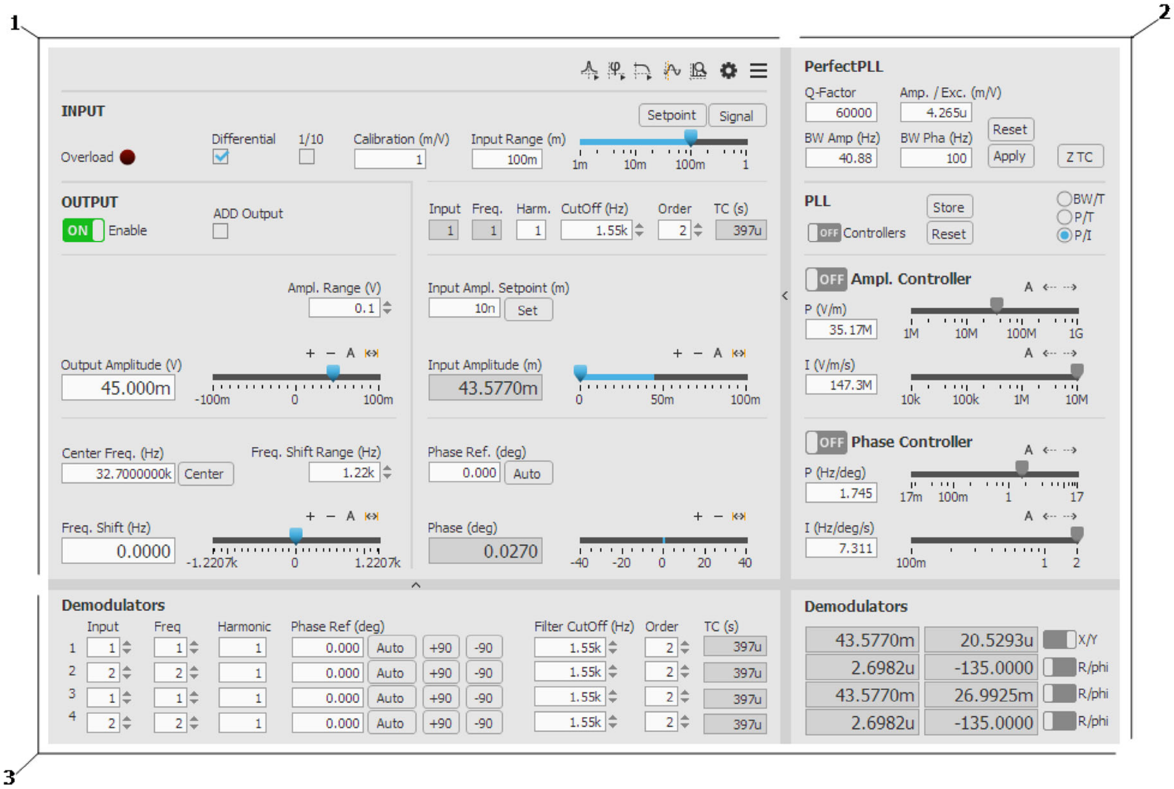
Back Home

Oscillation Control (optional module)

The Oscillation Control Window is used to adjust the parameters of the Phase Locked Loop (PLL). It allows the configuration of the modulator (excitation frequency and amplitude), the four demodulators, and the amplitude and phase controllers.

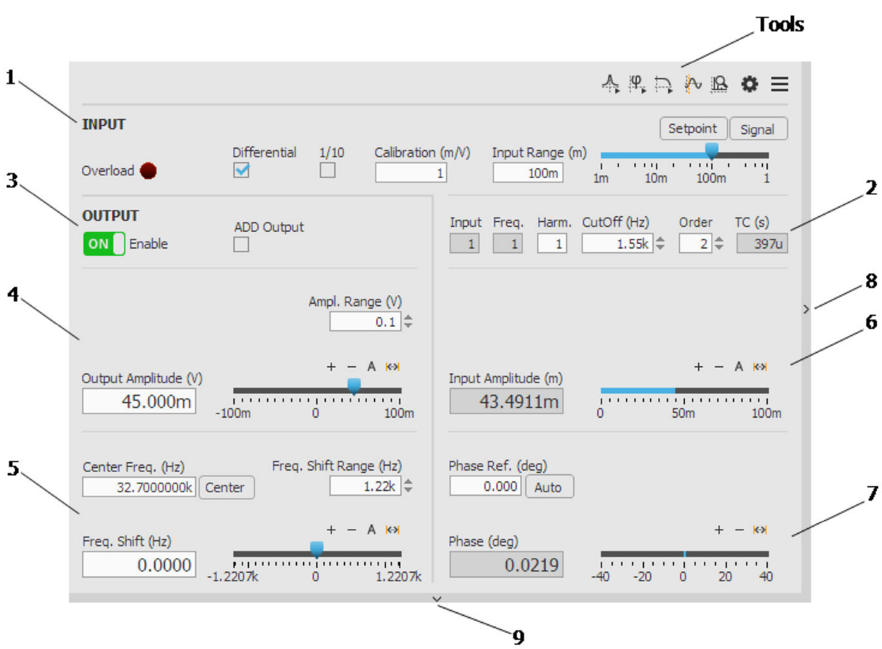
For dual OC4 system second oscillation control window is available to control the second PLL.

Instructions on how to set up the PLL can be found at the end of this document.



1	Input / Output settings	This area is always visible and contains the main controls for input / output variables such as amplitude and frequency
2	PLL settings panel	This area can be shown / hidden as required and contains the controls for various PLL functions
3	Demodulator settings panel	This area can be shown / hidden as required and contains settings and displays for various demodulator functions

Input / Output settings



1	Input Configuration	*Calibration* : Input calibration of your cantilever. This number is used to transform a voltage signal read at the input into a physical amplitude. If you don't know the number exactly, you can estimate it. It is only used for display. *Input Range* : The input range of the lock-in amplifier. You can either use the slider control or the numeric input to set the value such that the overload LED on the top right *does not* light up. Note that the amplitude can increase by several orders of magnitude on the resonance. Be sure to take this into account when you set the range. If you see that the range you selected is too small you can always adjust it. The input range is important for the accuracy of the controller. If the range is too small then it will go into overload at the resonance. If it is too big the full range of the AD converter is not used which will result in a poor resolution. The "Auto" button can be used to set the range to the optimal value. *Differential* : Switches the OC4 input between differential and single ended. *1/10* : When ticked, the input is attenuated by a factor of 10 (allowing input signals of up to 10V amplitude). *Overload* : If the range setting is bad (too small), this LED will light up. Adjust the *Input Range* if this LED is on. *Setpoint* button : set the input range to 125% of the *setpoint* of the amplitude controller. *Signal* button : sets the input range to 125% of the *Output Amplitude* value.
2	Input Demodulator settings	*Harm.* : Specify which harmonic of the input signal to demodulate in the lock-in. *CutOff (Hz)* : Cut off frequency of the low-pass filter of the lock-in amplifier. Should be higher than the desired demodulation bandwidth. Higher values will result in higher noise. *Order* : select the order of the low-pass filter for the lock-in amplifier. *TC (s)* : shows the estimated time constant of the current filter settings for the lock-in amplifier. *Input* : Shows the index of the OC4 input (OC4 # 1 input or OC4 # 2 Input) used for demodulation. *Freq* : Shows the index of the OC4 frequency (OC4 # 1 frequency or OC4 # 2 frequency) used for demodulation.
3	Output control	*Enable* : Switches the output of the OC4 on and off. *ADD Output* : Controls whether to add the signal at the “ADD” connector of the OC4 to the output signal or not.
4	Output Amplitude Configuration	*Output Amplitude (V)* : With a running controller the current amplitude is indicated. When the controller is switched off, this is the amplitude at which the resonator is driven. You can either use the digital control or the slider to the right of it. *Ampl. Range (V)* : The selector for the output range lets you select between 4 different ranges for the excitation output, 0.01, 0.1, 1 and 10 V. To get the maximum resolution out of the D/A converters, set the range to the lowest possible value, but make sure there’s still enough dynamic range for controlling the amplitude (when using the amplitude controller). Note that switching the range changes the overall phase and you might need to readjust the reference phase in the phase controller.
5	Output Frequency Configuration	*Center Freq. (Hz)* : The center frequency at which the phase locked loop runs. It can be adjusted with the digital control or the slider. *Freq. shift (Hz)* : Defines the shift from the *Center Freq. (Hz).* *Freq. shift range (Hz)* : Range of the slider to adjust *Freq. shift (Hz)*. *Center button* : Adjusts the *Center Freq. (Hz)* such that the current *Freq. shift (Hz)* is 0.
6	Input Amplitude Configuration	*Input Amplitude (m)* : Indicates the current amplitude.
7	Input Frequency Configuration	*Phase (deg)* : Indicates the current phase of the input signal relative to the output signal. *Phase Ref. (deg.)* : The phase of the loop is compensated by setting this value. The PLL usually runs at phase 0. The routine is to make a phase sweep to set the center frequency to the resonance, make a phase sweep and set the reference phase to such that the excitation is at a minimum.
8	Show / hide PLL settings	Show / hide panel with PLL settings(see below)
9	Show / hide demodulator settings	Show / hide panel with demodulator settings (see below)

Tools

·	Frequency Sweep	Launch the PLL Frequency Sweep module
·	Phase Sweep	Launch the PLL Phase Sweep module
·	Bode Plot	Launch the Bode Plot module to measure the transfer function over a frequency range
·	Signal Analyzer	Launch the PLL Signal Analyzer module to view the PLL input / output signals in the time domain
·	Zoom FFT	Launch the Zoom FFT module for high resolution analysis of the PLL demodulation
·	PLL Options	Open the PLL options window where several PLL related settings can be configured.

PLL Settings Panel

1	Show / hide PLL settings	Show / hide panel with PLL settings
2	PLL general settings	*Controllers* : Switch to start and stop both amplitude and phase controller *Store* button : Saves the current values for drive amplitude (excitation) and frequency shift. Use it to store good values once the oscillation control is running smoothly. *Reset* button : turns off both controllers, retrieves the stored values for drive amplitude and frequency shift and turns back on the controllers which were running before. *Parameter switch* : switches the display of the parameters for the PLL controllers to “Bandwidth / Time Constant”, “Proportional Gain / Time Constant” or “Proportional Gain / Integral Gain” modes. *“BW/T”* : G(s) = (1 + 1/(Ts)). *“P/T”* : G(s) = P(1 + 1/(Ts)). *“P/I”* : G(s) = P + I/s
3	Amplitude Control settings	*Ampl. Controller*: Switch to start and stop the amplitude controller. *BW (Hz) & TC (s)* : PI parameters when mode “BW/T” is selected **-or-** *P (V/m) & TC (s)* : PI parameters when mode “P/T” is selected **-or-** *P (V/m) & I (V/m/s)* : PI parameters when mode “P/I” is selected
4	Phase Control settings	*Phase. Controller*: Switch to start and stop the phase controller. *BW (Hz) & TC (s)* : PI parameters when mode “BW/T” is selected **-or-** *P (Hz/deg) & TC (s)* : PI parameters when mode “P/T” is selected **-or-** *P (Hz/deg) & I (Hz/deg/s)* : PI parameters when mode “P/I” is selected
5	Perfect PLL settings	Refer to PLL Setup section below
6	Demodulator display	This area displays the current result of the demodulators in the system as either X/Y or R/Phi depending on the chosen display mode. : Switches the display of the demodulated signals between X/Y and R/Phi.

Demodulator settings panel

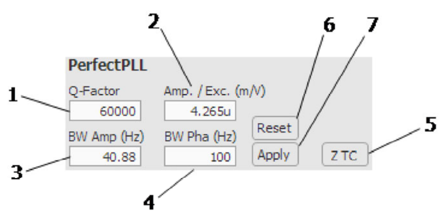
1	Show / hide Demodulator settings	Show / hide panel with demodulator settings

 2 3 4 5	Settings for: Demodulator 1 Demodulator 2 Demodulator 3 Demodulator 4	*Input* : Chooses which OC4 device input to use as a signal for demodulation *Freq* : Chooses which OC4 device frequency to use for demodulation *Harmonic* : Chooses which harmonic of the chosen frequency to use for demodulation *Phase Ref (deg)* : Adjusts the reference signal used for demodulation *Auto* : Sets the phase shift so Y component (or Phi component) are zero *+90* : Adds 90 degrees to the current phase ref *-90* : Subtracts 90 degrees from the current phase ref *Filter Cutoff (Hz)* : sets the cut-off frequency of the low-pass filter (affects the displayed TC (s) value). The filter is applied to the demodulated signals (X,Y). *Order* : Order of the low-pass filter (1 to 8) *TC (s)* : displays the calculated time constant of the applied low-pass filter settings
6	Demodulator display	This area displays the current result of the demodulators in the system as either X/Y or R/Phi depending on the chosen display mode.  : Switches the display of the demodulated signals between X/Y and R/Phi.

PLL setup

The PLL setup tool (PerfectPLL) uses an algorithm to optimize the amplitude and phase controller parameters (P and I gains or P gain and time constant) of the amplitude and phase controllers. It needs the Q factor, demodulation bandwidth and amplitude to excitation ratio to adjust the parameters. The Q factor and the amp/exc ratio can be identified using the frequency sweep tool.

The demodulation bandwidth is a tradeoff between speed and noise. You can set it to a high value (e.g. 100 Hz or higher) for fast measurements, but the noise level will also be higher. As the demodulation bandwidth defines the oscillation control speed, it is a good value to use as time constant for the Z-controller. Click the *Z TC* button to update the Z-controller time constant according to the *Phase Controller BW*.



1	Q factor	Q factor of the cantilever. Can be identified using the frequency sweeper.
2	Amplitude/Excitation ratio (m/V)	Amplitude to excitation ratio at the resonance. Can be identified using the frequency sweeper.
3	Amplitude Controller BW (Hz)	Demodulation bandwidth of the Amplitude controller.
4	Phase Controller BW (Hz)	Demodulation bandwidth of the Phase controller.
5	Update Z-controller time const	When clicking the **Z TC** button, the Phase controller bandwidth is used to set the Z-controller time constant. (Not available for OC4.5 station.)
6	Reset	Discard current changes to the Perfect PLL parameters and revert to the previously applied settings
7	Apply	Use the specified values to automatically optimize the amplitude and phase controller parameters (P gain and time constant)

Instructions on how to set-up the Nanonis PLL to track resonance frequency shift

1. Set reasonable values for *Output Amplitude (V)* and the expected resonance frequency of the cantilever (*Center Freq. (Hz)*) as well as appropriate *Freq. Shift Range (Hz)*.
2. From the *Tools* available on the main panel, select *Frequency Sweep*. A new panel will open that records the detected amplitude and phase as a function of frequency. The **zero** in the frequency shift (i.e. x-axis) corresponds to the center frequency set in the Oscillation Control panel (e.g. 32.7 kHz as displayed above). Perform a sweep either from the upper limit to the lower limit or vice versa.
3. Once the sweep is finished, the values for the resonance frequency *f_res*, *Q-factor*, *phase* and *Amplitude to Excitation ratio (A/U_exc)* are automatically measured and displayed in the corresponding plot information. If you click on *apply* then these new measured values will be stored in the appropriate fields in the Oscillation Control panel.
4. Note that several tries might be required before obtaining a good resonance curve. If you encounter some problems while sweeping the frequency make sure that the *Input Range (m)* is not overloaded (i.e. LED does *not* light up), the *Output Amplitude (V)* is not too high or that the sweeping speed is not too fast. Also note that there are several options to estimate the Q factor and the resonance frequency.
5. For non-contact AFM mode operation (or FM-AFM), the PI gains of the amplitude and phase controller have to be optimized. Choose the required values for *BW Amp (Hz)* and *BW Pha (Hz)* to set the required bandwidth of the amplitude controller and phase controller respectively. Clicking on *apply* will automatically adjust the PI parameters and the demodulator cut-off frequency to suitable values.
6. Turn both Amplitude and Phase controllers on. If necessary the dynamic reserve of the Lock-in can be adjusted via the slider *Input Range* in the Input Stage. The *Overload* LED should not activate (turn red).

Back Home

Signal Analyzer

The Signal Analyzer includes an oscilloscope and a spectrum analyzer that replace external devices to perform time and frequency-domain analysis.

The frequency spectrum is displayed as the square root of the Power Spectral Density (PSD).

A - Oscilloscope – Trigger Tab

A1	Signal	Select the signal channel you want to display.
A2	Save and Background Functions	- Save: save the curve to an ASCII file (.dat format). - Load: load a saved curve to the graph's background (reference curve). - Paste: paste the present curve to the graph's background. - Clear: delete the background curve. Note: Generic graph handling including cursor handling is explained in the user interface section.
A3	Zoom and Pan	Use those buttons to adjust the vertical scale and position of the trace.
A4	Trigger	Configure the oscilloscope trigger. - **Auto**: set the trigger parameters to pre-defined values. - **Mode**: - "Level": the trigger position is user-defined in the Level and Time fields. - "Immediate": the trigger is off. - **Source**: select the signal on which the trigger applies. If Source is different from Signal, the trigger cursor is not shown on the graph. - **Slope**: select the triggering direction. - **Level**, **Time**: define the trigger level and position. - **Arming**: Select whether the trigger is automatically or manually rearmed. Tip: to bring the trigger cursor to the center of the graph, right-click on the graph and select “Trigger>Bring to Center”.
A5	**Time Base**	Select the oscilloscope's time base (i.e. the x-axis range).
A6	**Update Rate**	Graph update rate. 1 for fastest rate, higher values reduce update speed, TCP traffic and CPU load.
A7	**PSD** show/hide	Show/hide the lower panel that contains the spectrum analyzer.

B - Oscilloscope – Measure Tab

B1	**Signal**	Display some useful information related to the current signal trace: - Peak-Peak: amplitude between the maximum and the minimum value of the signal displayed. - AC (rms): estimated Root Mean Square value of the AC component around the DC signal component.

		- DC: estimated mean value of the signal displayed. Note: AC and DC values are estimated and their accuracy relies on the appropriate choice of the Time base so that the signal's bandwidth is much lower than the Sampling rate.
B2	**Cursors**	The cursors are used to measure time difference and its equivalent frequency. Delta T: time difference between the two cursors. Freq. = 1/Delta T Note: Generic graph handling including cursor handling is explained in the user interface section. Tip: to bring the cursors to the visible portion of the graph, right-click on the graph and select “Cursors>Bring to Center”.

C - Spectrum Panel

C1	Save and Background Functions	- Save: save the curve to an ASCII file (.dat format). - Load: load a saved curve to the graph's background (reference curve). - Paste: paste the present curve to the graph's background. - Clear: delete the background curve.
C2	**FFT Window**	Select the window function applied to the timed signal before calculating the Power Spectral Density. Possibilities are "No windowing", "Hanning", "Hamming", "Blackman-Harris", "Exact Blackman", "Blackman", "Flat Top", "4 Term B-Harris", "7 Term B-Harris" and "Low Sidelobe".
C3	FFT **Averaging**	- Averaging: "No averaging", "Vector averaging", "RMS averaging" or "Peak hold". - Weighting: "Exponential" or "Linear". - Count: specifies the number of averages used for "RMS" and "Vector" averaging. If weighting mode is "exponential", the averaging process is continuous and new spectral data have a higher weighting than older ones. If weighting mode is "linear", the averaging combines count spectral records with equal weighting and then stops. - Restart: restart the averaging process.
C4	**Cursors**	Display the cursors coordinates (PSD vs. Frequency). The cursors can be hidden/shown by right-clicking on the graph and selecting Cursor>Show. The second cursor is only shown when the averaging type “RMS” is selected. The color indicates on which curve the cursor is snapped to. Note: Generic graph handling including cursor handling is explained in the user interface section. Tips: the cursor can be moved from one curve to the other by dragging its crosshair, if Cursor>Stick to one curve is deselected in the right-click menu. It is also possible to make it jump to the other curve by right-clicking on the graph and selecting Cursor>Jump to Other Curve. If you drag the cursor using its vertical line, it won’t jump to the other curve.
C5	Spectrum curve	Spectrum of the selected signal.
C6	Background reference curve	Spectrum of the loaded or pasted curve in the background. This curve can be used as a reference to compare the present spectrum to a previously acquired one. This is useful to compare noise levels and peaks amplitude. Tip: the color of this curve can be changed by clicking on the small box next to the curve legend (top left corner of the graph).

Back Home

Frequency Sweep

With this tool you can record the frequency response of the cantilever. See the general topic about measurements.
Make sure that the amplitude and phase controller in the Oscillation Control window are switched off. Adjust the span of the frequency sweep in the Sweep Range (Hz) control. Specify the number of data points you want to acquire in this frequency span (“2”).

Press the up or the down button to perform an up or down sweep. In the Frequency Shift (Hz) indicator you see the actual frequency shift from the center frequency as you do in the PLL window.

After a sweep has finished, the module will automatically determine the resonance frequency, Q-Factor and other properties. The Q-Factor, Amp./Exc. (m/V), Amplitude Bandwidth and Phase Bandwidth will be automatically copied to the **PerfectPLL** section of the Oscillation Control module, but they won't be really applied until pressing the **Apply** button in the Oscillation Control window.

The **Apply** buttons in the Frequency Sweep are used to update the Center Frequency (Hz) and Phase Reference (deg) values in the Oscillation Control module with the Resonance Frequency (Hz) and Phase (deg) values obtained in the Frequency Sweep.

After a sweep has finished, it's possible to re-estimate the Q-Factor with different parameters without starting again the whole sweep in the graphs just by updating the **Fit Length** value.

Note: Generic graph handling including cursor handling is explained in the user interface section.

1	Control Buttons and Frequency Shift (Hz) indicator	Performs a frequency sweep from the **Upper** limit (Center Frequency + Sweep Range Upper) to the **Lower** limit (Center Frequency - Sweep Range Lower). Performs a sweep from the Lower to the Upper limit. Stops a running sweep. **Frequency Shift (Hz)**: Indicates the actual frequency shift from the center frequency, and its range is set through the Sweep Range (Hz) control.
2	Sweep Configuration	**Sweep Range (Hz)**: Range of the Frequency Shift (Hz). The right click menu offers the option to select a different range. **# Points**: Specifies the number of points distributed over the frequency range. **Period (ms)**: Measurements period. Linked to Measurements period in TCP receiver. Defines the measurement speed. Due to the oversampling, higher values result in more accurate data. **Initial Settling time (s)**: Settling time of the signals before starting to sweep.
3	Display & Save	**Parameter Estimation Methods**: - **Res. Freq.**: The resonance frequency can be estimated through 3 different methods: **Maximum Amplitude** or **Amplitude Fit** of the amplitude curve, or **Phase Fit** of the phase curve obtained after sweeping. - **Q**: The Q-Factor can be estimated through 3 different methods: **Phase Slope** (adjusting the Fit Length), **Amplitude Fit**, or **Phase Fit**. **Fit Length**: Sets the number of samples used to draw the fit line when the Parameter Estimation Method for Q is Phase Slope. **Fit Color**: Sets the color of the fit (line or curve, depending on the Parameter Estimation Method for Q). **Show Fit**: When checked, the fit (line or curve, depending on the Parameter Estimation Method for Q) will show up at the end of the sweep. **Save Dialog**: When active, a save dialog will open when saving the data. If it is not checked, the data will be stored without further asking. **Auto Save**: When checked, the newly recorded data will be stored automatically. **Plot**: select to which plot (buffer) the next sweep will be saved to. : show/hide the plot curve on the graphs. : save the data to ASCII file (.dat). : clear the plot. **Characteristic values**:

		– Res. Freq. (Hz): Resonance frequency. The accuracy is limited by the range and # points settings. – Q: Q-Factor. – Phase (deg): phase at the resonance frequency. – A/U_exc (nm/mV): Amplitude to excitation quotient – #Pts: Number of samples **Apply**: Updates the Center Frequency (Hz) and Phase Reference (deg) values in the Oscillation Control module with the Resonance Frequency (Hz) and Phase (deg) values obtained in the selected sweep.
4	Amplitude Graph	Displays the Amplitude (m) over the Sweep Range (Hz). At the end of the sweep, it immediately places the cursor at the point where the resonance frequency has been estimated. This depends on the **Parameter Estimation Method** used for the resonance frequency. If the **Parameter Estimation Method** for Q is Amplitude Fit, it also draws the fit curve of the obtained amplitude sweep.
5	Phase Graph	Displays the Phase (m) over the Sweep Range (Hz). At the end of the sweep, if the **Parameter Estimation Method** for Q is Phase Slope, it draws the fit line based on the number of points defined by the **Fit Length**. This can be changed after the sweep has finished by modifying the Fit Length, and thus re-calculating the Q-Factor for that sweep. If the **Parameter Estimation Method** for Q is Phase Fit, it draws the fit curve of the obtained phase sweep.

Back Home

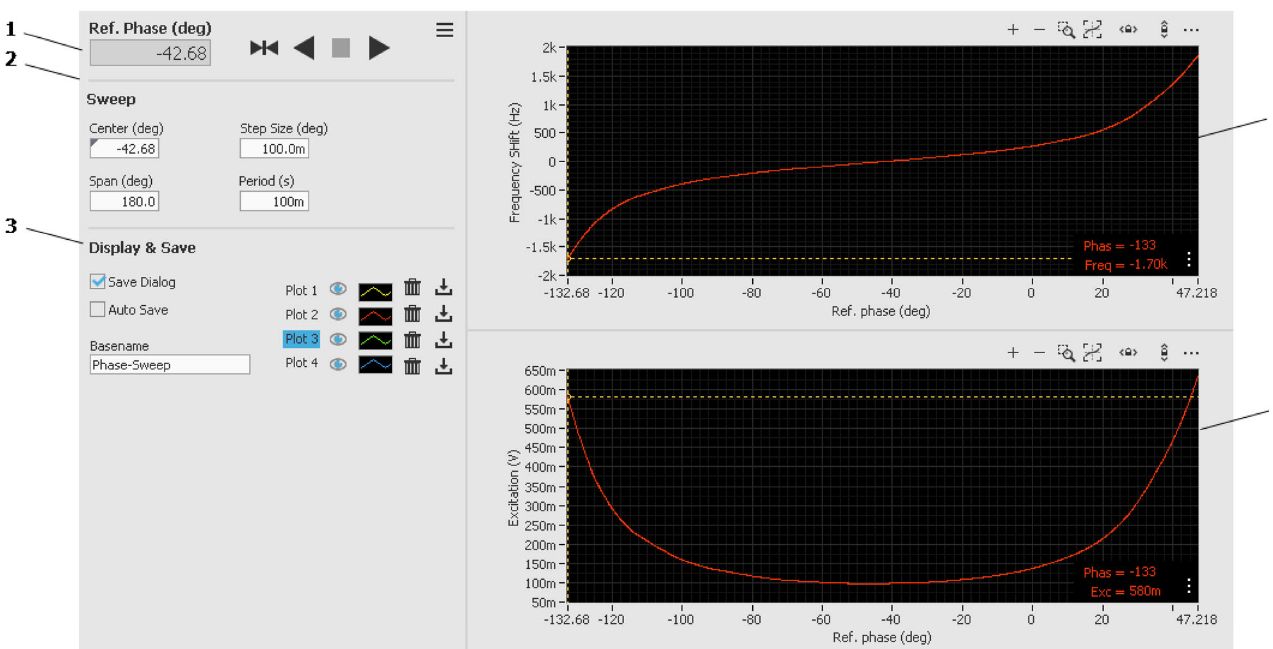
Phase Sweep

The Phase Sweep is used to record the frequency shift and the excitation amplitude as a function of the reference phase of the running PLL.

Before starting a sweep, make sure that the amplitude and phase controllers in the PLL window are switched ON and the PLL is running properly, close to the resonance frequency.

The sweep is controlled from the host computer (i.e. not from the real-time controller), therefore limiting the acquisition speed to about 50 pt/s.

Note: Generic graph handling including cursor handling is explained in the user interface section.



1	Control Buttons and Ref. Phase (deg) indicator	Starts to sweep from the current location, in the direction specified by the arrow. The sweep will stop when reaching the upper or lower limit specified by the **Center (deg)** and **Span (deg)** settings. Starts to sweep from the current location toward the **Center (deg)**. Stops a running sweep. **Ref. Phase (deg)**: Displays the current value of the Reference Phase.
2	Sweep Configuration	**Center (deg)**: Center of the sweep range. The right click menu offers the option to set it to the current phase value. **Span (deg)**: Range of the sweep.

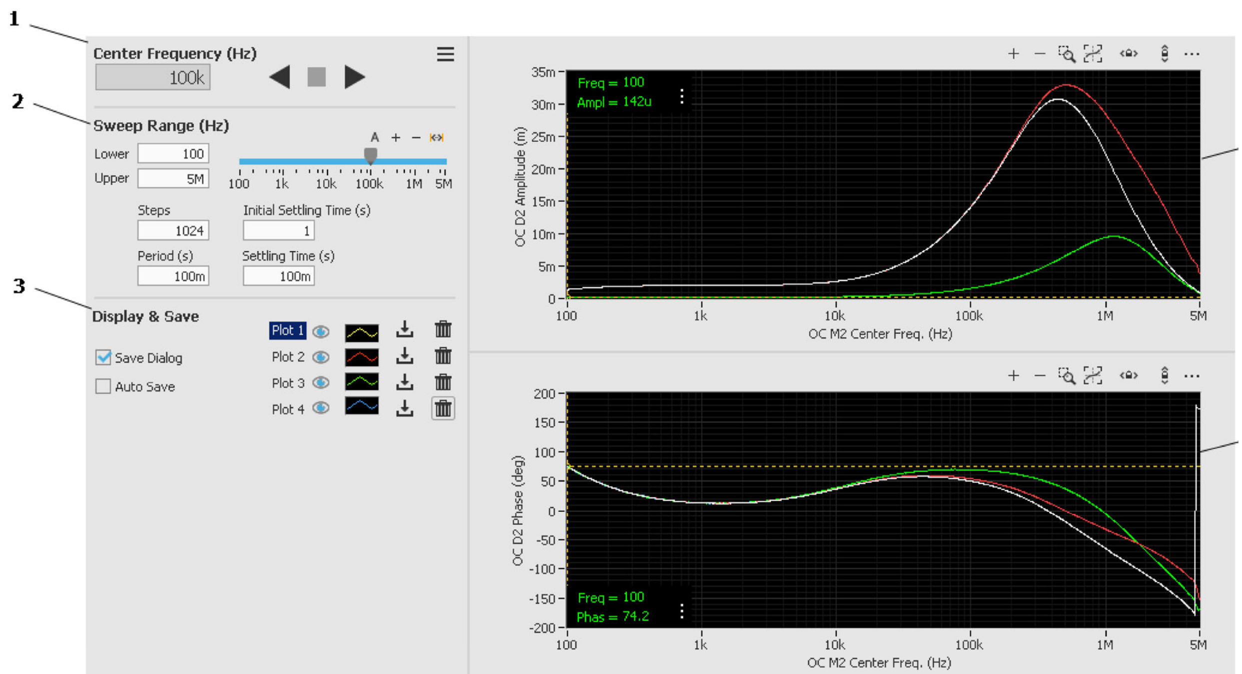
		Step Size (deg): Defines the phase step size during the sweep. **Period (deg)**: Defines the measurement speed. Higher values result in more accurate data because of the oversampling. This control is the same as the one in TCP receiver.
3	Display & Save	**Save Dialog**: When active, a save dialog will open when saving the data. If it is not checked, the data will be stored without further asking. **Auto Save**: When checked, the newly recorded data will be stored automatically. **Basename**: Defines the file basename when saving sweeps. The file names will be "BasenameNNN.dat" where NNN is an automatically increasing number. **Plot**: select to which plot (buffer) the next sweep will be saved to. : show/hide the plot curve on the graphs. : save the data to ASCII file (.dat). : clear the plot.
4	Frequency Shift Graph	Displays the Frequency Shift (Hz) versus the Reference Phase (deg).
5	Excitation Graph	Displays the Excitation (V) versus the Reference Phase (deg).

Back Home

Bode Plot

With this tool you can sweep the center frequency (in a logarithmic scale) and measure amplitude and phase. This results in the **Transfer Function** of the cantilever (bode plot).
Select the sweep range for the center frequency, the number of steps in this range, the settling time and the measurement period and click on the up or down arrow to start a sweep.

Note: Generic graph handling including cursor handling is explained in the user interface section.



1	Control Buttons and Center Frequency (Hz) indicator	Performs a center frequency sweep from the **Upper** limit to the **Lower** limit. Performs a sweep from the Lower to the Upper limit. Stops a running sweep. **Center Frequency (Hz)**: Indicates the current center frequency and its range is set through the Sweep Range (Hz) configuration. It slider.
2	Sweep Configuration	**Sweep Range (Hz)**: Range of the Center Frequency (Hz) defined by the **Lower** and **Upper** values. **Steps**: Specifies the number of steps distributed over the frequency range. **Initial Settling time (s)**: Settling time after setting the Center Frequency to the starting point of the sweep. **Settling time (s)**: Settling time after step of the sweep. **Period (s)**: Measurements period. Linked to Measurements period in TCP receiver. Defines the measurement speed.

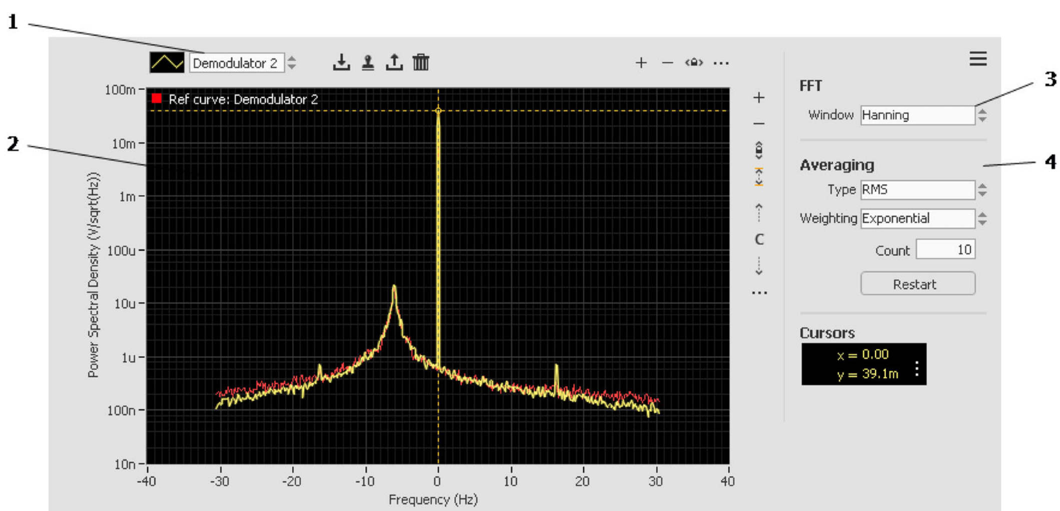
		Due to the oversampling, higher values result in more accurate data.
3	Display & Save	**Save Dialog**: When active, a save dialog will open when saving the data. If it is not checked, the data will be stored without further a **Auto Save**: When checked, the newly recorded data will be stored automatically. **Plot**: select to which plot (buffer) the next sweep will be saved to. : show/hide the plot curve on the graphs. : save the data to ASCII file (.dat). : clear the plot.
4	Amplitude Graph	Displays the Amplitude (m) over the Sweep Range (Hz).
5	Phase Graph	Displays the Phase (m) over the Sweep Range (Hz).

Back Home

ZoomFFT

This module calculates the spectrum (FFT) of the demodulated OC4 input signal around the center frequency. It calculates the complex FFT from the demodulated X/Y signals of any of the Oscillation Control demodulators. This is useful to analyze the signal in the frequency domain with high resolution around the center frequency set in the Oscillation Control module.

The bandwidth of the ZoomFFT is defined by the filter settings (cut-off frequency) for the corresponding demodulator.

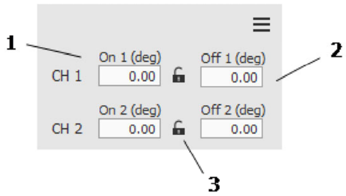


1	Demodulator	Selects from which demodulator signals to calculate the FFT.
2	Graph	Displays the ZoomFFT of the selected Demodulator ("1").
3	Window	Window function applied to the time signal before calculating the FFT.
4	Averaging	**- Type**: "No averaging", "Vector averaging", "RMS" or "Peak hold". **- Weighting**: "Exponential" or "Linear". **- Count**: specifies the number of averages used for "RMS" and "Vector" averaging. If weighting mode is "exponential", the averaging process is continuous and new spectral data have a higher weighting than older ones. If weighting mode is "linear", the averaging combines count spectral records with equal weighting and then stops. **- Restart** the averaging process.

Back Home

OC4 Sync (digital)

Use this module to control the digital I/O channels of the High-Speed Digital Out 1 and 2 channels of the RC5. It can be used to synchronize external devices with the Oscillation Control module.



1	On angle (CH1)	Angle of the Oscillation Control output (excitation) at which the digital channel 1 is set to **high**.
2	Off angle (CH1)	Angle of the Oscillation Control output (excitation) at which the digital channel 1 is set to **low**.
3	Link angles	When the chain is open, the **on** (“1”) and **off** (“2”) angles can be set independently. When the chain is closed, the difference between **off** angle and **on** angle is kept constant and only the **on** angle can be modified.

Back Home

Beam Deflection

The beam deflection module allows you to calibrate the deflection signals and it displays the horizontal and vertical deflection in a graph. There are two different modules available. One module just uses horizontal and vertical deflection inputs while the second module uses an additional intensity (sum) input.
Some systems show a dependency of the vertical deflection from the tip x/y position. The beam deflection module allows correcting for the effect (useful for intermittent contact or contact mode).

Horizontal and vertical deflection only

1	Tab selector	Switch between Calibration tab (configuration of deflection signals), Display tab (shows current deflection values) and Options tab (advanced settings related to beam deflection).
2	Horizontal deflection name	Enter a name for the horizontal deflection signal here. This name will then be used for the horizontal deflection signal throughout the software, e.g. it will be available for acquisition in scan control.
3	Horizontal deflection unit	Unit of the horizontal deflection signal. If you don't know the calibration you can leave it in Volts (V), or you can use a calibration ("3" and "4") to transform the signal to meters (m) or to a force in Newton (N).
4	Calibration	Calibration of the horizontal deflection signal. If you want to leave the signal in Volts, set it to1. If you want to transform the signal to meters or Newton, enter the calibration (in m/V or N/V) here.
5	Offset	Offset of the horizontal deflection signal (in physical units).
6	Auto offset	Automatically adjusts the offset of the horizontal deflection signal such that the current deflection is 0.
7	Vertical deflection	Configure the vertical deflection, similar to the horizontal deflection.
8	Graph	Displays the horizontal and vertical deflection as a spot in the graph.
9	Current deflection values	The display tab shows the current deflection values.
10	Vertical deflection	Allows correcting for a linear dependency of the vertical deflection of the tip position. The 2 parameters (X correction and Y correction) define

	correction	the plane which is subtracted from the measured deflection value.

Horizontal deflection, vertical deflection and intensity

The module is basically identical, it just adds a third input for the intensity (sum) signal.

Back Home

Interferometer Controller

The Interferometer controller window gives the user access to the PI parameters that control the interferometer piezo (W piezo) based on the interferometer readout signal. Use the Interferometer Calibration module to determine the working point for the interferometer controller.

1	Controller on/off switch	Turns the Interferometer Controller on or off.
2	Interferometer Signal	The controller's task is to continuously adjust the W piezo position in order to keep the interferometer signal as close as possible to the working point (defined by calibrating the interferometer system using the interferometer calibration module or set manually using the corresponding controls).
3	Controller parameters	Proportional and integral controller settings (or proportional gain and time constant). The higher the P and I gain values are set, the faster the controller reacts to deviations from the control signal to the setpoint. If the values are set too high, the controller will start to oscillate. If

		this happens, lower the controller parameters.
4	W piezo position	Indicates the position of the W-Piezo. The position can also be varied by using the slider bar when the controller is not running.
5	Show/Hide calibration	Shows/hides the calibration and setpoint controls on the right side of the window.
6	Interferometer Sensitivity	Sets the calibration of the interferometer signal to convert the voltage reading into a position in meters. This parameter is usually determined using the interferometer calibration module.
7	Setpoint	The setpoint is the value of the interferometer signal at the desired working point. This value can be used as offset for the interferometer signal (so the signal is controlled to 0) or as controller setpoint (so the interferometer signal is controlled to the setpoint). This parameter is usually determined using the interferometer calibration module.
8	Offset/Setpoint switch	Switch the setpoint control ("7") between offset and setpoint.
9	Controller parameter switch	Switches controller parameters between P/I (proportional gain/integral gain) and P/T (proportional gain/time constant). The formula for the controller in the frequency domain is: $G(s) = P + I/s = P(1 + 1/(Ts))$. The integral gain and time constant are related as follows: $I = P/T$.
10	Slope: pos/neg	Switches the controller slope between positive and negative. Depends on which edge (falling or rising) of the signal you want to control. Usually the slope is set using the calibration routine, but you can change it here if needed.
11	Reset	Switches off the controller (when running), sets the W-Piezo to 0 and restarts the controller (if it was running).
12	Null deflection	Adds the current interferometer signal to the offset so the actual value becomes zero. Only works in "Offset" mode.
13	Piezo calibration	Calibration of the W-Piezo. This parameter is usually determined using the interferometer calibration module.
14	Origin	Working point for the W-Piezo. This value is applied as offset to the W signal, so a value of 0 corresponds to the initial working point. This parameter is usually determined using the interferometer calibration module.

Tools

·	Sinusoidal Calibration	Opens the calibration routine for standard interferometers (resulting in a sinusoidal signal pattern when moving the piezo).
·	Fabry-Pérot Calibration	Opens the calibration routine for Fabry-Pérot (or other type) interferometers.

Back Home

Interferometer Calibration (sinusoidal)

The Interferometer calibration is used to calibrate the W-Piezo and Interferometer signal as well as to define the working point for the Interferometer Controller. This calibration routine works for standard interferometers showing a sinusoidal pattern in the interferometer signal when moving the W piezo.
The calibration routine will sweep the W-piezo continuously between 2 selected voltages and record the interferometer signal. Make sure to set the sweep limits to see at least one full period of the sine wave. Also set the desired sweep speed and the wavelength of the interferometer laser (required for correct calibration).
The routine will try to fit a sine wave to the acquired data. Based on the sine wave parameters (frequency and amplitude) the calibrations of the W-piezo and of the interferometer signal are determined. Depending on the slope setting (pos/neg) the working point will be set on the rising or falling edge of the sine wave, but always at the DC value of the interferometer signal to have sufficient room and most linear behavior in both directions.

1	Interferometer Signal	Displays the interferometer signal recorded during forward and backward movement of the W piezo and the sine wave fitted to best match both curves.
2	Sweep speed	Sets the speed for the W piezo movement during the calibration procedure.
3	W piezo limits	Sets the minimum and maximum limits for the W piezo movement during the calibration procedure.
4	Peak-peak	Displays the peak-peak voltage of the recorded interferometer signal.
5	Wavelength	Specify the wavelength of the interferometer laser for correct calibration.
6	Navigation buttons	• **next** will continue to the second step of the calibration procedure • **cancel** will stop the calibration procedure (no changes will be applied to the interferometer module).

7	Interferometer Signal	Displays the interferometer signal recorded during forward and backward movement of the W piezo and the sine wave fitted to best match both curves. The cursor shows the determined working point (based on the slope setting ("10")).
8	Iterations for fit	The sine fit algorithm is an iterative process. Specify how many iterations should be done for the fitting.
9	Calibration data	Shows the data resulting from the calibration: • **Piezo Calibration**: calibration of the W piezo. • **W-Piezo Origin**: working point for the controller. This value is used as offset for the W signal. • **Interferometer Sensitivity**: calibration of the interferometer signal (slope of the fitted sine wave) at the working point. • **Interferometer Setpoint**: Interferometer signal value at the working point. This will be used as setpoint for the Interferometer Controller.
10	Slope (pos/neg)	Switches between positive and negative slope. The working point will be set to the rising or falling sine edge depending on this setting.
11	Navigation buttons	• **back** will return to the first page of the calibration routine (e.g. to adjust the W piezo sweep range) • **cancel** will stop the calibration procedure (no values will be applied) • **accept** will apply the current values to the Interferometer Controller module.

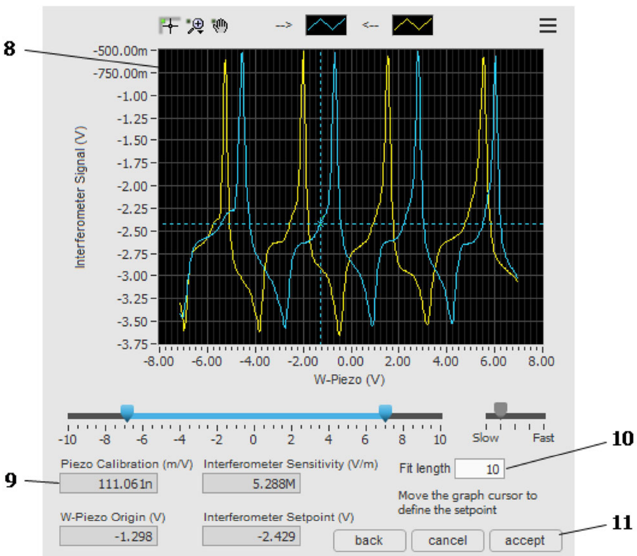
Back Home

Interferometer Calibration (Fabry-Pérot)

The Interferometer calibration is used to calibrate the W-Piezo and Interferometer signal as well as to define the working point for the interferometer controller. This calibration routine is intended for Fabry-Pérot type interferometers, but should work for other types as well.
The calibration routine will sweep the W-piezo continuously between 2 selected voltages and record the interferometer signal. Make sure to set the sweep limits to see at least two periods of the interferometer signal. Also set the desired sweep speed and the wavelength of the interferometer laser (required for correct calibration).
The routine will try to detect the maxima of the interferometer signal. Based on the distance of the maxima (and the wavelength) the calibration of the W-piezo is determined. In the second step of the procedure the user can define the working point by moving the cursor to the desired location. The slope at the selected point is the Interferometer signal calibration.

1	Interferometer	Displays the interferometer signal recorded during forward and backward movement of the W piezo.

	Signal	
2	W piezo limits	Sets the minimum and maximum limits for the W piezo movement during the calibration procedure.
3	Sweep speed	Sets the speed for the W piezo movement during the calibration procedure.
4	Peak-peak	Displays the peak-peak voltage of the recorded interferometer signal.
5	Wavelength	Specify the wavelength of the interferometer laser for correct calibration.
6	Peak detection parameters	• **Peak fit width** defines the width (number of points to use) for the peak detection algorithm (quadratic polynomial fit). • **Threshold factor** specifies the minimum value of a peak in standard deviations above the average value. The dashed line shows the threshold. This is used to prevent detecting false peaks since we only want to detect the maximum of each period of the interferometer signal. • **# Max found** indicates the number of maxima found. The maxima are also indicated in the graph using 'x' markers. Make sure to continue the calibration process only if the number of maxima detected is correct, otherwise the calibrations will be wrong. Due to the peak detection algorithm the displayed maxima ('x' markers) might not be at the highest points, but that should not affect the calibration procedure.
7	Navigation buttons	• **next** will continue to the second step of the calibration procedure • **cancel** will stop the calibration procedure (no changes will be applied to the interferometer module).



8	Interferometer Signal	Displays the interferometer signal recorded during forward and backward movement of the W piezo. Use the cursor (attached to the forward sweep curve) to define the working point.
9	Calibration data	Shows the data resulting from the calibration: • **Piezo Calibration**: calibration of the W piezo. • **W-Piezo Origin**: working point for the controller. This value is used as offset for the W signal. • **Interferometer Sensitivity**: calibration of the interferometer signal (slope of the fitted sine wave) at the working point. • **Interferometer Setpoint**: Interferometer signal value at the working point. This will be used as setpoint for the interferometer controller.
10	Fit length	Defines the width (in points) for the linear fit at the working point to determine the slope (= interferometer sensitivity) at the working point. The linear fit with the corresponding length is drawn in the graph.
11	Navigation buttons	• **back** will return to the first page of the calibration routine (e.g. to adjust the W piezo sweep range) • **cancel** will stop the calibration procedure (no values will be applied) • **accept** will apply the current values to the interferometer controller module.

Back Home

Z-Controller

The z-controller window gives the user access to the PI parameters that control the tip-sample distance.

For a short visual introduction watch the Z-Controller Video Tutorial on the Nanonis homepage.

1	Control Signal	The controller's task is to continuously adjust the Z position of the tip in order to keep the selected control signal as close as possible to the Set Point. Different control signals lead to different controller behavior. For STMs usually log current is used, for AFMs it can be frequency, phase, amplitude etc.
2	Controller on/off switch, Withdraw, Home, Tip Position, TipLift, Off Delay, Controller Status	**Controller on/off switch**: Turns the Z-Controller on or off. **Withdraw button**: Withdraws the tip to the upper end of the piezo range. Set withdraw slew rate in the controller configuration panel. **Home button**: Move the tip to the home position. **Home Absolute (m)/ Home Relative (m):** Defines the home position of the tip. The blue slider also represents this position. The tip will be set to home position if the home button is clicked or if the SafeTip condition is met. Make sure this level is above the sample. The home position can be absolute or relative (a specified value above current position). This can be switched from the controller configuration panel. **TipLift**: Lifts the tip by the specified amount when then controller is switched off. This can be a positive or a negative number, i.e. the tip can also be moved towards the sample. Be careful with sign and value when using this feature. **Switch off Delay**: Use this parameter for a reproducible position when switching off the Z-controller. When >0 and the Z-controller is switched off, it doesn't switch off immediately but continues to run for the specified time and averages the Z position. **Controller status**: Displays the controller status with different colors: - black: off - green: on - yellow: hold - red: SafeTip backdraw **Tip position**: Indicates the tip position Z(m). The tip position can also be varied by using the slider bar when the controller is not running. The slider bar shows as well the actual Z voltage in gray. The range for this value is always +/- 10V and this range doesn't change with the QuickScale buttons. The color will change from gray to red if the Z voltage rises above 90% of the full range.
3	Set Point and Current Value	**Set Point** is the desired value of the control signal. It can be set by editing the number or using the slider bar. Click the "Set" button above the input field to use the actual value as Set Point. The slider shows the Set Point as well as the current value. **Current value**: Indicates the name and current value of the control signal. Important Note: this indicator displays the value of the Z control signal, i.e. if the control signal is abs(Current) or log(Current) it shows the absolute current, not the signed current as in the current window. The controller type (abs, log, bipolar) is set in the controller configuration panel.
4	Controller parameters	Proportional and integral controller settings. The higher these values are set, the faster the controller reacts to deviations from the control signal to the Set Point. If the values are set too high, the controller will start to oscillate. If this happens, lower the controller parameters. The **Type** switches controller parameters between P/I (proportional gain/integral gain) and P/T (proportional gain/time constant). The formula for the controller in the frequency domain is: G(s) = P + I/s = P(1 + 1/(Ts)). The integral gain and time constant are related as follows: I = P/T.
5	SafeTip on/off	**SafeTip on/off**: Turns the SafeTip feature on or off. **SafeTip signal**: Displays name & value of the SafeTip signal. The value is also displayed in the slider bar. Important Note: this indicator displays the value of the SafeTip signal, i.e. if the SafeTip signal is abs(Current) or log(Current) it

shows the absolute current, not the signed current as in the current window. The controller type (abs, log, bipolar) is set in the controller configuration panel.

SafeTip Threshold: If the SafeTip signal meets the condition set by the two controls (signal >/< and threshold value), the tip is moved to the home position (2). The threshold can be edited in the input field or using the slider. To configure SafeTip, use the controller configuration panel.

Autorecovery: Automatically recovers from SafeTip backdraw after a specified amount of time (set in controller configuration) if Z-Controller was originally on.
Pause Scan: Automatically pauses/holds the scan on a SafeTip event. If "AutoRecovery" is on and Z-Controller was originally on, it automatically resumes the scan after re-engaging the Z-controller after a specified amount of time (set in controller configuration).

Tools

·	Controller configuration	Opens a window to configure user-defined Z controllers.
·	Piezos calibration	Opens the Piezo Calibration window to set the piezo characteristics and sample tilt in each dimension.
·	Auto-approach	Performs a coarse auto-approach of the tip to the sample.
·	Aproach-Retract curves	Approaches/retracts the tip to/from the sample and lets you observe different signals.
·	Z spectroscopy	Perform Z spectroscopy measurements like I/Z curves.

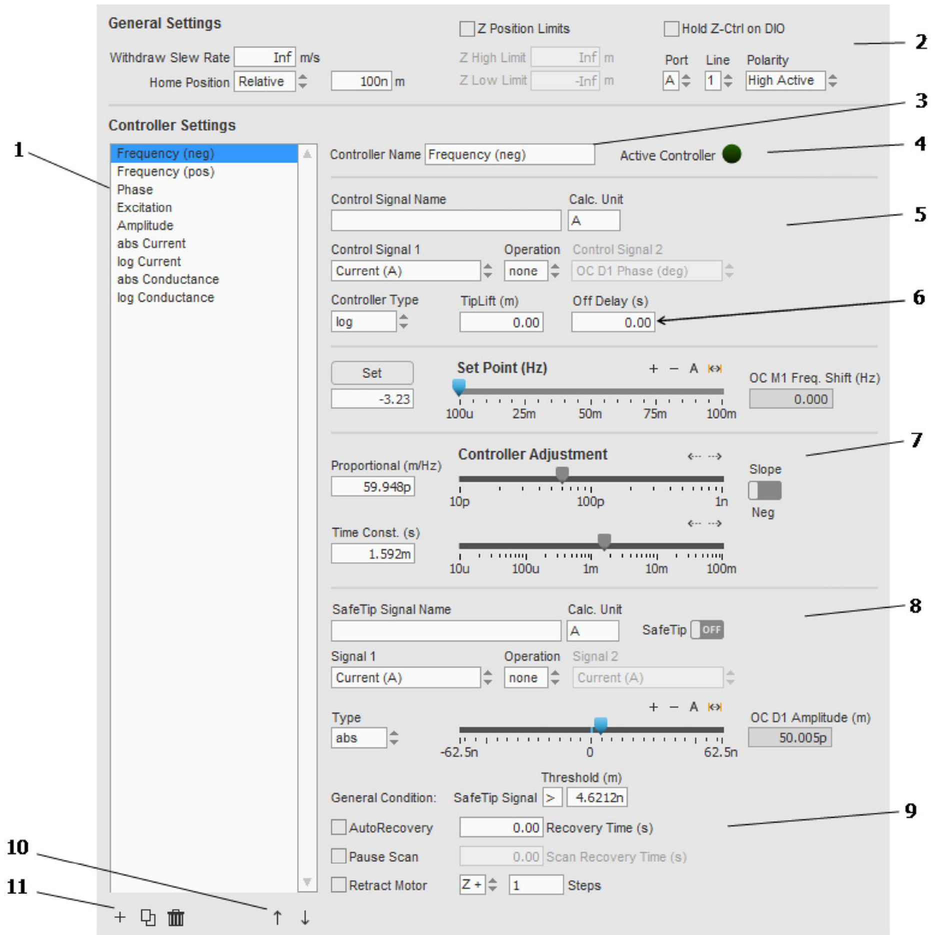
Note: Numeric control handling is explained in the user interface section.

Back Home

Z-Controller Configuration

The Z-controller configuration window lets you define your own Z controllers. The control variable can be an arithmetic operation of two signals, and you can select whether to control absolute values, bipolar values or logarithmic (absolute) values. Additionally, the SafeTip feature (per controller) can be configured here.

Remember to save the settings (in Z-Controller or main window) if you want to keep your changes.



1	Available Controllers	Lists all defined controllers. The values to the right always correspond to the controller selected in this list.
2	General Settings	These settings apply to all controllers: **Withdraw Slew Rate (m/s):** Configures the rate at which the tip withdraws. If set to Inf or 0, it withdraws instantaneously. **Home Position (m):** Configures Home position as absolute or relative (a specified value above current position). **Z Position Limits**: Enables the limitation of Z position. If this is not checked, the default Z high and low limits are set to the Z piezo range defined in the Piezo Calibration module. If this feature is enabled and the Z piezo range changes in the Piezo Calibration module, then the Z high and low limits are scaled accordingly. **Hold Z-Controller on DIO**: Allows to hold the Z-Controller on the configured DIO line.
3	Controller Name	Controller name. This name which will be displayed at places where you can select a controller.
4	Active Controller	Indicates whether the selected controller is currently active (selected in the Z-controller panel) or not. When the active controller is selected, some controls are disabled, e.g. it's not possible to delete the active controller or to change its control value calculation.
5	Controller Properties	In this area you can define the control variable. If you don't need an operation of two signals, set the operation to "none". If you select another operation, Control Signal 2 gets active and you can specify whatever combination you want. The controller type can be **bipolar** (signed values as they are), **abs** (absolute value) or **log** (logarithm of absolute value). The **TipLift** (see Z-Controller) can also be defined here.
6	Switch off Delay	This parameter specifies the behavior of the controller when it is switched off: - **0 (default):** Bumpless switching, i.e. the tip stops right at the position where it stands - **1 real-time cycle**: Integral part, i.e. move the tip to the position of the integral part of the PI controller. This avoids the oscillations of the P part to have a random effect on the stop position. - **>1 real-time cycle**: Averaging over the specified time. When the controller receives a request to switch off, it starts to average the position over the specified amount of time on the RT engine while the controller is still running. Then, the tip is moved to the average value. This procedure doesn't switch off the controller immediately, but it can be useful in cases where a reproducible position is required when turning off the controller.

		To find a good value, you can turn the controller on and off while watching the Z signal in the signal chart. For a value of 10ms..100ms, the position should be quite reproducible.
7	PI Settings	PI settings of the selected controller. See Z-Controller for more information.
8	SafeTip Configuration	Configure the SafeTip signal and condition (Comparison, Threshold) in this section.
9	SafeTip Recovery and Pause Scan settings	**AutoRecovery**: Automatically recovers Z-Controller from SafeTip backdraw after the specified amount of seconds (Recovery time) if Z-Controller was originally on. If AutoRecovery is checked and Z-Controller was originally on, it holds the scan, it waits until Z-controller is engaged again, and it automatically resumes the scan (it continues) after the specified Scan Recovery time (s). **Pause/Hold Scan**: Automatically pauses the scan on a SafeTip event. **Retract Motor**: Automatically retract the motor the specified number of steps.
10	Move up / down	Changes the order of the controllers by moving the selected controller up or down in the list ("1").
11	Delete, Add, Duplicate	Use these buttons to delete the selected controller, add a new controller or duplicate the selected controller.

Back Home

Approach-retract (Z-sweeper)

This module is designed to perform approach-retract experiments by controllably moving the tip along the Z direction.

It can, for example, be used in a typical AFM experiment that consist to measure force-distance curves by recording the cantilever deflection as a function of Z. STM users can also use this module for tip preparation by controllably indenting the tip into the sample.

It is indeed possible to define "retract conditions" that will automatically switch the sweep direction from "approach" to "retract". The direction can also be reversed manually during the sweep.

The sweep is controlled from the host computer (i.e. not from the real-time controller), therefore limiting the acquisition speed to about 50 pt/s.

Note: Generic graph handling and numeric control handling are explained in the user interface section.

1	Control	move the tip to **base position**. Before moving, the Z-controller will first be turned off. start the sweep in the selected **direction**. During the sweep the Z-controller is turned off. stop the sweep. pause the sweep. **Retract/Approach switch**: select in which direction the tip moves.
2	Retract conditions	Set the conditions that will switch the sweep direction from **approach** to **retract**. One condition per channel can be set. When at least one condition is met, the tip starts to retract until it reaches the **base**

		position. **Signal**: select on which channel the current condition applies. **Check-box**: activate the condition on the selected channel. **Value**: take the "original" or "absolute" value of the signal. **Comparison**: select if the condition is met when the signal is higher or lower than the threshold. **Threshold**: level of the signal that will trigger the condition.
3	Position	**Z**: current Z position of the tip. **Base**: Define the Z position at which the sweep ends when moving in the **retract** direction. It also defines the position where the tip goes when clicking on the button.
4	Channels	Select the channels you want to record. Tip: you can select more than one channel by pressing the Ctrl-key while selecting the channels with the mouse.
5	Steps size	**Steps Size**: Define the step size during the Z sweep. **Measurements Period**: Defines the measurement speed. Higher values result in more accurate data because of the oversampling. This control is the same as the one in TCP receiver.
6	Display & Save	**Plot**: select to which plot (buffer) the next sweep will be saved to. show/hide the plot curve on the graphs. save the data to ASCII file (.dat). clear the plot. Save options: **Dialog**: when selected, a dialog window will open before saving data. **Auto save**: when selected, the data are automatically saved after each sweep. **Basename**: defines the file basename when saving sweeps. The file names will be "BasenameNNN.dat" where NNN is an automatically increasing number.
7	Signal to display	Choose the signal you want to display in the graph.
8	Cursor	Display the cursor coordinates. The cursor can be hidden/shown by right-clicking on the graph and selecting Cursor>Show. The cursor can be moved from one curve to the other by dragging its crosshair. Note: Generic graph handling including cursor handling is explained in the user interface section.

Back Home

Automatic Coarse Approach

The coarse approach approaches sample and tip by using the Z-Controller and coarse movements (motor) in Z.

When starting the approach it will activate the z-controller and monitor the z-position. The procedure depends on the approach mode which can be selected in the advanced configuration section.

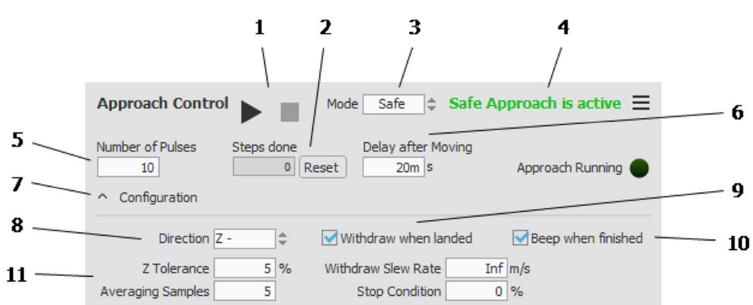
Safe Approach Mode: when the tip reaches the bottom end it is withdrawn, approached by one or more coarse step(s), and then the Z-Controller is restarted after a short delay. When it detects a reverse movement of the Z position and the Stop Condition is found, as it happens when the sample is reached, the auto approach stops.

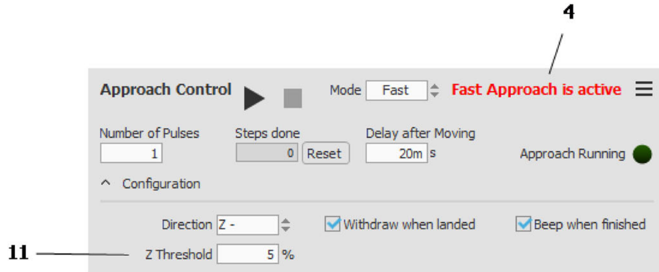
At the bottom of this page, you will find a comprehensive timing diagram. It explains all the timing settings of the Safe Approach mode.

Fast Approach Mode: when the tip reaches the bottom end, it is not withdrawn, but the motor starts to move until the tip is lifted above the specified Z threshold by the Z-Controller.

In Safe mode, if the auto approach stops after every step, try setting the P gain of the Z-Controller to 0 (in P/I controller mode) or increase the set point or modify the advanced auto approach settings.

Use the Motor Control panel for manual coarse positioning or to modify the pulse settings.

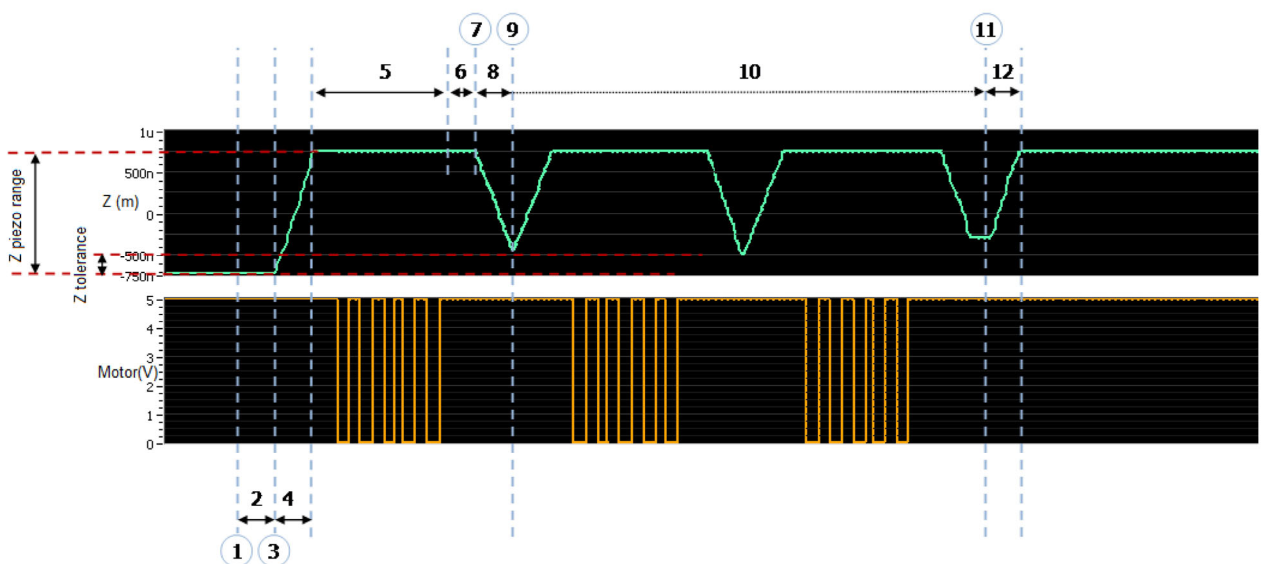




1	Start, Stop Approach	Starts/Stops the automatic approach process. When using a Nanonis HVA4 with gain readout and HV Z is not enabled, a pop-up window shows up to continue or cancel the approach.
2	**Steps done**, **Reset**	**Steps done**: Number of motor steps done in approach procedure. **Reset**: Resets the motor **Steps done** counter.
3	Approach **Mode**	Set the approach mode: Safe approach which executes woodpecker approach, and Fast approach which does not withdraw the tip when moving forward.
4	Status Indicator	Shows the approach mode which is currently activated. It's red when Fast approach is activated and green when Safe approach is activated.
5	**Number of Pulses**	Number of motor steps per approach cycle. After the tip is withdrawn (see mechanism description on top of page), this number of coarse steps is made.
6	**Delay after Moving** (s)	Delay after moving the motor before turning on the Z-controller.
7	Show advanced Configuration	Show/hide the advanced configuration settings below.
8	Approach **Direction**	Coarse positioning direction to approach.
9	**Withdraw when landed**	Withdraw the tip when landed.
10	**Beep when finished**	Lets your computer beep (only if there's a speaker) when the approach process is finished. If enabled, a button to stop the beeping is displayed below.
11	Advanced Configuration Section	**Z Tolerance** (%): Withdraw & drive motor when tip is within specified tolerance (in % of the piezo range) above minimum piezo range. The default value is 5%. **Withdraw Slew Rate** (m/s): The tip withdraws at the specified rate. If this value is set to Inf or 0, it withdraws instantaneously. This parameter is configured from the Z-Controller configuration panel too. **Averaging Samples:** Average the last specified samples of Z when checking Stop condition. The default value is 5 samples to average. **Stop Condition** (%): Stop the approach when the Z-controller causes the tip to be lifted by the specified % of the piezo range from the averaged position. The default value is 0%. **Z Threshold** (%): Stop approach when the Z-controller causes the tip to be lifted above the specified value from the lower limit (in % of the piezo range). The default value is 5%.

Safe Approach Timing Diagram

The two graphs below display the Z position and the digital pulse line used to move the motor versus time:



1. Start. The Z-controller is switched ON. In this example, the Z-Controller was already ON when the auto-approach starts (tip at the piezo range low limit).
2. **Z Tolerance** and **Stop Condition** are checked.

3. The tip is within the % of the piezo range above the piezo range low limit specified by **Z Tolerance**, the Z-Controller is switched OFF, and the tip is withdrawn to the piezo range high limit.
4. The tip withdraws at the **Withdraw Slew Rate** (m/s).
5. The motor moves the **Number of Pulses** specified in the auto-approach module at the frequency set in the motor module.
6. When the coarse motion finishes, wait the time specified by **Delay after Moving** (s).
7. The Z-Controller is switched ON.
8. Z ramps starting from the piezo range high limit until **Z Tolerance** or **Stop Condition** are met.
9-10. In this example, the tip is within the % of the piezo range above the piezo range low limit specified by **Z Tolerance,** and the whole sequence is repeated from steps 2 to 8 until Stop condition is met.
11. **Stop Condition** is met. The tip is lifted the % of the piezo range from the averaged position specified by the **Stop Condition** (0% in this example).
12. If **Withdraw when landed** is checked, the tip withdraws to the piezo range high limit.

Back Home

Automatic coarse approach (using Generic PI controller)

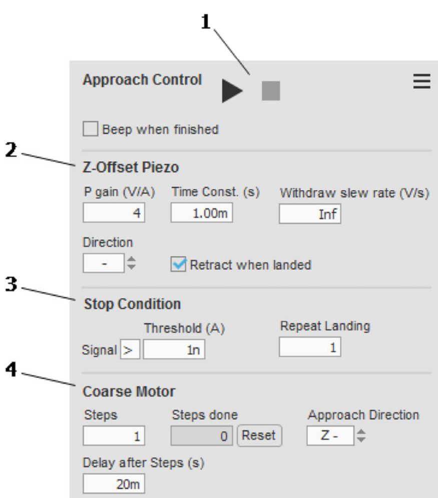
This module performs an automatic approach to bring the tip in contact to the sample.
Contrary to the regular Auto-approach module, this module does not use the Z-controller to ramp the Z-piezo but the Generic controller instead.

It is then possible to **control two Z-piezos**: one for scanning (controlled by the **Z-controller**) and one for approach or offset (controlled by the **Generic controller** through this module).
To change the position (voltage) of the offset piezo after the approach, use the slider or the input field of the Generic controller.

Operation principles:
- The module uses the **Input signal** configured in the Generic controller as the controlled signal. The output voltage that controls the piezo offset is available on Output 2.
- The output voltage is ramped linearly until one of those two stop conditions arises:
    (1) **Max voltage** is reached, |U|=Umax.
    (2) Controlled signal reaches the stop condition, i.e. |Signal| >/< Threshold.

    - if (1) arises (i.e. no contact found), the piezo is withdrawn and the coarse motor is stepped. Then the ramp procedure starts again.
    - if (2) arises (i.e. contact found), the ramp + stop condition is performed **Repeat landing times**. If all the attempts proved to be successful, the approach is over.

Configuration:
- Configure the **Output signal** (name, unit, calibration, limits) which is used to drive the piezo in the Generic controller.
- Configure the **Input signal** in the Generic controller.
- Use the Generic controller to find good controller parameters for the PI controller (P gain and time constant).
- Copy P gain and time constant values to auto approach module
- Configure other settings (**Withdraw slew rate**, **Stop condition**, **Coarse motor**) in the auto approach module.



1	Start approach	Start/stop the approach procedure.
2	Z-piezo offset	- **P gain**: define the ramp speed of the piezo by controlling the P gain of the Generic controller. - **Time constant**: time constant of the piezo control loop. This will be set in the Generic controller. - **Direction**: select the piezo orientation that corresponds to the approach direction. - **Withdraw slew rate**: defines the speed for withdrawing the piezo when not landed before doing motor steps. - **Retract when landed**: when checked, the piezo is withdrawn after successful approach process (when landed).
3	Stop condition	- **Condition**: specifies the comparison operator to check the signal () with the threshold (> or <). - **Threshold**: signal value (of the **Input Signal** defined in the Generic controller) that needs to be reached to stop the voltage ramp. - **Repeat landing**: define how many successful landings must be done to say that the approach is over.
4	Coarse motor	- **Steps number**: define how many steps to perform with the motor after no contact has been found. - **Delay after steps**: time to wait after stepping. - **Approach direction**: select the motor orientation that corresponds to the approach direction

---

Back Home

Z spectroscopy

The Z spectroscopy module lets you perform fast Z sweeps while measuring arbitrary channels. Usually, it is used to record I(Z) curves.

Unlike the Approach-retract sweep, the spectroscopy curves are performed on the real-time engine.

A sweep can be started at any time using the "Start" button. If you start a sweep while scanning, the scan is paused for the duration of the spectroscopy measurement. The Z-Controller is set to hold (deactivated) for this time, so you don't have to turn it off manually.

The module holds the data of 10 sweeps in its buffer, and lets you save the sweeps manually or automatically after they have been recorded. It displays all spectroscopy curves acquired, whether it is started from this panel or any other experiment that performs Z spectroscopy measurements.

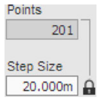
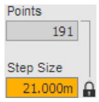
When saving a Z spectroscopy curve, the saved file will always contain a data column "Z rel (m)". This is the Z position of each point acquired relative to the initial averaged controlled Z position, Z_{aver}.

At the bottom of this page, you will find a comprehensive timing diagram. It explains all the timing settings of the *Z spectroscopy* module.

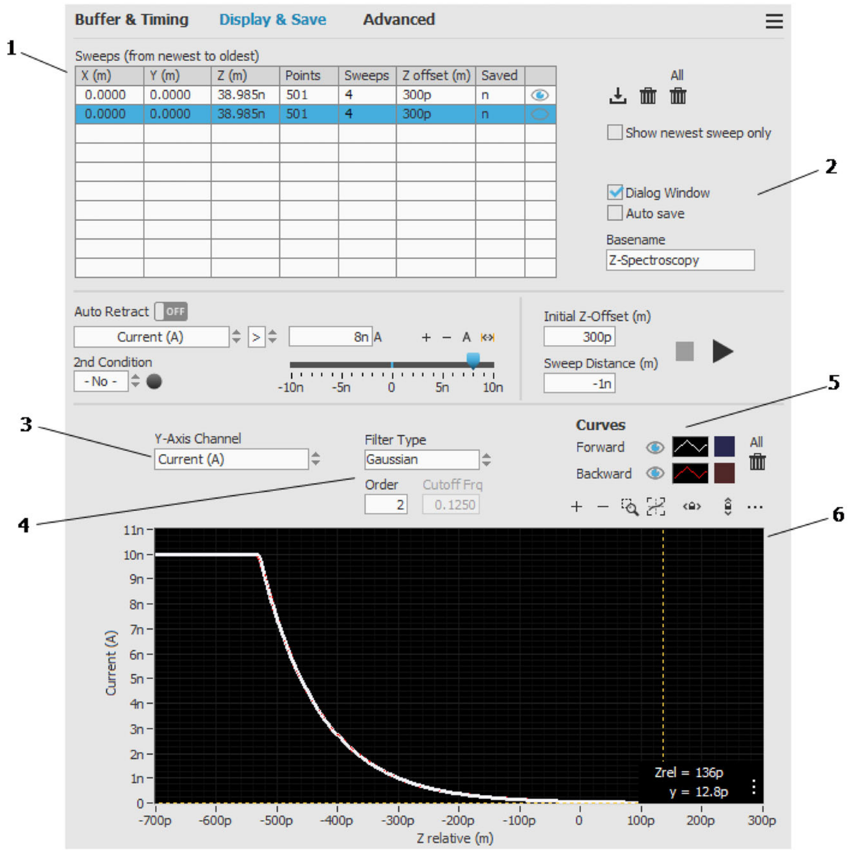
Note: Generic graph handling and numeric control handling are explained in the user interface section.

I) Buffer & Timing tab

1	Points Step Size Lock	When defining how many data points to record during a spectroscopy measurement, both **Points** and **Step Size** can be entered directly and the other is automatically calculated using the entered value (either **Points** or **Step Size**) and **Sweep Distance (m)**. In order to be able to reliably react to changes to **Sweep Distance (m)**, either **Points** or **Step Size** is declared as a "master" parameter. Here the **Points** parameter was entered manually and is now defined as "master". Changes to **Sweep Distance (m)** in this

		state will automatically recalculate the **Step Size** parameter.  Here the **Step Size** parameter was entered manually and is now defined as "master". Changes to **Sweep Distance (m)** in this state will automatically recalculate the **Points** parameter.  If an entered value for **Step Size** results in a non-integer **Points** parameter, the **Step Size** parameter is highlighted with an associated tipstrip that with the current settings, the actual last Z value for the spectroscopy measurement does not exactly match the specified **Initial Z.-Offset** + **Sweep Distance**.
2	Acquisition	**Channels**: select the channels you want to record. **Points**: number of points per curve to record. **Backward sweep**: select whether to measure the backward (return) sweep or the forward only. **Save all**: when selected, the data from the individual sweeps is saved along with the average data of all of them.
3	Sweeps	Number of Sweeps to measure and average
4	Initialization	**Z averaging time**: duration over which the Z position is recorded and averaged before and after the sweep (the latter only if **Record final Z position** is selected in the Advanced section). After the initial averaging time, if **Z-Controller to Hold** is selected in the Advanced section, the Z-Controller is set to hold and the tip is placed at the averaged position Z_{aver} (plus **Z offset)**. **Initial settling time**: additional time to wait (in addition to the Settling Time) after the tip has moved to the starting point of the sweep. Adjust this parameter to allow the system to settle down after **Initial Z-offset** has been applied and right before starting to sweep.
5	Sweep timing	**Max. slew rate**: maximum rate at which the tip position changes (before, during and after sweeping). **Settling time**: time to wait between moving the tip to a new Z level and starting to acquire data. Adjust this parameter to allow the system to settle down after each step and reduce transient effects. **Integration time**: time during which the data are acquired and averaged.
6	End of sweep:	**End settling time**: time to wait after the sweep has finished and the tip has moved back to its initial position Z_{aver}. **Z control time**: time during which the Z-Controller is enabled once a sweep has finished. When averaging multiple sweeps (i.e. **Sweeps**>1), the Z-Controller is enabled for this duration between each sweep. After the last sweep, it will wait the specified time before continuing a running scan. This insures each sweep reliably starts from the same position. **Sweep time**: time taken to acquire one sweep in one direction. If *backward sweep* is enabled, it will take twice as long. This value only indicates the time needed for the actual sweep; *Z averaging time* and *Z control time* are not included.
7	Auto retract	Configure a local SafeTip condition to prevent the tip from crashing. The auto-retract signal can be the SafeTip signal defined for the Z-Controller in the Z-Controller configuration panel, or it can be any signal. **Auto retract on/off:** enable/disable the auto-retract condition. **Signal selector:** the auto-retract signal can be the SafeTip signal defined for the Z-Controller in the Z-Controller configuration panel, or it can be any of the other channels available on the Nanonis Control System. **Comparison + Threshold:** define which condition triggers the auto-retract of the tip. **Slide**: the bar indicates the current auto-retract signal level. The slider corresponds to the *Threshold* value. **2nd Condition**: configures the use of a second signal comparison, configured in the Advanced Section, in combination with the main signal comparison configured above: -No-: disables the use of a second signal comparison. OR: Auto-Retract will execute if the 1^{st} or the 2^{nd} condition is met. AND: Auto-Retract will execute if the 1^{st} and the 2^{nd} condition are met at the same time. THEN: the 2^{nd} condition is only checked once the 1^{st} condition has been met.
8	Limits (m)	**Initial Z-offset**: offset added to the initial averaged position Z_{aver} before starting to sweep. Note: the **TipLift** offset, defined in the Z-Controller, is not applied during a spectroscopy since the Z-Controller is not turned off but set to hold. **Sweep distance**: distance the tip will travel during the sweep, starting from the "Z_{aver} + *Initial Z-offset*" position. Note: for both Z-*offset* and *Sweep distance*, the sign defines the direction: positive value means away from the sample, negative means toward.
9	Control buttons	Start/stop the spectroscopy measurement.

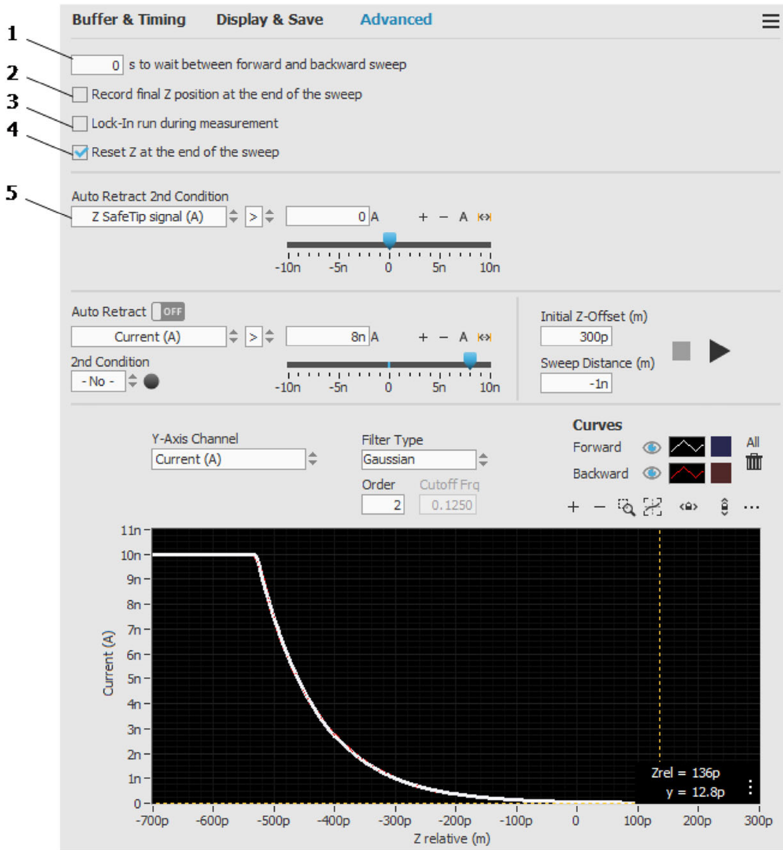
II) Display & Save



1	Sweeps list	List the last 10 acquired sweeps. The *X*, *Y* and *Z* columns indicate the tip position before starting the experiment. **Sweeps** initializes to 0 when starting the experiment and it shows the number of completed sweeps when the experiment finishes. If it is manually stopped and the total number of sweeps is not reached, it shows the number of completed sweeps in red. **Saved** indicates if the selected curve has already been saved into file (n=no, y=yes). Show newest sweep only activates display of the last measured curve. The checkmarks to the right of the table allow to show/hide the corresponding plots in the graph. · save the selected sweeps · delete the selected sweeps All delete all sweeps *Tips:* *A menu appears when right-clicking on the list.* *Double-clicking on the title bar select all the sweeps.* *When the sweep selected in the Sweeps list has been saved, the full file path appears when hovering the mouse over the list. If several are selected, only the path of the first selected sweep appears.*
2	Save options	**Dialog window**: when selected, a dialog window will open before saving data. **Auto save**: when selected, the data are automatically saved after each sweep. **Basename:** defines the file basename when saving sweeps. The file names will be "*Basename*NNN.dat" where NNN is an automatically increasing number. *Tip:* *When the sweep selected in the Sweeps list has been saved, the full file path appears when hovering the mouse over the list. If several are selected, only the path of the first selected sweep appears.*
3	Signal to display	Select the signal you want to display in the graph.
4	Data filtering	**Filter type**: select a filter to smooth the displayed data. Note: When saving, if any filter selected, filtered data are saved to file along with the unfiltered data. Some extra parameters are available for some filters: **Order**: filter order of a dynamic filter or width (in number of points) for the gaussian filter. **Cutoff frq**: cutoff frequency for dynamic filters. This assumes the acquired data have a sampling frequency of 1 Hz, regardless of the sweep timing. This parameter is very important when filtering as it defines how "strong" the filter is.
5	Curves	Define the curves display properties: **Checkmark**: show/hide the forward and/or backward plots. **Color box**: select the color of the plot #10. The colors of plots 2-9 are interpolated between the first and last plot colors.
6	Cursor	Display the cursor coordinates. The cursor can be hidden/shown by right-clicking on the graph and selecting Cursor>Show. The cursor can be moved from one curve to the other by dragging its crosshair.

		Note: Generic graph handling including cursor handling is explained in the user interface section.

III) Advanced



1	Delay (s)	Time to wait in seconds between the forward and the backward sweep.
2	Record final Z	Select whether to record the Z position during **Z averaging time** at the end of the sweep (after **Z control time**) and store the average value in the header of the file when saving. Using this option increases the sweep time by **Z averaging time**.
3	Lock-In run	Select whether to set the Lock-In to run during the measurement. When using this feature, make sure the Lock-In is configured correctly and settling times are set to twice the Lock-In period at least. This option is ignored when Lock-In is already running.
4	Reset Z	Reset the Z position at the end of the sweep to the initial averaged value calculated during **Z averaging time** at **Max slew rate**. If not selected, the Z position stays at the last value of the sweep (at least during **Settling time** + **End Settling time** if the Z controller was switched on before starting the sweep).
5	Auto Retract 2nd Condition	Configuration of the 2nd condition to meet (in combination with the main one) to Auto Retract. **Signal selector:** the auto-retract signal can be the SafeTip signal defined for the Z-Controller in the Z-Controller configuration panel, or it can be any of the other channels available on the Nanonis Control System. **Comparison + Threshold:** define which condition triggers the auto-retract of the tip. **Slide**: the bar indicates the current auto-retract signal level. The slider corresponds to the *Threshold* value for the 2nd condition.

IV) Timing diagram

The two graphs below display the Z position versus time:
the upper graph shows the full area covering the whole sweep.
the lower graph shows a zoom around the controlled Z position.

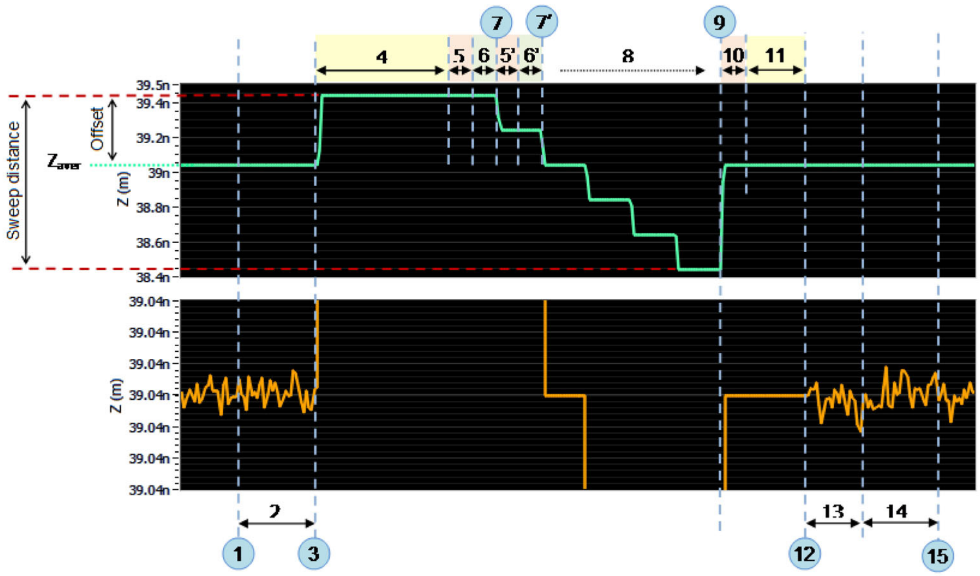


Figure: Z spectroscopy timing diagram for one forward sweep.

1. Start. If the tip is scanning, the scan is paused.
2. Measure the Z position during **Z averaging time** => Original averaged position, Z_{aver}, is calculated.
3. Set Z-Controller to hold (i.e. disable it during the sweep); ramp Z to "Z_{aver} + **Initial Z-offset**" at the **Max. slew rate**; set Lock-In to run (i.e. enable it during the sweep if the *Lock-In run* checkbox is selected).
4. Wait **Initial *settling time*** to let the system settle after having changed the Z position in phase 3.
5. Before acquiring data, wait **Settling time** to let the system settle on the new Z level. This value is usually shorter than **Initial *settling time*** because the Z steps during the experiment are quite small.
6. Acquire data of the selected channels during **Integration time**. The averaged measurement will be the recorded value for this Z position.
7. Ramp to next Z level at the **Max. slew rate**. The step size depends on the sweep distance and the number of steps.
5'. Wait **Settling time** (same as phase 5).
6'. Acquire data (same as phase 6).
7'. Ramp to next Z level (same as phase 7).
8. Repeat steps 5-7 until the *Sweep distance* is reached or the retract condition is met.
9. Sweep (data acquisition) finished; ramp Z back to its initial value, Z_{aver}, at the **Max. slew rate**, unless the flag to reset Z (in the advanced section) is unchecked, so that the Z position remains at the last value of the sweep.
Switch off Lock-In (if specifically set to run during sweep).
10. Wait **Settling time** (same as phase 5).
11. Wait **End *settling time***.
12. Switch Z-Controller to original on/off state.
13. Wait **Z control time** to let Z position settle.
14. Measure the Z position during **Z averaging time** => Final averaged position is calculated.
15. Resume scan, if the tip was scanning before starting the spectroscopy.

Note: on the left side of the graph, the green dotted line indicates the averaged controlled Z position, Z_{aver}. The "Z rel (m)" channel that is saved in each spectroscopy data file gives the relative Z position with respect to this level.

Back Home

Piezo Calibration

The Piezo Calibration module is used to define the X Y Z piezos calibration. It also contains other piezo-related settings like sample tilt and advanced functions like the Drift Compensation, 2nd order correction, Hysteresis Compensation and Voltage Limits for the scan piezos. It also allows compensating for a non-orthogonal scanner coordinate system.

From the **Settings** menu you can load and save the configuration. Don't forget to save after editing the calibration values.

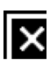
Calibration

This tab is used to set the calibrations for all 3 piezos and select the active calibration. This is useful to store calibrations for different high voltage amplifier gains or temperatures.
CAUTION: retracting the tip a few steps before changing the active calibration or the values of the active calibration is highly recommended.

1	Calibration list	Display the list of piezo calibrations. : add a new calibration. : duplicate the selected calibration. : delete the selected calibration. : move up/down the selected calibration. The **Apply** button activates (apply) the selected calibration. Notes: - It is possible to rename the calibration by clicking directly on the list item. - The tick on the left side of the list indicates the active calibration.
2	Calibration values	Displays the calibration for all three piezos and allows changing them. There are 3 controls available: **Calibr. (m/V)**, **Range (m)** and **HV gain**. Two of them are enough to define the calibration, so when changing any value one of the other values will be updated automatically. For example, changing the **HV gain** will update the **Range**. For some systems HV gain readout is available, so for those systems the HV gain should be adjusted automatically when the gain is changed at the high voltage amplifier. Note: the **Calibr.** values can be either positive or negative. Changing the sign reverse the piezo orientation.

Z-correction

This tab is used to set the sample tilt (in degrees, first order correction) and a curvature (can be set approximately to the length of the piezotube). The tilt can also be accessed in scan control.



Drift Compensation

This tab is used to compensate for the thermal drift your system might have. Once the drift correction for one axis accumulates 10% of the full range, it'll stop compensating for this axis and the corresponding **Saturation** LED will light up. If this is the case and you still want to compensate further, switch drift compensation off and on again.

The drift compensation will basically adjust (move) the coordinate system center position. When switching the drift compensation off the software will optionally (through the **Offset** button) add the drift correction to the X,Y,Z positions and reset the compensation to 0 (i.e. the tip does not jump, but the coordinate system is reset and thus the X,Y,Z positions change).

1	Compensation	Use the button to turn on/off the drift compensation
2	Offset	Add the accumulated drift vector to the scan frame center. The **Offset** button becomes active when switching off the drift **compensation**. When switching the drift **compensation** on and off several times and the **Offset** button is not used, the accumulated drift is stored and only applied when the **Offset** button is switched off (then the accumulated drift is zeroed).
3	Speed	Define the drift speed for all three axes. When the **compensation** is on, the piezos will start to move at that speed.
4	Saturation status	The saturation LED will turn on, if the compensated distance reaches 10% of the axis full range. When saturated, the compensation will stop for that axis. To reactivate the drift **compensation** you have to switch the button off and on again.

2nd order correction (optional)

If you know them, you can enter the 2nd order piezo characteristics to compensate for it. The following equation shows the interpretation of the 2nd order correction parameter:

For the X-piezo:
$U_x = 1/c_x \cdot X + c_{xx} \cdot X^2$

with units:
$[V] = [V/m] \cdot [m] + [V/m^2] \cdot [m^2]$

where c_x is the calibration of the piezo X and c_{xx} is the 2nd order correction.

Non-orthogonal scanner (optional)

If the x and y axes of your scanner are not orthogonal, an angle can be specified here to compensate for this. The angle is the deviation from the orthogonal coordinate system, i.e. the deviation from 90° between x and y axes. 0 (default) means orthogonal system.

1	Y angle	Deviation from 90° of the angle between the scanner x and y axes. 20 deg for instance means the angle between x and y is 110°.

Hysteresis compensation (optional)

Some scanners show a significant hysteresis of the displacement vs. applied voltage. This results in a distorted image (stretched at the beginning of the scan lines, compressed towards the end of the scan lines).
This hysteresis effect can be compensated using a curve with compensation values. The compensation values are added to the piezo voltages in order to compensate for the hysteresis.

Use the Hysteresis Compensation Editor to create or modify the compensation curves. The compensation values are scaled with the actual scan frame size.

To determine the compensation curve we recommend scanning a calibration grid with the maximum scan range (so the full output voltage range is used).
Since the calibration grid will only have a few grid lines in the scan range this will result in only a few points to determine the required compensation voltage. These points can be used to get started in the Hysteresis Compensation Editor, where spline interpolation will be used to create a curve through the specified points.
Then this first estimate of the compensation values can be used to scan a surface with smaller structures. The compensation values can then be adjusted so the scaling of the structures is more or less equal over the whole scan range.
One possibility is to freeze the slow scan axis and compare the forward and backward scan lines in a line scan monitor. Adjust the compensation values until the forward and backward lines match.

1	Hysteresis compensation on/off	Switches the hysteresis compensation on or off. When off, no correction voltages are applied to the piezo voltages even if compensation curves are defined.
2	Hysteresis Editor	Open the Hysteresis Compensation Editor used to create or adjust compensation curves.
3	Compensation value	Displays the compensation voltages for the fast and slow scan axes. The abscissa is the full low voltage piezo range (-10..10V) while the ordinate shows the compensation voltages. Use the Hysteresis Compensation Editor to modify the curves. The applied correction voltages are scaled with the actual scan frame size.

Piezo voltage limitation

The voltage applied to the scan piezos should not exceed the limits specified by the piezo manufacturer. The first choice to have the correct voltage range at the piezos is to use a matching high-voltage amplifier gain. If the correct gain is not available this tab allows setting limits for the low voltage scan outputs of the SC5. E.g. if the HV gain is 15 but the maximum piezo voltage is ±120V one can limit the SC5 scan outputs to 8V (15 * 8V = 120V).
Unlike most other parameters, changes to the piezo voltage limits are saved immediately in a system configuration file (load/save settings does not apply to the voltage limits).

1	Enable voltage limits	When ticked the voltage limits are applied for the corresponding SC5 outputs.
2	Voltage limits	Voltage limits for the X, Y and Z piezos. For a bipolar output the output range is [-Lim, Lim], for a unipolar output the range is [0, Lim].
3	Actual output range	Displays the actual output range, taking into account the enable switch (“1”), the limits (“2”) and the output polarity set by jumpers inside the SC5.

Back Home

Hysteresis Compensation Editor

The Hysteresis Compensation Editor is used to define configure the hysteresis compensation look-up tables (LUT) for the fast and the slow scan axes.
It uses a spline fit through user defined points to create the curve. In addition to the points, the slopes at the beginning and at the end of the curve can be specified.
The compensation value at the beginning and end of the curve are fixed to 0. This is to prevent voltage jumps. It might be that you have to adapt the linear calibration in the Piezo Calibration module slightly when using the hysteresis compensation feature.

To save compensation curves you have to apply the changes so they are reflected in the Piezo Calibration module, then save settings there.

Also in the Piezo Calibration module there’s a recommendation on how to determine the compensation values.

1	Scan Axis	Select which scan axis (fast or slow) should be edited.
2	Display Range	Selects the voltage range to display in the **graph** (“5”). This doesn’t affect the compensation values; this is just for display purpose.
3	Scale Curve	Use this slider to amplify or attenuate the compensation values. The shape of the curve stays the same, it is just scaled.
4	Scan Range Scaling	Expand the window to show the Scan Range Scaling section. This allows arbitrary scaling of the hysteresis compensation values with the scan range (instead of the standard linear scaling).
5	Compensation curve graph	Displays the compensation curve for the selected **scan axis** (“1”). Move the cursors using the mouse or edit the coordinates in the **cursor list** (“6”) to adjust the curve. The curve shows the compensation values for a scan over the full range (-10..10 V on the low voltage side). The curve is scaled to the actual scan frame size.
6	Cursor list	The first two cursors (red) are used to define the slope of the compensation curve at the beginning and the end of the curve. The other cursors specify points which the curve has to cross. Use the **add/remove buttons** (“7”) to add or remove cursors. Usually 1 to 4 points (in addition to the slopes) should be sufficient to generate an accurate compensation curve. If you measured points (using a calibration grid) you can enter the values directly in the list. Just click the X or Y coordinates of any point to

		modify them.
7	Add/remove cursor	If you need more points to design your compensation curve, click the button to add a cursor. To delete a cursor, click it in the graph so it is marked with a special cursor icon (see cursor at 0.6538 V in the graph above), then click the button to delete this cursor.
8	Apply	Use this button to apply the changes. When editing the curves no changes are applied automatically. Click this button to apply the changes so the modified curves become active. You can always see the active compensation curves in the Piezo Calibration module.

Back Home

Lock-In

The Lock-in module works similar to an external lock-in. It allows you to modulate (add a sine signal of any frequency (up to 40 kHz) and amplitude) any signal and demodulate any other signal at two selectable harmonics (2 demodulators). It returns the amplitude and phase or Lock-in X,Y components of the demodulated signal. For SPM applications, this can be used for example to record dI/dV curves (when using together with the bias sweep or bias spectroscopy module).
The Lock-in module provides configurable filters, allowing the best suited filtering options for any measurement.
A high-pass filter can be used on the demodulation signal (before the demodulator). The high-pass filter can effectively remove a DC component from the demodulation signal. Choose the cut-off to not suppress the desired component but remove the DC or possible low-frequency interfering signals.
A low-pass filter can be used on the demodulated signals (X,Y). Choose filter order and cut-off to remove undesired components but still having sufficient bandwidth on the demodulated signals. The time constant of a first order low-pass filter is given by $T_1 = 1/(2*pi*fc)$ with fc being the cut-off frequency. For a higher order filter the time constant is approximately $T_n = n*T_1$ with n being the filter order and T_1 the first order filter time constant.
A sync filter can be used on the demodulated signals (X,Y). This filter averages the X,Y signals over one period of the base frequency (harmonic 1) and only outputs a new value after each period. This removes harmonic components very effectively, but has poor suppression of interfering signals and poor dynamic reserve.
Each of the filters can be switched on or off for each demodulator, allowing high flexibility.

For a short visual introduction how to setup the digital Lock-in and record I/V and dI/dV curves watch this Video Tutorial on the Nanonis homepage (it is for V4, but the concept is still the same).

8 signals on the real-time system are reserved for the Lock-in. You can choose freely which of the 8 available demodulated signals you want to assign to any of the 24 channels for data acquisition in the Signals Manager. The available signals are X (in phase component of demodulated signal), Y (out-of-phase component), R (amplitude), phi (phase of demodulated signal with respect to reference phase) of both first and 2nd demodulator.

1	Lock-in on/off	Switch the lock-in modulation on or off. When changing the modulated signal (“2”), the lock-in modulation is automatically switched off (the selected amplitude would probably be way off for the newly selected signal).
2	Modulated signal	This is the signal on which the modulation (sine) will be added.
3	Amplitude	Sets the amplitude (in physical units of the modulated signal) of the sine modulation.
4	Frequency	Sets the frequency of the sine modulation added to the signal to modulate. The frequency range is from 10 mHz to 40 kHz (corresponding to the SC5 output filter cut-off). When working with harmonics, make sure the harmonic frequencies don’t exceed ~100 kHz (SC5 input filter cut-off). Remember the SC5 hardware filters will affect the amplitude when getting close to their cut-off frequencies.
5	Demodulated signal	This is the signal which will be demodulated, in order to determine the amplitude and phase at the frequency set in the **Frequency** field (“4”) or harmonics.
6	Harmonic	Order of the harmonic oscillation to detect on the demodulated signal (with respect to the modulation frequency). If you work with higher harmonics make sure the frequency does not exceed the analog signal bandwidth (100 kHz). The highest harmonic which can be demodulated is the 63rd, or 63x the fundamental frequency of the modulation.
7	Reference phase	Reference phase for the sine on the demodulated signal with respect to the modulation signal. The determined phase (phi) is displayed with respect to the reference phase. - **Auto Phase**: sets the reference phase such that the phase shift is zero. - **+90** / **-90**: shift the reference phase by +/- 90 degrees.
8	Filter settings	Configures the filters for the demodulator. - **HP Order** / **Cut-off**: Order and cut-off frequency of the high-pass filter applied on the demodulation signal. This is used mainly to suppress a DC component of the demodulation signal, which would lead to a component at modulation frequency in the demodulated signals. - **LP Order** / **Cut-off**: Order and cut-off frequency of the low-pass filter applied on the demodulated signals (X,Y). Reducing the bandwidth or increasing the order reduces noise on the demodulated signals, but will require longer settling and measurement times. - **Sync Filter**: Switch the Sync filter on the demodulated signals (X,Y) on or off. The sync filter operates after the low-pass filter and averages the demodulated signals over one period of the modulation frequency.
9	Signals display	Select the Lock-in signals to display in the indicators below (“10”). This is purely a display switch and does not affect the Lock-in operation or signals available in other modules.
10	Indicators for Lock-in signals	Displays the values of the signals selected in the **Signals display** (“9”).

11	Transfer function	Opens the frequency sweeper (logarithmic scale) for the lock-in to record a transfer function.

Back Home

Lock-In Frequency Sweeper (Transfer function)

This module lets you sweep the frequency of a lock-in frequency generator and record the amplitude and phase (or X and Y components) of the demodulated signals. This technique can be used to measure the transfer function of a closed-loop controller. For instance, a controller transfer function can be measured by modulating the input (controller input) and demodulating the output (controller output signal).

The sweep is controlled from the host computer (i.e. not from the real-time controller), therefore limiting the acquisition speed to about 50 pt/s.

1	Frequency Selector	Select which frequency generator to sweep. Be aware that the module will sweep the generator frequency. Several modulators or demodulators might use this frequency generator (or harmonics thereof), and all of them will be affected by the sweep.
2	Control	start a sweep from the current frequency to the **Upper** limit. start a sweep from the current frequency to the **Lower** limit stop the sweep.
3	Frequency (Hz)	The digital control and the slider can be used to manually set the lock-in frequency. They are coupled with the controls on the Lock-in module. The blue fill bar represents the sweep range defined by the **limits** (“4”).
4	Limits (Hz)	Define the **lower** and **upper** limits of the frequency sweep. The **lower** value must be smaller than the **upper** one. Otherwise, the fields will blink and it won’t be possible to start a sweep. Right-click the input fields for more options, like setting the current frequency to the lower or upper limit.
5	Number of steps	Define how many frequency steps are in the sweep.
6	Settling time	Time to wait before acquiring data at each point of the sweep.
7	Integration time	Integration time for the measurements given in number of periods. It defines the measurement speed by setting the value of the **measurement period** (see TCP receiver). Higher values result in more accurate data because of the oversampling.
8	Save options	- **Dialog Window**: when selected, a dialog window will open before saving data. - **Auto save**: when selected, the data are automatically saved after each sweep. - **Basename**: defines the file basename when saving sweeps. The file names will be "**Basename**NNN.dat" where NNN is an automatically increasing number.
9	Plot list	**Plot**: select to which plot (buffer) the next sweep will be saved to. · show/hide the plot curve on the graphs. · save the data to ASCII file (.dat). · clear the plot.
10	Signal to display	Select which data to display on the graphs (the demodulated signals X,Y,R,phi are available).
11	Scale options	

		Generic graph handling like Zoom and Pan options are explained in the user interface section.
12	Cursor	Display the cursor coordinates. The cursor can be hidden/shown by right-clicking on the graph and selecting Cursor>Show. The cursor can be moved from one curve to the other by dragging its crosshair. Note: Generic graph handling including cursor handling is explained in the user interface section.

Back Home

Scan Control

The Scan Control window is used to control the tip and various scan operations.

For a short visual introduction watch the Scan Control Video Tutorial on the Nanonis homepage.

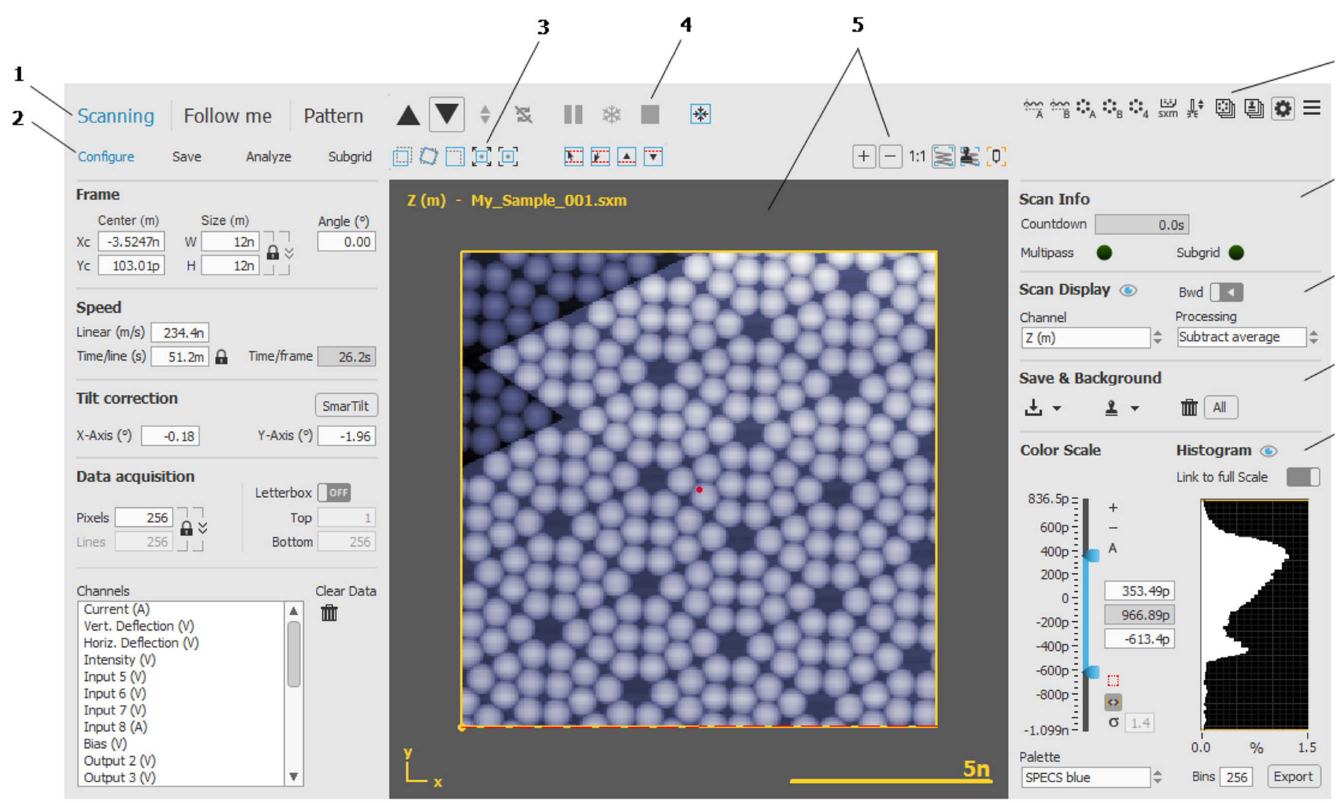
The Scan Control window is divided horizontally into three parts. The left part contains all the controls used to configure the various operations and modes. The central part contains the visualization panel and the buttons used to control the scan. The right part contains the controls used to operate on the visualization panel and some useful information. Please note that some advanced options and parameters can be found in the Scan Options window.

On the top-left corner, three buttons are used to switch between the different modes: **Scan**, **Follow me** and **Pattern**.

The visualization panel can hold the scan data plus the background images. As long as they have not been deleted, the scan data are available to the user and can be retrieved for various purposes (save to file, cross-section analysis, display or color scale change…). On the opposite, the background image is a snapshot of the scan and its data cannot be retrieved, changed or saved. It is used to give a qualitative and visual representation of the pasted scan.

Note: Saving settings from this window also saves the settings of the different tools (Line monitors, Scan monitors, Scan Options window) available on the top right corner of the window.

I) Scan Control window



1	Mode	Select in which mode to operate. - Scanning mode is used to configure the scan parameters and save options. It also contains the Analyze section to measure distances and angles and perform cross-section, and the Subgrid section to perform experiments while scanning in a user-defined grid. - Follow me mode is used to freely move the tip, manipulate atoms, record data along path and perform lithography. - Pattern mode is used to carry out experiments at specific locations controlled by different patterns: grid, line and free points.
2	Section Settings	Navigate through the different sections (f.i. Configure, Save, Analyze, Subgrid in the Scanning mode) to configure each section settings (more information is given in each corresponding section below). Sections change between the different modes.
3	Section Tools	These tools change when switching the section, and are used to modify something in the visualization tool (like moving the scan frame,

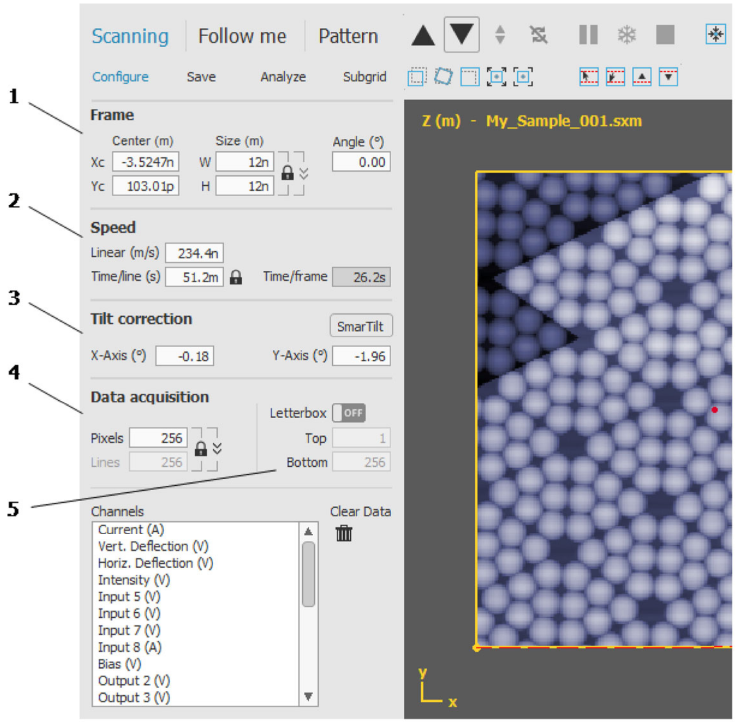
		reshaping a grid, etc).
4	Scan Controls	Buttons to control the scan and save operations: Continuous scan on/off. Start a new up/down scan. Pause/resume the scan. When paused, it is possible to change parameters, move the tip in **Follow me** mode or perform grid experiment. When resuming, the tip will move back to its original position and the scan will continue. Freeze the slow scan axis: the frozen line will be scan over and over until you unfreeze it. When pressed, the freezing process will apply on the next backward line. Stop the scan. Invert the scan direction. Move the **tip to the center** of the scan image.
5	Visualization Panel & Camera Tools	Display the sample surface from the virtual camera point of view. On the top-left are the available tools to operate on the specific elements, depending in which settings section you are. On the top-right are the tools to operate on the camera: Zoom in/out. The mouse wheel can also be used to zoom in/out around the cursor position. Adjust the camera such as one acquisition pixel corresponds to one screen pixel. Adjust the camera to the scan frame. Adjust the camera to the framework composed by the scan images pasted in the background. Adjust the camera to the grid, line or points. Adjust the camera to the full piezo range.
6	Scan Info	**Countdown**: Display the remaining time of scanning. It also shows if **Multipass** and **Subgrid** are currently active.
7	Scan display	**Show/Hide** the scan image. **Channel**: select which scan channel to display. **Bwd/Fwd**: select the scan direction to display. **Processing**: select which filter to apply to the displayed data. The processing is not applied on the saved data!
8	Save & Background	**Save** immediately the scan data to file. When **Next** is activated, the scan will be saved as soon as it has finished. When **All** is activated, all the completed scans will be automatically saved. The file format is fully described here. Tip: moving the mouse over the save button will display the next filename. **Paste** the current scan data to the background. When **Next** is activated, the scan will be pasted as soon as it has finished. When **All** is activated, all the completed scans will be automatically pasted. The background image is a snapshot of the current scan and its data cannot be retrieved, changed or saved. It is used to give a qualitative and visual representation of the pasted scan. The background can retain several images at the time, and they can be organized in the Multibackground configuration panel available in the Tools menu. **Delete** the pasted scan image from the active background. When **All** is used, all the pasted images are deleted so that the background is emptied.
9	Color scale & Histogram	**Color Scale**: defines to which portion of the Z-axis the color palette applies. The color scale can be adjusted automatically through different methods: **Full Scale** button adjusts the color scale to the full range of the data buffer. **Standard Deviation** button adjusts the color scale to the mean value of the entire data buffer plus/minus the standard deviation multiplied by a factor. This factor is set by the user. **Region Of Interest** works in combination with one of the latter buttons. When full scale and region of interest are pressed, the color scale adjusts to the full range of the data buffer contained in the ROI area. When standard deviation and region of interest are pressed, the mean value is calculated in the ROI area. When only the ROI button is pressed, the behavior is the same as if the full scale were pressed. The region of interest can be defined in the Analyze section. **Palette**: select which color palette to use. The palette can be edited in the **Color Palette** located in the System>Options panel of the main window menu. Switch on/off the **histogram**. **Link to full scale/Link to Col. Sliders**: it links the histogram range to the color scale full range or to the color scale sliders respectively. **Bins**: how many bins to display in the histogram. **Export**: export the histogram data into a .dat file.

Tools

Multipass configuration	Open the Multipass configuration window
Line monitor A & B	Open one of the two Line scan monitors to observe the data for each scanned line.
Scan monitor A & B	Open one of the two Scan monitors to display the scan image of a selected channel.
Quad scan monitor	Open the Quad scan monitor to display 4 scan images simultaneously.
Scan inspector	Open the Scan inspector to browse and display saved images.
Tip shaper	Open the Tip Shaper to shape a STM tip through a sequence of three steps.
Multibackground configuration	Open the Multibackground configuration panel.
Options	Open an Option window to configure advanced and special settings related to the Scan Control.

II) Scanning>Configure section

Configure the most common settings of the scan. Advanced and special options can also be found in the Scan Options>Scan.

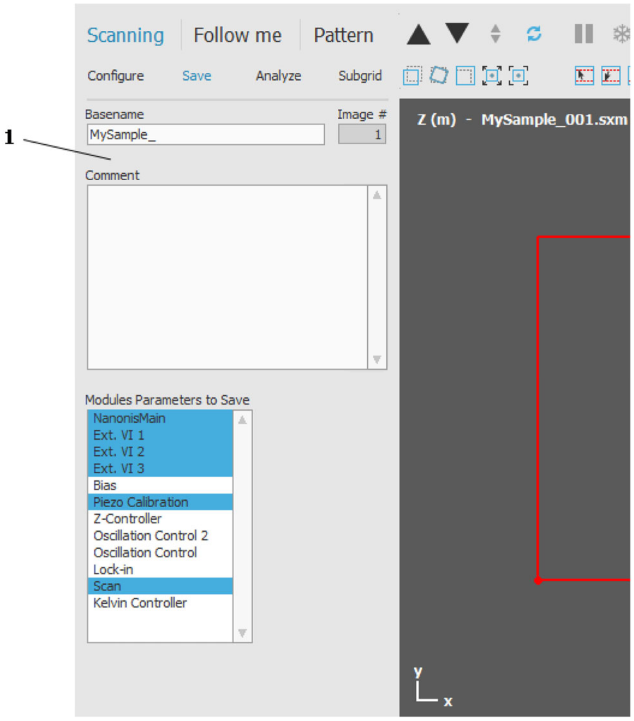


1	Frame	Configure the scan frame. The **lock** button can be used to keep the **Width/Height** ratio constant. The **double arrow** button is used to square the scan field by setting the **Height** equal to the **Width**. These tools can be used to modify the scan frame directly from the visualization panel: · Move the scan frame. · Rotate the scan frame. · Resize the scan frame. · Define a new scan frame. · Adjust the scan frame size to the camera view. Tip: it is possible to undo the current tool action by pressing the right mouse button before releasing the left button.
2	Speed	Define the scan speed in the forward direction. If needed, the backward speed can be configured in the Scan Options>Scan panel. The speed can be defined either as **Linear** or **Time/line**. The **lock** button selects which speed remains constant when adjusting the scan frame size. When hovering the mouse over **Time/frame**, a tip strip informs about the current RT frequency and the RT cycles per pixel. Note: the scan speed directly affects the data oversampling and therefore the signal to noise ratio. The slower the speed the higher the oversampling and the lower the noise.
3	Tilt Correction	The **SmarTilt** button initiates the procedure for a compensation of the sample tilt. This is only possible when demodulating Z (requires that the Z-controller is on). The sample tilt is not indicated in this module, you have to open the PiezoCalibration module and switch to the Z-correction tab. The procedure sets first the amplitude according to the size of the scan frame of the ScanControl module, then adjusts the frequency such that the tip velocity is as specified in the **speed** section of **ScanControl** module, then the slope in x and y is measured and added to the slope compensation of the PiezoCalibration module. After that, the xy modulation is turned off again.
4	Data acquisition	**Channels**: select in the list which channels to acquire. Only the selected channels can be displayed and saved. **Pixels & lines**: define the image resolution. The **lock** button can be used to keep the ratio constant. The **double arrow** button is used to square the scan resolution by setting the **Lines** equal to **Pixels**. Delete the scan data.
5	Letterbox	The Letterbox configuration modifies the region of lines to scan out of the total number of lines. This allows to scan a specific region inside the scan area. This region is defined by the **Top** and **Bottom** line numbers, where the Top line is always its topmost edge and the Bottom line is always its lowest edge (when neither scan image nor camera are rotated). The line numbers range from 1 to the total number of lines, meaning that when Top line is 1 and Bottom line is the total, the scan lines selection covers the entire scan frame, independently from the scan direction. The Scan Lines can also be used through the following mouse tools: · Select the **Top** line. · Select the **Bottom** line.

		• Start a **down scan** at the mouse position. This actually sets the Top line to the mouse position. • Start an **up scan** at the mouse position. This actually sets the Bottom line to the mouse position.

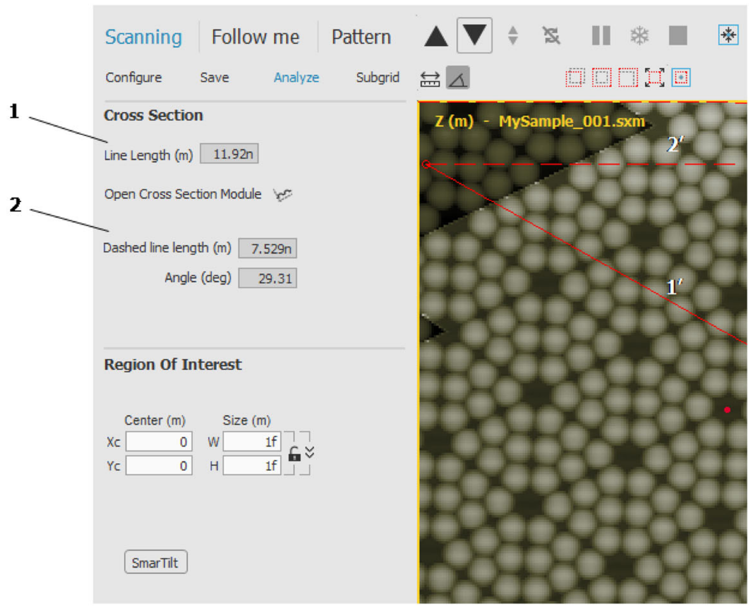
III) Scanning>Save section

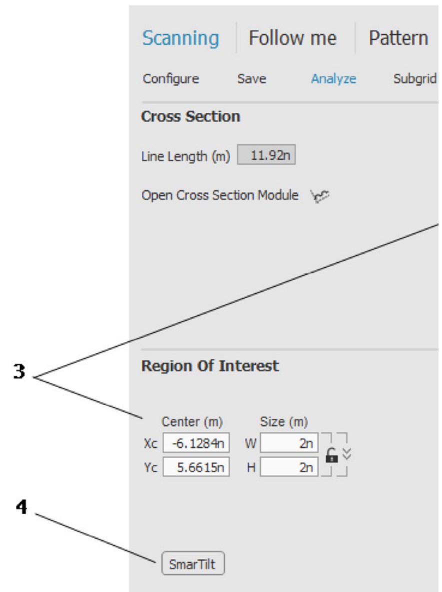
A session mechanism is implemented so you can save subsequent scans automatically. The filenames are generated automatically given the **Basename** and **Image number**. All files will be stored in the current session directory.



1	Save settings	**Basename**: defines the file basename when saving scans. The file names will be "BasenameNNN.sxm" where NNN is an automatically increasing number. **Image #**: index of the next saved scan. **Comment**: text you want to include in the file header. **Modules parameters to Save**: select from the list which additional parameters to include in the file header. Tip: special tags can be used in the Basename to include additional timing information: %a (abbreviated weekday name), %b (abbreviated month name), %c (locale-specific date/time), %d (day of month), %H (hour, 24-hour clock), %I (hour, 12-hour clock), %m (month number), %M (minute), %p (a.m./p.m. flag), %S (second), %x (locale-specific date), %X (locale-specific time), %y (year within century), %Y (year including century) An optional # modifier before the format code letter (e.g. %#d) removes the leading zeros. Additionally, the tag %R used in the basename removes the index number 1. If more images are saved afterwards with the same basename, the %R tag won't have any effect (i.e. if %R is used in the basename and there is no file with the same basename saved in the session directory, then the saved filename won't include the index number). Example: %y%m%d_MySample

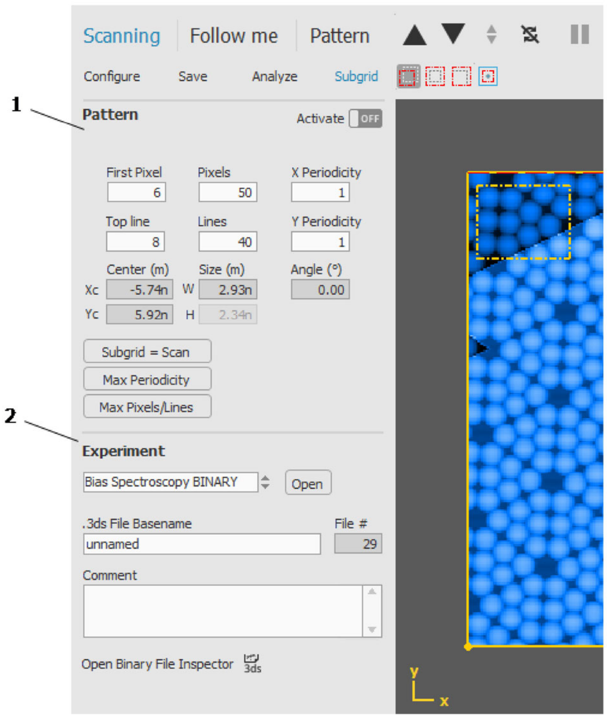
IV) Scanning>Analyze section





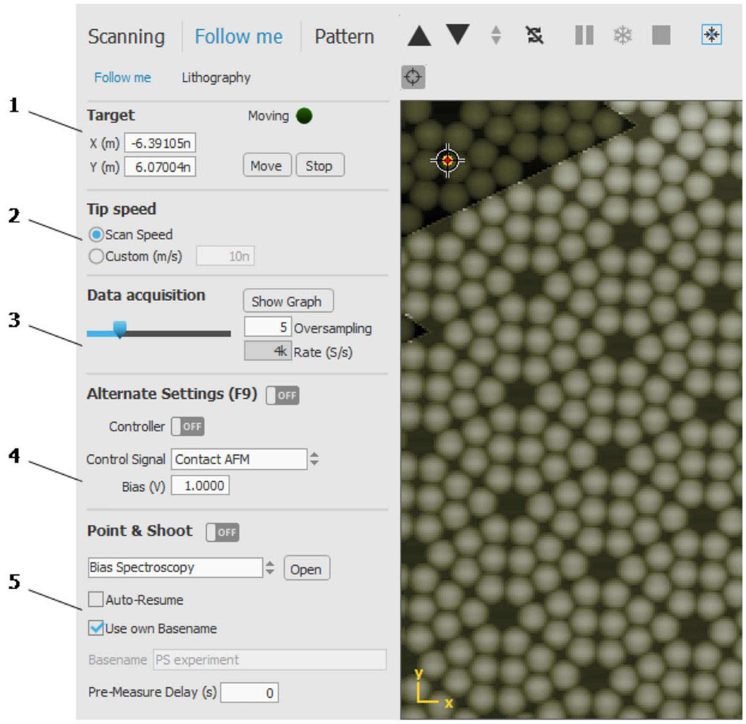
1, 1'	Cross Section Line	Use the **ruler** tool to display the plain line only. The plain line can be used as a ruler and for cross section. When displaying the cross section through the **Open Cross Section Module** button , it switches accordingly when switching between forward and backward in the scan module.
2, 2'	Dashed line	Use the **protractor** tool to display the dashed line. The dashed line can be used as a ruler and a protractor (angle measured between the 2 lines).
3, 3'	ROI frame	Define the ROI center coordinates and size in meters. The resulting frame is displayed in the visualization panel. The **lock** button can be used to keep the **Width/Height** ratio constant. The **double arrow** button is used to square the scan field by setting the **Height** equal to the **Width**. These tools can be used to modify the scan frame directly from the visualization panel: · Move the ROI frame. · Scale the ROI frame. · Resize the ROI frame. · Span a new ROI frame. · Center the ROI frame to the scan frame.
4	ROI SmarTilt	This button initiates the procedure for a compensation of the sample tilt. This is only possible when demodulating Z (requires that the Z-controller is on). The sample tilt is not indicated in this module, you have to open the Piezo Calibration module and switch to the tab Z-correction. The procedure sets first the amplitude according to the size of the ROI frame of the Scan Control module, then adjusts the frequency such that the tip velocity is as specified in the **speed** section of Scan Control module, then the slope in x and y is measured and added to the slope compensation of the Piezo Calibration module. After that, the xy modulation is turned off again.

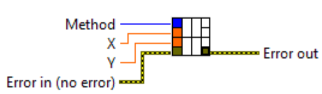
V) Scanning>Subgrid section



1	Pattern	The **Activate** switch button sets the Subgrid to active, meaning that once scan starts, the selected experiment will be performed automatically at each point of the Subgrid. This section allows the user to set the dimensions of the Subgrid. **First Pixel**, **Top Line**, **Pixels** and **Lines** should be set. The **X Periodicity** and **Y Periodicity** define the number of scan pixels and lines respectively distributed between each Subgrid point. This resolution defines the **Center (m)** and **Size (m)** of the Subgrid. The **Angle (m)** is defined by the angle of the scan frame itself. The button **Subgrid=Scan** sets the Subgrid equal to the scan frame area and resolution. The button **Max Periodicity** adjusts the Subgrid frame to its maximum size keeping constant the number of Subgrid pixels and lines. The button **Max Pixels/Lines** adjusts the Subgrid frame to its maximum size keeping constant the periodicity of Subgrid pixels and lines.
2	Experiment	Select from the list which experiment to perform at each point of the Subgrid: - **.3ds File Basename**: defines the file basename of the experiment binary data. The file names will be "BasenameNNN.sxm" where NNN is an automatically increasing number. - **File #.**: index of the next saved file. - **Comment**: text you want to include in the file header. Open the Binary File Inspector to view 3ds files.

VI) Follow me>Follow me section

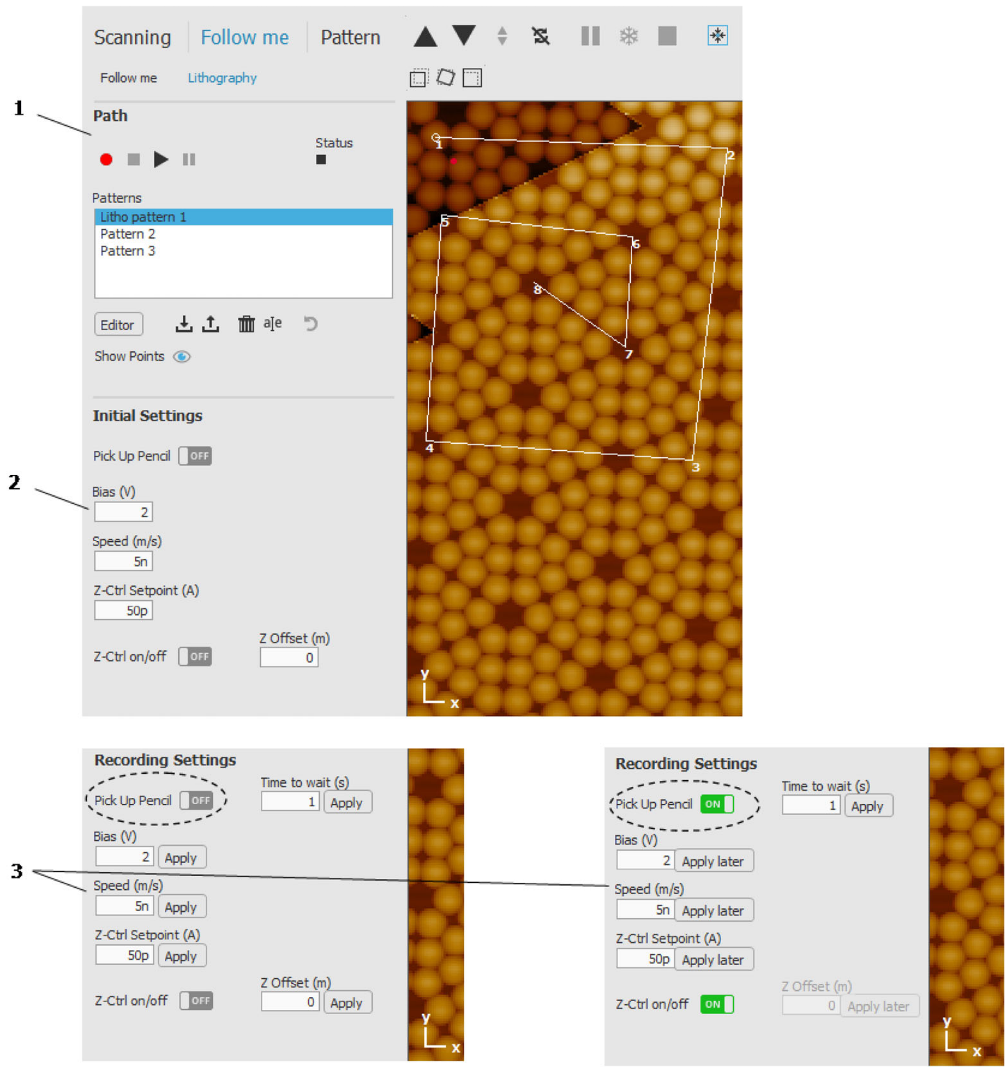


1	Target	Define the (x,y) point coordinates where the tip will move to. The **Move** and **Stop** buttons control the tip motion. With the tool selected, you can also move the tip by clicking with the mouse in the visualization panel.
2	Tip speed	Define at which speed the tip moves. **Scan speed**: use the same speed as the **Linear** scan speed. **Custom**: define custom speed value.
3	Data acquisition	**Oversampling**: select the oversampling of the data when the tip is moving **Show graph:** open the Tip Move Recorder, a graph which displays the data acquired along the tip path. **Rate(S/s):** display at which rate the data are sampled while the tip is moving.
4	Alternate Settings	Allows to quickly switch to different settings. **Button** or F2: enable/disable the alternate settings. **Controller**: switch on or off the Z-Controller. **Control signal**: select on which signal the Z-Controller is assigned. **Bias**: select the bias voltage. These settings can be used together with the **tip-lift** setting in the Z-Controller, e.g. for atom manipulation. Note: Every time you change those settings, you need to turn off and on the switch (or press F2) to apply them.
5	Point & Shoot	Select from the list which experiment to perform automatically when the tip finishes the movement. 1) The experiments with saving options are saved in ASCII format. 2) The **External VI** option allows you to select a VI which will be called when the tip reaches the point. The external VI's connector must be **exactly** as follows:  - **Method**: enum control with the three modes: init, run, end. - **X**: DBL (x coordinate) - **Y**: DBL (y coordinate) - **error in**: error cluster - **error out**: error cluster. If error out contains an error, the program will show a dialog indicating the error and ask whether to continue or stop the experiment. The init and end methods can for example be used to open/close references. They can be stored in shift registers for use in the run method. Please refer to the example VIs included in the programming interface. This will be a good starting point to create your own external VIs. The **Open** button opens the experiment window in order to configure it. **Use own basename**: by default those experiments which save data to file use their own file basename (defined in their own module), but this option allows to define a different basename for the files created through the Point & Shoot experiments, so that when the experiment finishes, the original basename is retrieved to the experiment module. **External VI path**: select the user specific VI you want to run.

		Pre-measure Delay: time to wait once arrived to the point before performing the experiment. **Auto-Resume**: When clicked, it re-actives the scan if paused.

VII) Follow me>Lithography section

The lithography section provides the capability to create a path and then make the tip follow it. Several settings can be changed during the tip travel.

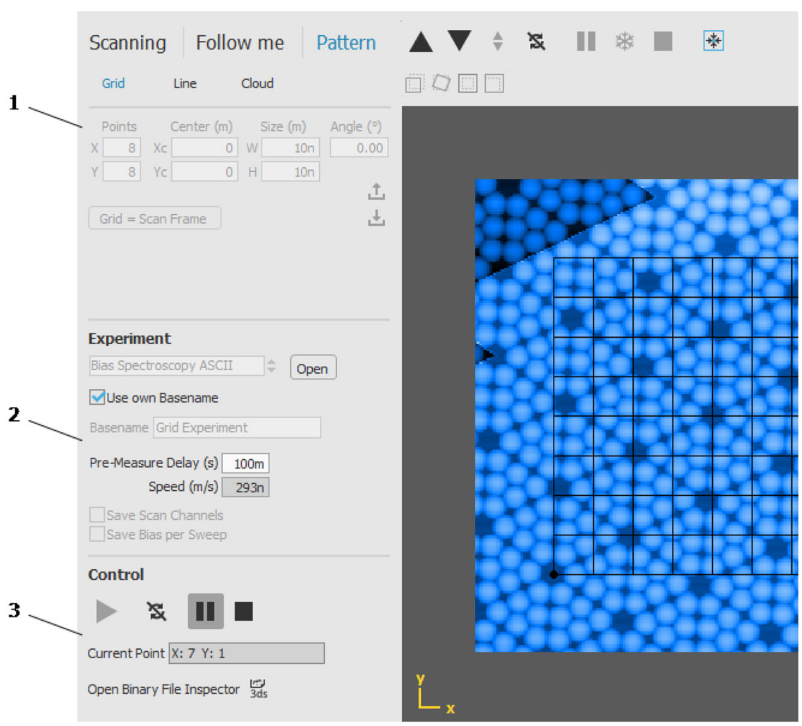


1	Path	Define and execute the pattern that the tip will follow. · Record: Define the pattern while drawing the desired path and selecting different settings in each section of the path. · Stop: Stop recording or executing the pattern. · Execute: Execute the selected pattern. · Pause: Pauses the lithography pattern on execution. When resuming, the tip will finish the ongoing movement. **Patterns**: List of defined patterns to follow. Editor: Open a text editor containing all the commands defined for the selected pattern from the list. **Show points**: Show numbers associated to the points which define the selected pattern. · Load: Load a pattern from a nanonis lithography file (.nlt). · Save: Save the selected pattern to a nanonis lithography file (nlt). · Delete: Delete the selected pattern. · Rename: Rename the selected pattern. · Undo: Undo the last command. **Status**: Indicate the current status of the lithography section:

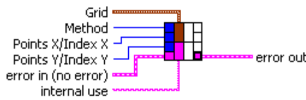
		• Idle (not recording pattern, not executing pattern) • Recording selected pattern • Executing selected pattern with Pick Up Pencil disabled • Executing selected pattern with Pick Up Pencil enabled The tools can also be used to modify the pattern directly from the visualization panel: • Move the pattern. It allows to move the whole pattern at the same time, or individually selected points. • Rotate the pattern. • Reshape the pattern.
2	Initial Settings	Settings applied right before the pattern starts executing if Pick Up Pencil is down. If Pick Up Pencil is up, the pattern starts with the same settings that were working before the pattern started, and then the Initial Settings are applied right after switching Pick Up Pencil to down. **Pick Up Pencil**: When activated, all the settings used before starting recording are used. **Bias (V)** **Z-Ctrl Setpoint (A)** **Speed (m/s)** **Z-Ctrl on/off** **Z offset (m)**: It adds this offset value to Z(m) position. This is only possible if Z-controller is off.
3	Recording Settings	Settings applied during the execution of the pattern if Pick Up Pencil is down. The Recording Settings (all except Time to wait) which are manually applied when Pick Up Pencil is up will be actually applied and executed when switching back Pick Up Pencil to down. This is why, when Pencil is up, the Apply buttons show a small clock indicating this behavior. **Pick Up Pencil**: When activated, all the settings used before starting recording are used. **Bias (V)** **Z-Ctrl Setpoint (A)** **Speed (m/s)** **Z-Ctrl on/off** **Z offset (m)**: It adds this offset value to Z(m) position. This is only possible if Z-controller is off. **Time to wait (s)**: It adds a time to wait.

VIII) Pattern section

With the Grid mode you can perform different measurements on various user-definable patterns: grid, line or a cloud of points. For example, you can define a grid of 128x128 points and take a bias spectroscopy curve at each point. You can also implement your own measurement routines to be called at each point.



1	Grid/Line/Cloud	Select and configure on which pattern to perform the experiment. The tools can also be used to modify the pattern directly from the visualization panel: • Move the pattern. • Rotate the pattern.

		· Resize the pattern (**Width/Height** ratio is kept constant). · Reshape the pattern. For the **Cloud** pattern: · Add a new point. · Delete a point. · Move a point. · · Change the point order in the list.
2	Experiment	Select from the list which experiment to perform on the pattern. 1) The **ASCII** experiments save one ASCII file per point. 2) The **BINARY** experiments save all the measurements in a single binary file. 3) The **External VI** option allows you to select a VI which will be called at each point. The external VI's connector must be **exactly** as follows:  - **Grid**: Cluster containing five SGL controls (single precision float) specifying the grid frame: center X, center Y, width, height, angle. The coordinates are in meter, the angle in degree. Only provided in the init-method. - **Method**: enum control with the three modes: init, run, end. - **Points X/Index X**: I32 (init: number of points in X, run: X index in the grid (starting at 0)) - **Points Y/Index Y**: I32 (init: number of points in Y, run: Y index in the grid (starting at 0)) - **error in**: error cluster - **internal use**: string - **error out**: error cluster. If error out contains an error, the program will show a dialog indicating the error and ask whether to continue or stop the experiments. The external VI will be called with the init-method before starting the measurements. Then, it will be called on every points of the grid with the run-method. After the last point, it will be called once more with the end-method. The init and end methods can for example be used to open/close references. They can be stored in shift registers for use in the run method. Please refer to the example VIs included in the programming interface. This will be a good starting point to create your own external VIs. Note: if the pattern has 100000 points or more, only **BINARY** experiments and the **External VI** are available. The **Open** button opens the experiment window in order to configure it. **Use own basename**: by default those experiments which save data to file use their own file basename (defined in their own module), but this option allows to define a different basename for the files created through the experiments on a grid, so that when the experiment finishes, the original basename is retrieved to the experiment module. **External VI path**: select the user specific VI you want to run. **Pre-measure delay**: time to wait on each point before performing the experiment. **Speed**: indicate at which speed the tip moves between points. This speed is the same as the one defined in the **Follow me** section. **Save scan channels**: save the scan channels into the grid experiment file. In binary files, the scan channels names are saved into the experiment parameters section of the header, and the scan channels data along with the parameters data. In ASCII files, scan channels names and values are saved into the header. In the DIO Trigger experiment, scan channels are saved into a .sxm file.
3	Control	Start the experiment on the pattern. Pause/resume the experiment. Pause occurs right after the experiment has finished on a point. Stop the experiment. 3ds Open the Binary File Inspector to view 3ds files. Continuous experiments on/off. **Current Point**: indicate the progress status of the experiment on the pattern.

Back Home

Multi-Pass configuration

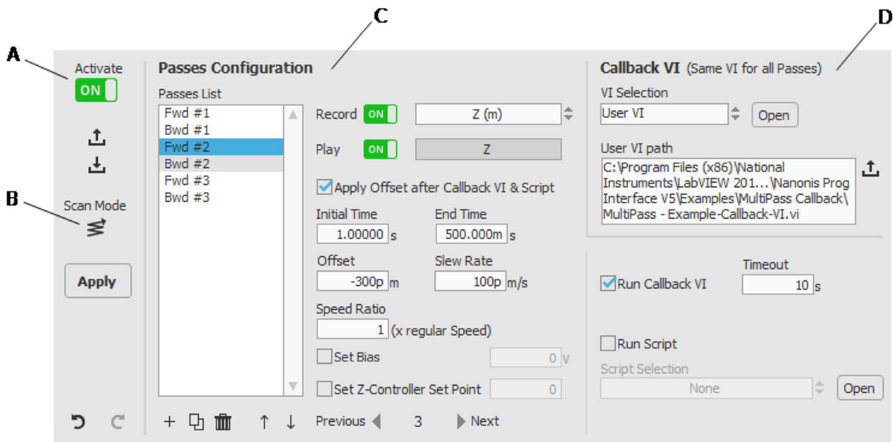
Multi-pass scanning allows to scan a line several times using different parameters (like Bias voltage or Z Controller set point) or recording a channel in one pass and playing it back (with an offset) in the following pass(es).
Multi-pass is quite versatile and can be used for different experiments like:

- Scanning forward and backward scan using different bias voltages or Z controller set points (single pass mode with different fwd/bwd parameters)
- Scanning the same line several times using different bias voltages or Z controller set points
- Recording topography (or any other channel) in the first pass and playing it back in the next pass(es) (with an offset if desired)
- Executing some code (callback VI or script) at the beginning of each scan line

The multi-pass configuration window can be opened from the corresponding button in the top right corner of the Scan Control module.

Currently the scan data acquisition channel selection is the same for all passes. When doing multiple passes the channel names will have the pass number as prefix, i.e. they will look like [P1] Z (m).

Saved scan files will contain one frame per channel and pass. The header of the saved file contains the multi-pass configuration that was used to acquire the image. The saved files are compatible with normal single scan files, the same software to open the files should work fine (except Gwyddion).



A) Activate and Apply

On the left side of the configuration panel there are 2 very important buttons: *Activate* and *Apply*. Also, you can save and load multi-pass configurations using the corresponding buttons.

1	Activate	Enables/disables multi-pass scanning mode. When it is off, the configuration is ignored and Scan Control will perform a normal (single pass) scan.
2	Load / Save	Allows to save the multi-pass configuration to disk or load a previously saved configuration. The configurations are stored in the "<All users application data>\Nanonis\Certificate <certificate no>\Multi Pass" folder (in Windows XP: "C:\Documents and Settings\All Users\Application Data\Nanonis\Certificate <certificate no>\Multi Pass") by default. Don't forget to press the **Apply** button after loading a configuration if you want to use it.
3	Apply	Applies the current configuration for scanning (if multi-pass is active). Any changes made in the configuration will **not** be applied until the **Apply** button is pressed. This is also true if you load a configuration from a file. It will display in the configuration window but will not be applied until the **Apply** button is clicked. When clicking **Apply** while a scan is running, the active scan is stopped.

B) Scan mode

Also on the left side the scan mode can be selected. Like the other settings this is only applied when the **Apply** button is clicked.

- **Normal scan mode** uses a constant speed in the slow scan direction. The slow scan axis is only stopped for the specified delay at the beginning of each pass. In normal scan mode 2 passes are not exactly on the same line since the slow scan axis advances.

Linefeed scan mode freezes the slow scan axis until all passes (forward and backward) of one line have been completed. Then it advances to the next line (without acquiring data) and scans the next line. So all passes are (when neglecting other effects) exactly on the same line. The line feed is done at scan speed (pass #1 forward speed). If the delay at the beginning of pass #1 fwd is long enough for the linefeed there will be no additional delay, else the delay will be extended to leave enough time for the linefeed.

C) Passes configuration

As mentioned before, changes to the passes configuration are only applied once the **Apply** button is clicked. This will stop an active scan.

Important: the total time to wait at the beginning of the line (BOL) depends on several things: **Initial Time**, **End Time**, Callback VI **Timeout**, the time it takes to apply the playback **Offset**, the time it takes to run the **Script**, and the flag to **Apply Offset after Callback VI & Script**. Examples:
Play Z, Initial Time=2s, End Time=1s, Offset=10nm, Slew Rate=5nm/s: the total time at the BOL is exactly 5s.
Play Z, Initial Time=2s, End Time=1s, Offset=10nm, Slew Rate=5nm/s, Callback VI that takes roughly 5s with Timeout=10s, Apply Offset after Callback VI=false: the total time at the BOL is roughly 5s.
Play Z, Initial Time=2s, End Time=1s, Offset=10nm, Slew Rate=5nm/s, Callback VI that takes roughly 5s with Timeout=10s, Apply Offset after Callback VI=true: the total time at the BOL is roughly 10s.
Play Z, Initial Time=2s, End Time=1s, Offset=10nm, Slew Rate=5nm/s, Callback VI that takes roughly 5s with Timeout=2s, Apply Offset after Callback VI=true: the total time at the BOL is exactly 7s.

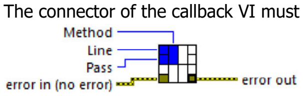
1	Passes list	The Passes list displays the passes that have been configured. Each pass can be configured individually or in combination with other passes. When selecting multiple passes simultaneously, the difference in configuration is highlighted in orange in the configuration panel on the right and changes in configuration are applied to all selected passes. Use the buttons below the list to add, duplicate or delete a pass, or to rearrange the passes.
2	Record	Switch it on and select the signal if you want to record a signal during the selected pass(es). There is one buffer for forward scan data and one buffer for backward scan data. When recording a channel in a forward pass it can only be played back in a forward pass. The buffer remains active after playing back, so it can be played back multiple times (e.g. using different offsets) until another signal is recorded.

3	Play	Allows playing back a signal that was recorded in a previous pass.
4	Apply Offset after Callback VI & Script	This is only enabled if there is a Play signal and a Callback VI or a Script configured in the selected pass. If it is unchecked or it is not enabled, Initial Time (s) is applied right at the beginning of the line. If it is checked, Initial Time (s) is applied only after the Callback VI finishes or timeouts, or until the Script finishes.
5	Initial Time (s)	If a Play signal is configured in the selected pass, Initial Time is the time to wait before applying Offset. Otherwise, it defines a delay applied at the beginning of the line before starting the selected pass. This is useful to let the system settle when changing a parameter for example.
6	End Time (s)	This is only enabled if there is a Play signal. It defines the time to wait after applying Offset before starting the selected pass.
7	Offset	It specifies an offset for the playback signal. For instance when recording Z (topography) it can be played back using an offset of e.g. 100pm. This is applied right after Initial Time (s).
8	Slew rate	Sets the offset increment per second at which the playback offset is applied. Inf or 0 mean that the full offset is instantly applied.
9	Speed ratio	Speed factor for the selected pass(es). The regular scan speed (set in Scan Control) is multiplied with this factor. The default is 1, i.e. regular scan speed is used. To scan a pass in twice the regular speed, use a factor of 2. When changing the regular scan speed (in Scan Control) the speed of all passes will adjust accordingly. The regular scan speed can change after each pass during a scan. The **speed ratio** can't be changed during a scan.
10	Set Bias	Overrides the normal Bias voltage with the value specified here for the selected pass. It is applied right at the beginning of the line before starting the selected pass.
11	Set Z controller set point	Overrides the normal Z controller set point with the value specified here for the selected pass. It is applied right at the beginning of the line before starting the selected pass.
12	Previous/Next	Use these to cycle through the configured passes. It's the same as clicking a single pass in the **Passes list**.
13	Run Callback VI	If checked, the specified Callback VI (unique for all passes) will run for the selected pass. **Timeout**: Sets the maximum time to wait in case the Callback VI didn't finish its execution.
14	Run Script	If checked, the specified Script will run at the beginning of the line before starting the selected pass. **Script Selection**: Displays a list of the multipass scripts deployed to the real time controller. A different script can be set per pass. If the Script has not been yet deployed in the Scripting Tool, a warning message is displayed. The **Open** button opens the Script module, where the multipass scripts can be created and deployed.

D) Callback VI

For maximum flexibility of the multi-pass feature a user defined VI (LabVIEW Virtual Instrument) can be called at the beginning of each line. In this VI you can use the Programming Interface to switch any setting in the software (like switching on/off Oscillation control or Lock-In modulation) or you can control an external device. There are a few things to consider when using a Callback VI:

- The callback VI is the same for each pass. It will get the line and pass numbers so it can be decided what to do where.
- The callback VI sends a handshake when it finishes executing, but a **timeout** can be configured so that the scan will continue after it even if the callback VI is still running.
- The connector of the callback VI must exactly match the following pattern:



  Method (enum I16): **init** executes right after starting the scan and before running Line 0/Pass 0, **run** executes during the scan, and **end** executes right after finishing the scan or when stopping it, and it can be used to reset parameters or to close references.

  - **Line** (I32) is the line number (0 based).
    Pass (I32) is the pass number in the current line (0 based). Even pass numbers are forward lines, odd pass numbers are backward lines.
    error in/error out hold the error cluster. When an error occurs the callback VI is no longer called.

- Example: when doing a scan with N lines and 2 passes the (Line,Pass) parameters will be: (0,0), (0,1), (0,2), (0,3), (1,0), (1,1), ..., (N-1,2), (N-1,3), (-10, 0).
- There's an example callback VI installed along with the Programming Interface (MultiPass - Example-Callback-VI.vi). You can find it in the examples directory.

Note: LabVIEW must be installed and running on the host PC if you want to use this feature. VI Server must be enabled in LabVIEW on TCP/IP port 3363. To do this, open LabVIEW and go to 'Tools>Options… >VI Server: Configuration' and activate TCP/IP protocol on port 3363.

1	VI Selection	Set to **None** (default) to disable the callback VI. Set to **User VI** and specify a VI path to use a callback VI.
2	Open button	If a callback VI is specified, this button will open it in LabVIEW so you can modify or configure it.
3	User VI path	Path of the callback VI.

Back Home

Multibackground configuration

The Multibackground configuration panel is used to organize the images pasted in the backgrounds. This module allows to reorder the images to set which one is on top of the other, to hide or lock or delete them, to adjust the scan camera (the scan visualization panel) to one or all of them, etc…

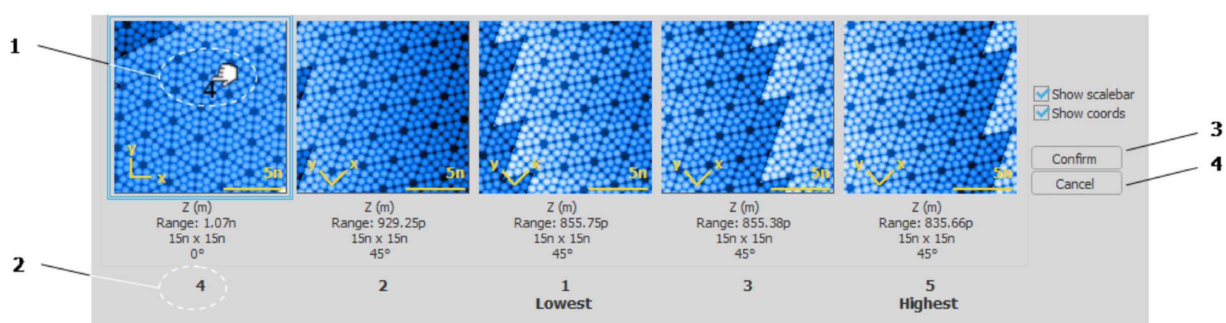
The leftmost background is the lowest layer, and the rightmost background is the highest layer being on top of the others.

It can be opened from the corresponding button in the top right corner of the Scan Control module.

1	Thumbnail Tools	· Pastes the scan data to this specific background. · Adjusts the camera (visualization panel in the Scan Control module) to the pasted image in this background. · Deletes the pasted image contained in this background. · · Shows/Hides the pasted image contained in this background. Its data is not deleted so it can be displayed back at any time.
2	Active Background	Background currently selected (out of the total number of backgrounds) to automatically paste the next scan data when using the buttons in the Scan Control module. The active background is framed by a green outline which automatically jumps to the next background (if it is not locked) when the scan image is pasted.
3	Image Info	Information of the pasted image above. It displays the channel, the range, the scan frame size (width x height) in meters, and the scan frame angle in degrees.
4	Lock/Unlock	Blocks the access to this specific background, so that its buffer is never replaced by a new pasted image unless unlocking. A locked background cannot be an active background, so the green outline skips the locked background and jumps to the next available one.
5	Image Order Indicator	Displays the position (which background is on top of the other) when hovering the mouse over the background thumbnails.
6	Delete All	Deletes all pasted images.
7	Show Scalebar/Coords	Displays a scalebar to have a visual help of the scaling inside each thumbnail. Displays a coordinates system inside each thumbnail showing the rotation angle of every pasted image.
8	Reorder	Switches to the reorder mode to change the backgrounds positions (which background is on top of the other). See below to have more details about how to reorder.

Reorder

The Multibackground configuration panel shows a different aspect when clicking the Reorder button. It allows to re-arrange the different background layers.



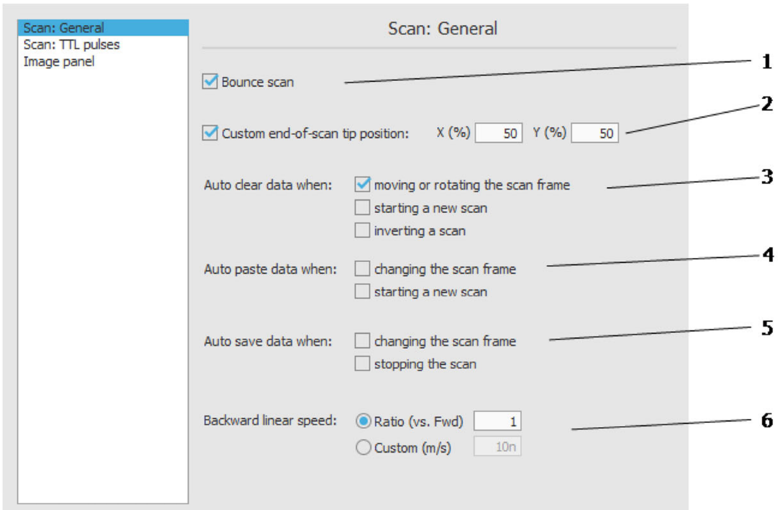
1	Selection mouse	Selects which is going to be the position for the selected background. The mouse changes its appearance (increasing the number by 1) when clicking a background thumbnail, and this number sets the future position of the clicked background. The reorder position number below each thumbnail changes accordingly. The changes don't take effect until pressing the Confirm button.
2	Reordered position	Shows the position that the above background will have when confirming the changes.
3	Confirm	Applies the changes so that the background layers reorder accordingly.
4	Cancel	Cancels the changes so that the backgrounds positions stay as before.

Back Home

Scan Control Options

The Scan Options dialog window is used to configure scan options/parameters and display settings for the Scan Control module. Select the category on the left side, configure the settings on the right side of the window.

Scan: General



1	Bounce scan	Defines the scan directions for continuous scans. When checked, an 'up' scan is followed by a 'down' scan, then again an 'up' scan and so on. This prevents tip jumps. When not checked the scan is always performed in the same direction.
2	Custom end-of-scan tip position	Normally the tip will stay at the current position when a scan finishes (and continuous scan is off). When checking this option the tip will move to the specified position (in percent of the scan frame) when a scan finishes.
3	Auto clear data	Choose at which actions the data of the current scan should be cleared.
4	Auto paste data	Choose at which actions the data of the current scan should be pasted to the background.
5	Auto save data	Choose at which actions the data of the current scan should be automatically saved.
6	Backward speed	Defines the speed for the backward scan lines. Default is a speed **Ratio** of 1 (thus using the same speed as for the forward lines). To use a different speed you can define a speed **Ratio** (e.g. 2 for twice as fast as the forward lines) or a **Custom** linear speed.

Scan: TTL pulses

Allows to generate TTL pulses on a digital line at the beginning of each line, end of each line and at the beginning of each pixel. The pulse width will be 1 real-time cycle (when using a real-time frequency of 10kHz the width is 100us).
To configure, first select the digital port (DIO port A or B) to use for the TTL pulses.

To enable pulses at the beginning of each line, specify the digital line to use, set the polarity (High Active means 3.3V for the pulse), then check the **Enable** box.

Caution: Make sure not to select digital lines which are connected to other equipment like a piezo motor driver or a digitally controlled preamp. Always select the correct port, line and polarity before checking the **Enable** box.

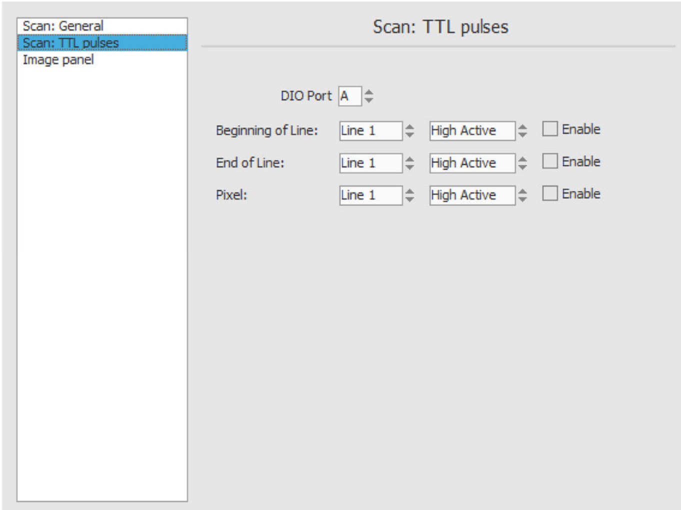
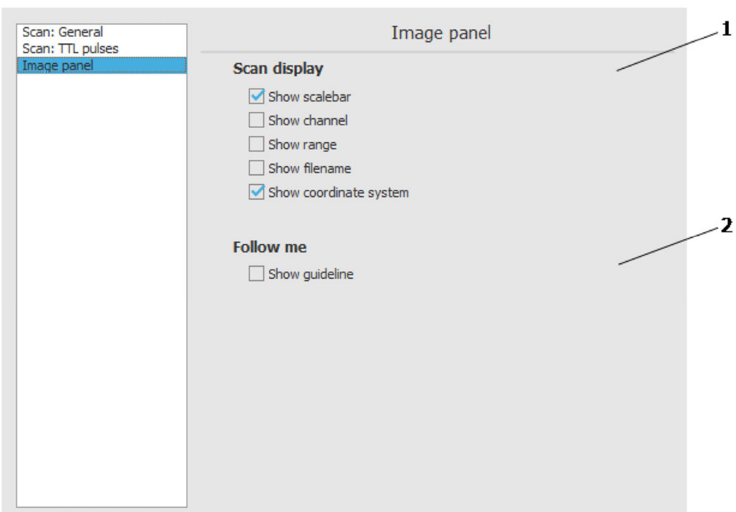


Image panel

Contains settings for the scan image display in the scan control.



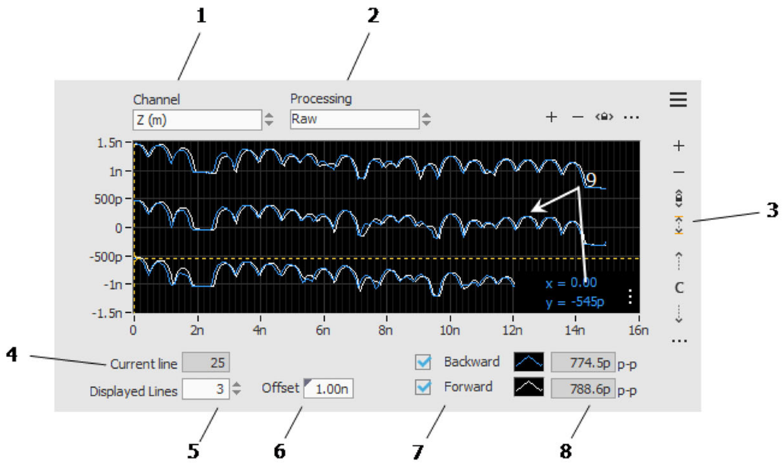
1	Scan display	Select what information should be displayed in the scan image display of the Scan Control module when in **Scanning** mode.
2	Follow me	Select what information should be displayed in the scan image display of the Scan Control module when in **Follow Me** mode. **Show guideline**: when checked a line between the current tip position and the mouse position is displayed. This is the path of the tip when initiating the movement (by clicking at this location).

Back Home

Line Scan Monitor

The Line Scan Monitor displays the selected channel scan line while you are scanning an image.

The forward and backward scan lines are plotted individually.



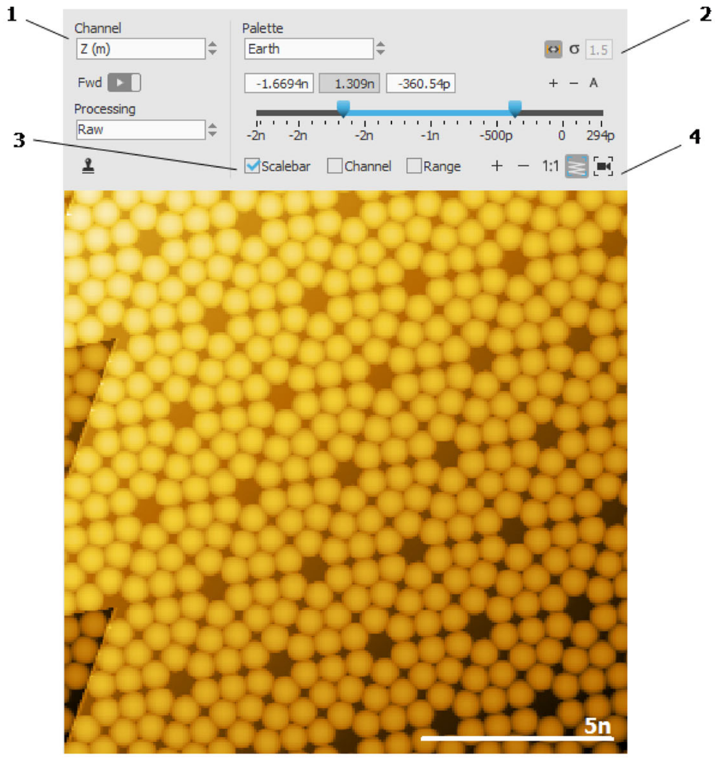
1	Channel	Select the channel you want to observe in the graph while you are scanning an image. The available channels from the pull-down selector correspond to those selected in the **Buffer** tab in the Scan Control module.
2	Processing	Processing applied to the scan lines. Possible processing are: **Raw**, **Subtract average**, **Subtract slope**, **Subtract average & slope** and **Differentiate**.
3	Zoom, Pan and Drop-Down menu	Use those buttons to **zoom** and **pan** the trace. The drop-down menu prevents further display options. Note: Generic graph handling is explained in the user interface section.
4	Current line	Index of the current scanned line.
5	Displayed lines	Set the number of scan lines you want to display (from 1 to 4). When displaying multiple curves, and for a positive **Offset**, the lowermost curve corresponds to the latest scanned line.
6	Offset	Offset that separate the curves for clarity. Tip: right-clicking on the control to select pre-defined offset values.
7	Scan direction	Select which scan direction you want to display on the graph.
8	Peak to peak	Peak to peak value of the most recent scan line for the forward and backward directions.
9	Cursors	Right-click on the graph and select **Cursors>Show** to make cursors visible, bring them to center, or assign them to the desired plot. Note: Generic graph handling including cursor handling is explained in the user interface section.

Back Home

Scan Monitor

Two Scan Monitors (A & B) are available. They display the scan image of one channel each.

They load/save their settings when loading/saving settings either from the Main Window file menu or from the Scan Control settings menu.



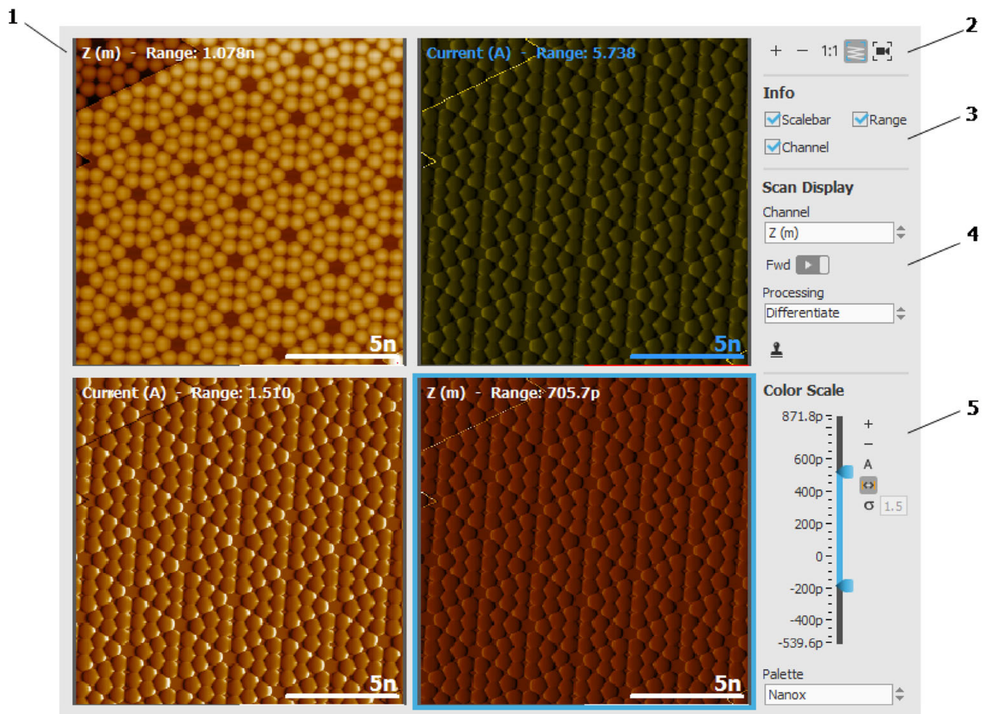
1	Scan display	Control the data to display in the monitor. **Channel**: select which scan channel to display. **Direction**: select the scan direction to display. **Processing**: select which filter to apply to the displayed data. · Paste the scan data to the background of the Scan control visualization panel.
2	Color scale	Control the coloring of the displayed data. **Color scale**: defines on which portion of the Z-axis the color palette applies. The color scale can be adjusted automatically through two different methods: Full scale button adjusts the color scale to the full range of the data buffer. Standard deviation button adjusts the color scale to the mean value of the entire data buffer plus/minus the standard deviation multiplied by a factor. This factor is set by the user. **Palette**: select which color palette to use. The palette can be edited in the **Color Palette** located in the **System>Options** window.
3	Info	Select which information to display in the monitor.
4	Camera	Tools that modify the camera view: · Lock the view to the scan control camera. · Lock the view to the scan frame. · Adjust the camera such as one acquisition pixel corresponds to one screen pixel. · · Zoom in/out. Tip: the mouse wheel can also be used to zoom.

Back Home

Quad Scan Monitor

The Quad Scan Monitor has the same functionality as the ordinary scan monitors, but it shows 4 channels simultaneously. To activate one of the monitor, just click on it.

The Scan Monitors load/save their settings when loading/saving settings either globally (from main window) or from the Scan Control.



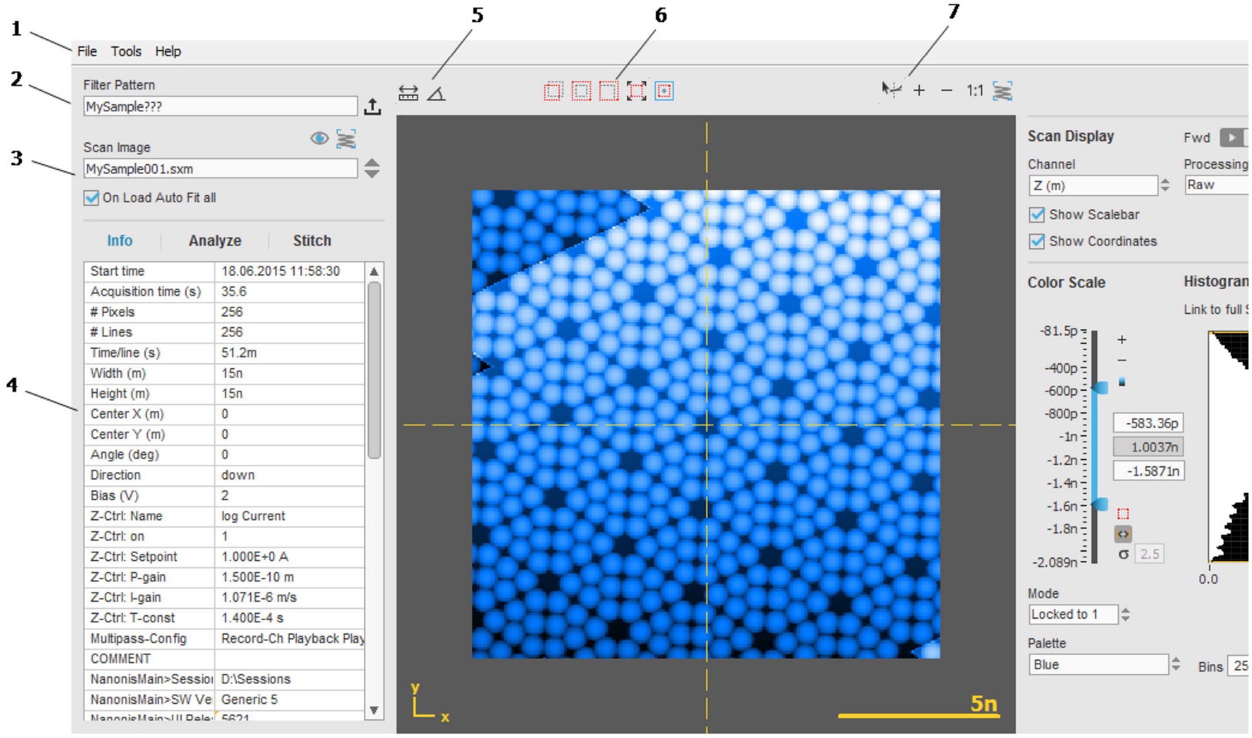
1	Monitors	Display the sample surface from the virtual camera point of view. The red box indicates the active monitor for which the **Scan Display** and **Color Scale** settings apply.
2	Camera	Tools that modify the camera view: · Lock the view to the **Scan control camera**. · Lock the view to the **Scan frame**. · Adjust the camera such as one acquisition pixel corresponds to one screen pixel. · · Zoom in/out.
3	Info	Select which information to display in each monitor.
4	Scan display	Control the data to display in each monitor. **Channel**: select which scan channel to display. **Direction**: select the scan direction to display. **Processing**: select which filter to apply to the displayed data. · Paste the scan data to the background of the Scan control visualization panel.
5	Color scale	Control the coloring of the displayed data. **Color scale**: defines on which portion of the Z-axis the color palette applies. The color scale can be adjusted automatically through two different methods: Full scale button adjusts the color scale to the full range of the data buffer. Standard deviation button adjusts the color scale to the mean value of the entire data buffer plus/minus the standard deviation multiplied by a factor. This factor is set by the user. **Palette**: select which color palette to use. The palette can be edited in the **Color Palette** located in the **System>Options** window.

Back Home

Scan Inspector

The Scan Inspector is the perfect tool to quickly browse your saved scans. It can display up to 4 scan images simultaneously in the visualization area.
With the **Filter Pattern** capabilities it is possible to precisely define which files within the selected folder you want to access.

When installing the Nanonis software you get two versions of the Scan Inspector: one is available from the corresponding button in the top right corner of the Scan Control module and the other one is a stand-alone version that you can access from the Windows “Start>Programs>Nanonis” menu.



1	Menus	File: - **Open file or folder (Ctrl+O)**: open a .sxm or .1xm scan image file, or select a folder containing scan images. - **Export graph as PNG or BMP (Ctrl+G)**: export the main scan image (plus axes and color scale) into an image file type (PNG by default). - **Export main image as PNG or BMP (Ctrl+I)**: export the main scan image (only the image in the resolution of the scan) into an image file type (PNG by default). - **Export frame data as ASCII (Ctrl+A)**: export the data associated to the selected channel of the main scan image into an ASCII file. - **Copy images to Clipboard (Ctrl+B)**: export the current picture view into the Clipboard - **Justify file names to right/left**: change the alignment of the scan images names and filter pattern to the right/left. This can be very convenient when the filenames are too long and they don't fit in the assigned space. - **Open File Load Folder**: open the folder where the images are being loaded from. - **Reopen same image on next start**: open the same image the next time the software restarts. - **Exit (Ctrl+Q)**: quit the Scan Inspector. Tools: - **Thumbnail viewer**: open the Thumbnail viewer. - **Show Spectroscopy locations**: open the Spectroscopy locations window. - **External data processing**: open either the file with all images or just the selected visible frame in the selected third party data processing application. This option is available only if you have the corresponding application installed on your system. Note for Windows 7 and Vista users: if not able to open a file or frame with WSxM, run it always as administrator by right-clicking on its exe and checking Properties>Compatibility>Privilege Level> Run this program as an administrator. Then run the Scan inspector also as administrator. Help: - **Reference (F1):** open the Help documentation of the Nanonis controller software. - **Info**: open the Info window with the version and release number of the Scan Inspector.
2	Browse settings	Click on the **Browse** button to select a file or a folder. Selecting a file will open it in the graph as the main scan image and will set the **Filter pattern** to match its basename so that the **Scan image** list contains all the images with the same basename stored in the same folder. The **Scan image 2**, **Scan image 3**, **Scan image 4** lists within the Stitch tab are filled up with the same items. Selecting a folder will set the **Filter Pattern** to "*" so that the **Scan image**-s lists contain all the scan files present in the selected folder. The **Filter pattern** follows the Windows matching rules: use the question mark character (?) to match any single character and the asterisk character (*) to match any sequence of one or more characters. Note: the selected path will appear in the window title bar. Tip: Both **Filter Pattern** and **Scan Image** allow dragging & dropping a folder/file into them.
3	Scan image	Select which image to display as the main scan image. The list contains the scan files from the selected folder that match the **Filter pattern**. Use the up/down arrows to quickly browse through the list. · Paste image: paste the main scan image into the Scan Control background. This option is available only if you opened the Scan Inspector from the Scan Control. · Hide/Show image: hide the main scan image (make it invisible in the image view) or show it back again. · Adjust view to the scan image: adjust the camera view to the frame size of the main scan image.

		On Load Auto Fit all: it sets the images to auto fit when loading a new file.
4	Info	Display the information saved in the header of the main scan image file.
5	Cross section tools	The tools provided in this section can be used as a ruler, protractor, and as a display of the cross section. More information in Analyze tab.
6	ROI tools	These tools can be used to modify the ROI (Region Of Interest) frame directly on the visualization panel. More information in Analyze tab.
7	Camera tools	These tools are used to control the camera view so that it is possible to zoom in/out, adjust scale and move around within the visualization panel. Zoom in/out. The mouse wheel can also be used to zoom in/out around the cursor position. Adjust the camera such as one acquisition pixel corresponds to one screen pixel. Adjust view to the framework composed by all the images currently selected (main scan image plus images selected in the Stitch tab).
8	Scan display	Control the information to display in the graph. **Channel**: select which scan channel to display. **Fwd/Bwd**: select the scan direction to display. **Processing**: select which filter to apply to the displayed data. The processing is not applied on the saved data! **Show Scalebar**: display a scale bar on the right bottom corner of the visualization panel. **Show Coordinates**: display a coordinate system on the left bottom corner of the visualization panel which will show up rotated according to the rotation of the main scan image.
9	Color scale & Histogram	**Color scale slider**: defines to which portion of the Z-axis the color palette applies. If the **color scale mode** is set to **Locked to** something, then a change in the color scale value will be applied to all images. If **Unlocked** is selected as the mode, the new color scale value will be applied to the selected image. Zoom in/out to adjust the limits of the slider. Color ramp: maximize the slider (adjust the limits to the current Z-axis range) and show up the color ramp according to the Z-axis range and the color palette. **Color scale automatic buttons**: The color scale can be automatically calculated from the image data through different methods (the image or images where the data is coming from is defined by the **Color scale mode**): Full scale: adjust the color scale to the full range of the data. Standard deviation: adjust the color scale to the mean value of the entire data buffer plus/minus the standard deviation multiplied by a factor. This factor is set by the user. Region of interest works in combination with one of the latter buttons. When **full scale** and **region of interest** are pressed, the color scale adjusts to the full range of the data buffer contained in the ROI area. When **standard deviation** and **region of interest** are pressed, the mean value is calculated in the ROI area. When only the ROI button is pressed, the behavior is the same as if the **full scale** were pressed. The region of interest can be configured in the Analyze tab, and it can only be used on the main scan image. **Color scale mode**: define the image or images where the data is coming from, in order to define the color scale: – **Locked to All**: if any of the above buttons is pressed, it takes the data from all selected images into account to define the color scale (f.i. if full scale is pressed, it will adjust the color scale to the maximum and minimum values found within the combined data from all images). – **Locked to 1** (default): if any of the above buttons is selected, it takes the main scan image into account to define the color scale. All images will adjust their color to the color scale calculated for the main scan image. – **Locked to 2**: if any of the above buttons is selected, it takes the scan image 2 into account to define the color scale. All images will adjust their color to the color scale calculated for the scan image 2. – **Locked to 3**: if any of the above buttons is selected, it takes the scan image 3 into account to define the color scale. All images will adjust their color to the color scale calculated for the scan image 3. – **Locked to 4**: if any of the above buttons is selected, it takes the scan image 4 into account to define the color scale. All images will adjust their color to the color scale calculated for the scan image 4. – **Unlocked**: if any of the above buttons is selected, it takes the selected scan image into account to define the color scale. Only the selected image will adjust its color to the new color scale. **Palette**: select which color palette to use.

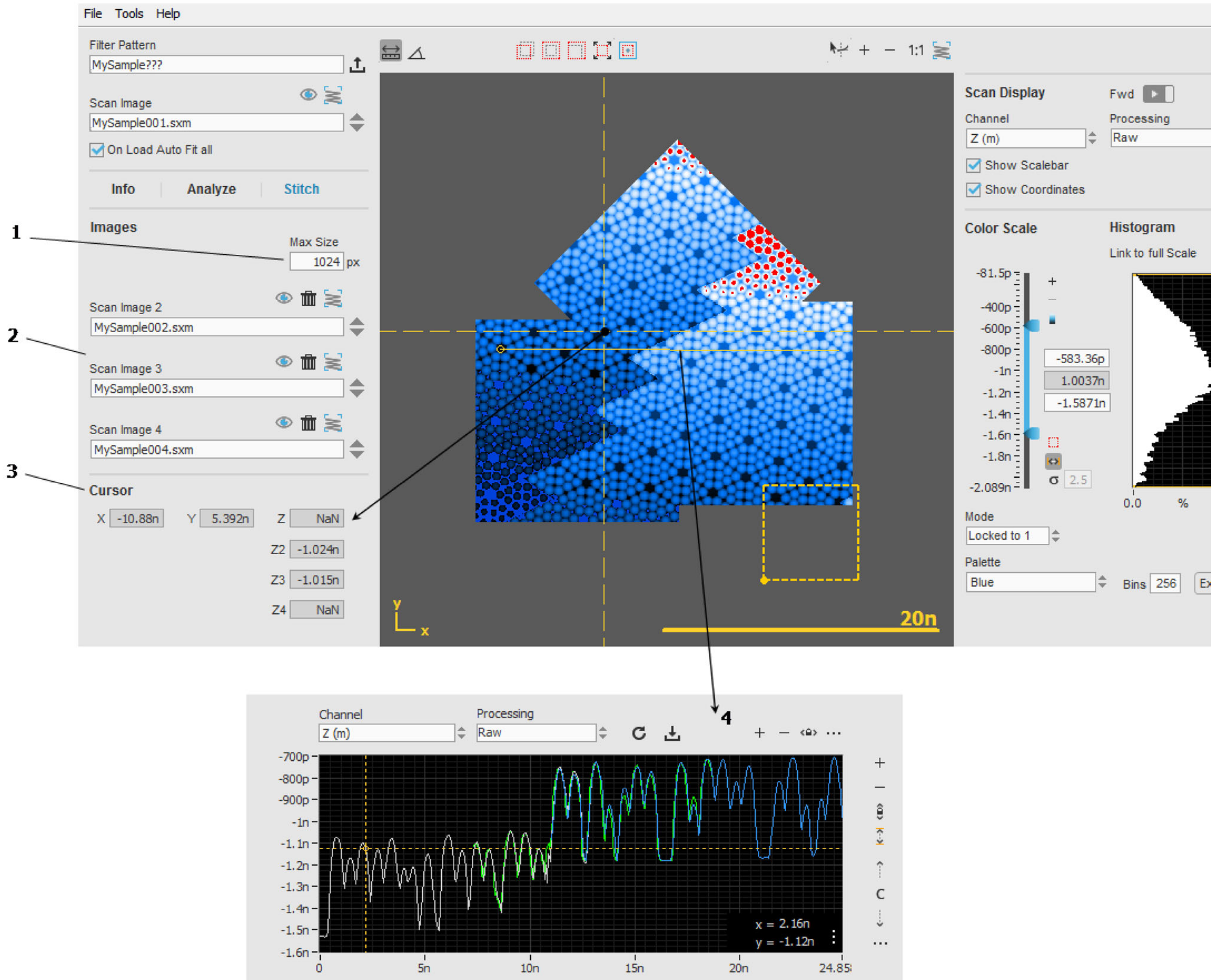
Analyze tab

The analyze section provides some tools to measure the image data in different ways, like configuring and visualizing a cross section, defining a region of interest (ROI) area which the color scale adjustment buttons will use to calculate the color scale range, and displaying a cursor which can show the coordinates at a certain point.

1, 2, 3	Cross section tools	· **Ruler**: display a plain line on the image view which can be manually resized and repositioned by the user. It can be used as a ruler and for the cross section display. · **Protractor**: display a dashed line on the image view which can be manually resized and repositioned by the user. It can be used as a ruler and a protractor (angle measured between the 2 lines).
4	Cross section graph	Display the cross section along the plain line defined by the **Ruler**. It provides some data processing methods to be applied to the displayed data. It allows saving the cross section data, and it displays a cursor along the curve.
5	Cursor	Display the channel value at the cursor position (X, Y) for the main scan image. The switch **Abs/Rel** allows to change the cursor coordinates between absolute (scan piezo range) and relative (scan frame).
6, 7, 8	ROI tools	A ROI (**Region Of Interest**) can be defined to calculate the color scale range through the **full scale** and the **standard deviation** buttons. The ROI can be specified on the main scan image. The resulting color scale is applied to all images currently present on the visualization panel. The ROI tools above the visualization panel can be used to modify the ROI frame directly from the image view: · Move the ROI frame. · Scale the ROI frame. · Resize the ROI frame. · Span a new ROI frame. · Center the ROI frame to the scan frame.

Stitch tab

The stitch section is the place to select more images to show in the visualization panel. The total number of images that can be shown is four.
The **color scale mode** explained in the top section of this page plays a very important role when displaying more than one image.



1	Max size	Images are downsized to the size in pixels specified in this field. The maximum size is 1024 pixels (default value).
2	Images	Select additional images to display along with the main scan image in the visualization panel. The scan image lists contain the scan files from the selected folder that match the **Filter pattern**. Use the up/down arrows to quickly browse through the lists. The images are ordered numerically so that the main scan image is displayed on top of the others whereas the **scan image 4** is displayed at the bottom layer. When one of the images contains no data for the selected channel, it is solely displayed in the visualization panel as its scan frame with a dotted outline. · Paste image: paste the selected scan image into the Scan Control background. This option is available only if you opened the Scan Inspector from the Scan Control. · Remove image: remove the image from the selection. This would be the same as selecting **no image** from the list. This does not delete the image file. · Hide/Show image: hide the selected scan image (make it invisible in the image view) or show it back again. · Adjust view to the scan image: adjust the camera view to the frame size of the selected scan image.
3	Cursor	Display the channel value at the cursor position (X, Y) for all images. If the cursor is at a position where some images overlap, it will show the channel value for the overlapped images. The configuration of the cursor can be set in the Analyze tab.
4	Cross section graph	Display the cross section along the plain line defined by the *Ruler*. More information in the Analyze tab. When multiple images are displayed in the visualization panel, the graph shows the cross section of all the images crossed by the **Ruler**, differentiating the images by using different curve colors.

Thumbnail viewer

It can be accessed through the Tools menu in the Scan Inspector window. It shows in the same window all the scan images which are saved in the selected Folder.
You can double-click an image to show it in the Scan Inspector window, or you can right-click an image to delete it from disk.

Show Spectroscopy locations

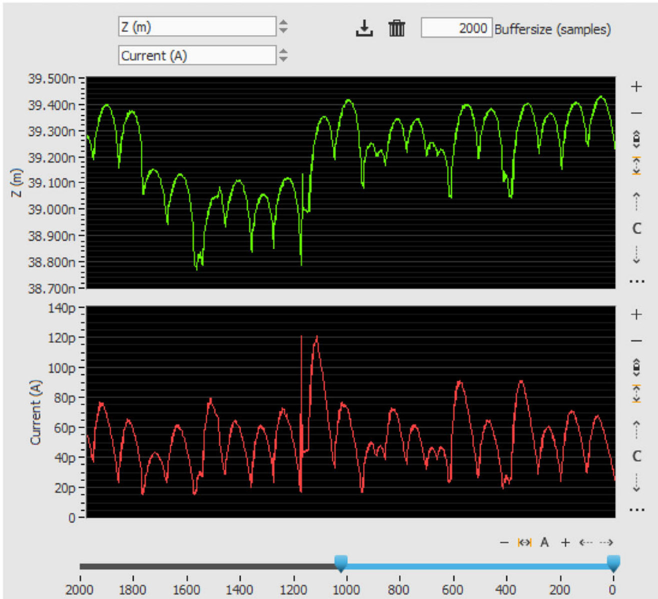
It can be accessed through the Tools menu in the Scan Inspector.
It's only enabled when the selected folder contains a list of valid spectroscopy files (.dat) which keep the X and Y position in the header.
It shows a pop-up window with the list of all valid spectroscopy files labeled with a correlative number next to the file name.

When selecting one or more files, the number assigned to the .dat file is drawn on the image view at the XY position where the spectroscopy was executed. The number is centered at this position.
When clicking on one of these numbers, the Data File Viewer window opens showing the spectroscopy data of the selected file. In order to be able to open the spectroscopy file, no tool must be selected so that the mouse pointer becomes when hovering the visualization area.

Back Home

Tip Move Recorder

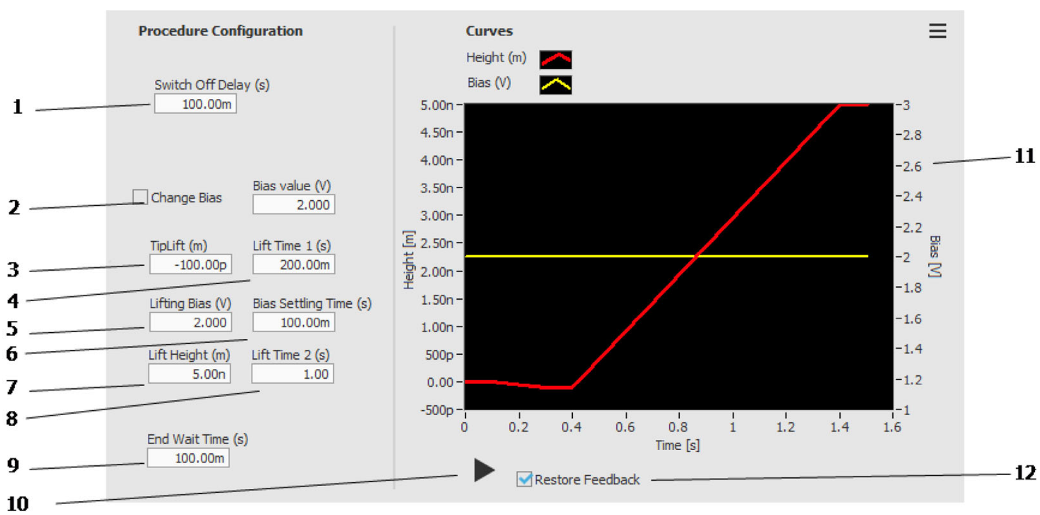
The Tip Move Recorder module displays the selected channels along the path in the follow-me mode.
It works with a **ring buffer** with a changeable size (Buffersize). The slider control below the graph lets you select a portion of the buffer to display in the graphs.



Back Home

Tip Shaper

The Tip Shaper module executes a sequence where the user can configure some timing/value parameters to first ramp Z (usually to approach the tip), then apply a bias voltage, and finally ramp Z again (usually to withdraw the tip). This procedure is mostly used to shape a STM tip, dipping it into the sample and pulling it back slowly. The initial bias voltage is restored after the whole sequence. If Restore Feedback is active, the initial Z-controller status is restored too.



1	Switch Off Delay (s)	Use this parameter for a better reproducible position when switching off Z-controller. When >0 and Z-controller is switched off, it doesn't switch off immediately but continues to run for the specified time averaging Z position.
2	Change Bias	If checked, Bias value (V) is applied right before the first ramping.
3	Tip Lift (m)	Defines the height the tip is going to ramp from the current Z position in the first position ramping.
4	Lift Time 1 (s)	Defines the given time to ramp Z in the first ramping.
5	Lifting Bias (V)	This bias voltage is applied just after the first ramping and during the second ramping. When the second ramping finishes, the initial bias voltage is restored.
6	Bias Settling Time (s)	Time to wait after applying the Lifting Bias, before ramping Z the second time. It is also used as the time to wait after applying Bias value (V), before ramping Z for the first time.
7	Lift Height (m)	Defines the height the tip is going to ramp from the current value in the second position ramping. The current value is the initial Z position plus the Tip Lift.
8	Lift Time 2 (s)	Defines the given time to ramp Z in the second ramping.

9	End Wait Time (s)	Time to wait after restoring initial bias voltage which takes place just after finishing the second ramping.
10	Start button	Executes the tip shaping sequence.
11	Graph display	Shows the resulting Bias and Height curves. It updates everytime one parameter changes.
12	Restore Feedback	If active, it restores the initial Z-controller status at the end of the tip shaping sequence.

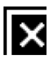
Back Home

Digital Line Interface

The digital line interface is the low-level panel to gain access to the digital lines. Set lines as in- or outputs, set static lines or pulse lines from this window. The pin assignment is listed below.

Usually there's no need to work with this module directly since the devices controlled via digital lines can be accessed from the corresponding modules (e.g. Motor Control module for a coarse motion controller).

If input lines are left unconnected, they might show as 1 (high).



1	Status	Shows the actual line status for both in- & outputs.
2	Direction	Set digital lines to input or output.
3	Polarity	Set output lines to low or high active.
4	Static	Set output lines to active or inactive. Output lines will be 1 (3.3 V) if **polarity** = high active & **static** = active, **polarity** = low active & **static** = inactive or when pulsed.
5	Pulsing	Select the lines used in the pulse generator. There is only 1 pulse generator, i.e. all selected lines are pulsed synchronously.
6	Pulsing parameters	Any 2 out of the 5 parameters define the pulse shape. Changing one of the parameters will adjust the others accordingly. The parameters are: - **Pulse width**: width of a pulse - **Delay**: delay between 2 pulses - **Duty cycle**: pulse width divided by period - **Frequency**: 1/period - **Period**: pulse width plus delay
7	Number of pulses	The number of pulses generated after clicking the **start** button ("8").
8	Start pulses	Starts the pulse generator with the defined parameters. Clicking this button when running will stop the pulse generator.

Pin assignment

DIO	Line	Pin Sub-D9
DGND	-	1
DIO0	1	6
DIO1	2	2
DIO2	3	7
DIO3	4	3
DIO4	5	8
DIO5	6	4
DIO6	7	9
DIO7	8	5

Back Home

Coarse Motion

The Coarse Motion (Motor Control) panel is used to manually control the coarse positioning.

This module depends on the system, so there are different versions. Please select your motor control type from the list below.

- Nanonis PMD4
- Curlew
- Curlew Multiprobe
- Jeol
- Unisoku
- attocube ANC150
- Omicron MSCU
- Step Motor

Back Home

Coarse Motion (Nanonis PMD)

This module controls the Nanonis PMD4 piezo motor driver. The PMD4 Tyto version with Z-end-position is explained in the section below.

PMD4 Version

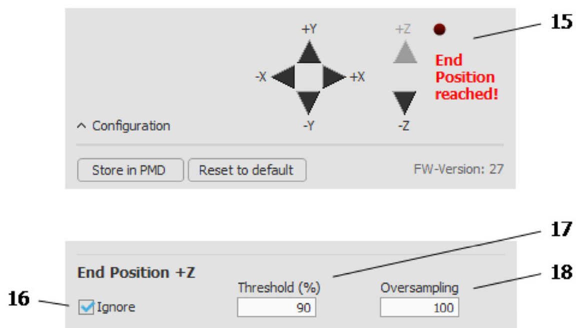
The PMD4 DIO port A connector must be connected to SC4 DIO port A using a shielded DIO cable (provided by SPECS Zurich).

The PMD4 comes with a handset. If the handset is on (indicated in the software and on the device) the PMD4 can't be controlled from the software and some controls are disabled. To use the software the handset must be off.

1	**PMD Status** LEDs	These status LEDs are mirroring the LEDs on the PMD4 device. When the **Handset** LED is lit (i.e. when the handset is activated) motor control from this module is disabled.
2	Axis LEDs	The axis LEDs are mirrored from the PMD4 device. They light up when the corresponding axis is moving.
3	**Amplitude**	Controls the waveform amplitude in volts. This is only applied when moving from this module, the handset has its own amplitude knob. The maximum voltage is 400V, but it can be limited with the Max. Amplitude ("11") setting.
4	**Frequency**	Waveform frequency in Hz. This is only applied when moving from this module, the handset has its own amplitude knob.
5	Group selector	Selects which group to move. The PMD4 supports 6 groups of 3 axis each (except group 6, this only has 1 axis). For instance, group 1 could move the scan table, group 2 the mirror etc. Use the channel configuration ("10") to configure which physical output channel to move for each axis in the selected group. The group names can be changed, right-click the group selector and select "Edit Group Name" to modify the name of the selected group.
6	Manual Move	The Move buttons move the motor a defined number of steps (**Steps XY** / **Steps Z**) in the corresponding direction. The **Stop** button will stop any movement immediately. If a manual move button is kept pressed a wait time of 0.5s, the motor will start moving continuously until the button is released. You can use the cursor keys for X (Left, Right) and Y (Up, Down) and the Page Up and Down keys (PgUp, PgDn) for the Z-Axis to move the motor from the keyboard (works only when the motor control module is the active window). When holding down the Shift key when pressing any of these keys it will move continuously immediately until the button is released. Note: The Move and Stop Keyboard Shortcut Assignment is visualized in a separate document.
7	**FW-Version**	Indicates the FW-Version currently installed on the PMD4.
8	PMD4 device configuration commands	- **Store in PMD**: The configuration of the PMD4 is stored in a flash memory in the device itself. When the software is started it will load the configuration from the PMD4. Any changes in the configuration are applied immediately, but when you switch off and back on the PMD4 it will load the configuration stored in its flash memory. When you're done configuring the device, click **Store in PMD** to make sure the change is persistent. Configuration stored in the PMD4 include: **Overload LED** / **Beeper** ("9"), channel configuration ("11"), **Max. Amplitude** ("12"), **Relay hold-on** ("13"). - **Reset to Default**: Resets the configuration to default. This does not reset the configuration in the PMD4 flash memory. To do so, click **Store in PMD** after clicking this button.
9	Overload Events	The PMD4 has one overload LED, but there's 3 overload conditions (**Amplitude**, **Speed**, **Current** overload). In this section you can configure at which overload event the LED goes on and the beeper in the PMD4 is activated. When an overload event occurs you'll see which event it is next to the event name.
10	Axis Labels	Customizable Axis Labels. The Z-Axis has
11	Channel configuration	Configure which physical channel should be active when moving the corresponding axis and how the waveform should look. The 3 displayed axes (X, Y, Z) correspond to the group selected in the Group selector ("5"). - **Output Relais**: define to which physical output(s) the generated high-voltage waveform should be applied - **Direction**: switch waveform direction. If the piezo motor moves in – direction when clicking the + button, change direction here. - **Polarity**: the polarity of the high voltage output.

		- Bipolar creates waveforms in the voltage range between –Amplitude..+Amplitude. - Unipolar + creates waveforms between 0..+Amplitude (output voltage is always positive). - Unipolar – creates waveforms between –Amplitude..0 (output voltage is always negative). - **Capacitance** (Cap.): for best performance, specify the capacitance of the piezo motor connected.
12	**Max. Amplitude**	Limits the maximum output voltage to this value. Maximum is 400V. This limit also applies for the handset, so when setting the amplitude knob on the handset to 50% at a Max. Amplitude of 400V the output voltage will be 200V. If the Max. Amplitude is 300V the output voltage will be 150V.
13	**Relay hold-on**	Specifies how long the output relay(s) remain closed after moving. After this amount of time the output is grounded.
14	**Settling Time**	Settling time after each step (steep flank of the output waveform) before the next ramp starts.

PMD4 Tyto Version (with Z-End-Position Detection)



15	**End Position** Indicator	The end position status LED with its warning text indicates that the end position is reached. It can be ignored by the **Ignore** checkbox.
16	**Ignore**	Allow movements even when +Z end position is detected.
17	**Threshold** %	+Z end position detection threshold in % of **Oversampling**.
18	**Oversampling**	+Z end position detection oversampling to avoid communication noise when moving.

Back Home

Curlew Motor Control

This module controls the Curlew SPM closed-loop piezo motors for tip coarse positioning and automatic tip/sensor exchange.

The SmarAct piezo motors used in the Curlew SPM come with a handset. If the software is running the handset is automatically switched off (indicated on the device).

1	**Get Positions**	Updates the tip position (XYZ).
2	**Tip Position**	Shows the tip position (XYZ) in meters. Note that the tip position is depending on the **Tip Type** ("6") in XYZ and on the **Offset from standard sample height** ("8") in Z.
3	Manual move	The buttons move the motors (XYZ) a defined **Step size** (in meters) at a defined **Speed** (in % of maximum speed) in the corresponding direction. The **Stop** button will stop any movement immediately. You can use the cursor keys (arrow keys) and PgUp/PgDn to move the motor from the keyboard (works only when the motor control module is the active window). Note that the right-handed coordinate system's origin is fixed to the sample, Y is oriented along the sensor carrier direction and Z is defined such that a positive move retracts the tip from the sample.
4	**Procedures**	A procedure is a predefined motor move sequence. The button Go starts the selected procedure from the drop-down menu. The Stop button will stop any movement immediately. The procedures can be executed only if the motors are referenced ("10"). The following procedures are available: • Unmount Tip: The motors move to a park position. After having inserted the transfer shuttle the UnmountTip procedure continues to unmount the tip and returns to the park position. • MountTip: The motors move to a park position. After having inserted the transfer shuttle the MountTip procedure continues to mount the tip and returns to the park position. • TipRetract: The tip is first lifted from the sample by 0.5 millimeters in +Z direction and is then retracted to a park position. • SampleCenter: The tip is moved to the XY origin (X=0, Y=0). The tip is not moving in Z direction. • SampleExchange: The motors move to a park position that allows safe sample exchange. The tip is first lifted from the sample in +Z direction and is then retracted to a park position. • Memory: The tip moves to the previously stored memory position ("5") in X and Y. The tip is not moving in Z direction.
5	**Memory**	The button M stores the current XY tip position. The latest stored values will be used in the Memory procedure ("4").
6	**Tip type**	Correct tip positions relative to the sample origin rely on the properly select tip type. The following tip types are available: • Akiyama: Akiyama Probe AFM sensor. The tip geometry is fixed. • STM: Tunneling tip. The measured tip geometry needs to be inserted in the **STM param** input fields **l** and **h**. See the manual for the definition of l and h. • LER: LER quartz based AFM sensor. The tip geometry is fixed.
7	Tip position display	The red dot indicates the tip position within the full motor range shown by the yellow square.
8	**Offset from standard sample height**	The offset from the standard sample height allows adjusting the tip position in Z ("2"). A modification of this value will not move the motor. The Auto button adjusts the Z tip position to zero. Note that once the tip is in contact with the sample the Auto button allows to adjust the absolute Z origin to the actual sample height for the given sample and tip.
9	**External step size**	It is used to set the Auto-Approach step size in nanometers. And it is also used when controlling the motor through the MotorStartMove function in the Programming Interface vis.
10	Motor referencing	The Find Reference button starts the motor referencing procedure. Only once the motors are referenced (green/red indicator) **Procedures** ("4") can be executed.

Back Home

Curlew Multiprobe Motor Control

This module controls the Curlew Multiprobe closed-loop piezo motors for tip coarse positioning and automatic probe exchange.

The piezo motors used in the Curlew Multiprobe come with a handset. If the software is running the handset is automatically switched off (indicated on the device).

Note: The Curlew Multiprobe Sample Stage (optional module) controls the sample stage piezo motors of this Curlew Multiprobe Motor Control.

1	Probe selection	Selection of the activated probe from the drop-down menu.
2	Tip position	Shows the tip position (XYZ) in meters in its right-handed probe coordinate system ("9"). Note that the tip position is depending on the Tip type ("7") in XYZ and on the offset from standard sample height ("11") in Z.
3	Manual move	The buttons move the motors (XYZ) a defined Step size (in meters) at a defined Speed (in % of maximum speed) in the corresponding direction. You can use the cursor keys (arrow keys) and PgUp/PgDn to move the motor from the keyboard (works only when the motor control module is the active window). Note that the right-handed probe coordinate system's origin is fixed to the sample center, Y is oriented along the sensor carrier direction and Z is defined such that a positive move (PgUp) retracts the tip from the sample.
4	Procedures	A procedure is a predefined motor move sequence. The button Go starts the selected procedure from the drop-down menu. The procedures can be executed only if the motors are referenced ("10"). The following procedures are available: - ParkPosition: The motors move to a park position. Note that before inserting a probe shuttle always move all probes to the park position. - UnmountTip: After having inserted the transfer shuttle the UnmountTip procedure unmounts the tip and returns to the park position. Note that the UnmountTip procedure can be executed only if the Sample Stage (in the Motor Control Tools menu > Sample Stage, refer to "Curlew Multiprobe Sample Stage Motor Control") is moved to the activated probe end stop. - MountTip: After having inserted the transfer shuttle the MountTip procedure mounts the tip and returns to the park position. Note that the MountTip procedure can be executed only if the Sample Stage (in the Motor Control Tools menu > Sample Stage, refer to Curlew Multiprobe Sample Stage Motor Control) is moved to the activated probe end stop. - Memory: The tip moves to the previously stored memory position ("5") in X and Y. The tip is not moving in Z direction.
5	Memory	The button M+ stores the current XY tip position of the activated probe. Each probe has its own memory position. The latest stored values will be used in the Memory procedure ("4").
6	Stop	The Stop button will stop any movement immediately.
7	Probe type	Correct tip positions relative to the sample origin rely on the properly select probe type. The probe geometry can to be inserted in the input fields **length** and **height** in mm. See the manual for the definition of the length and height parameters. The following probe types are available: • Akiyama: Akiyama Probe AFM sensor. • LER: LER quartz based AFM sensor. • STM: Tunneling tip. • Prober: Proper tip.
8	External step size	If selected, it is used to set the Auto-Approach step size in nanometers. And it is also used when controlling the motor through the MotorStartMove function in the Programming Interface vis.
9	Tip position display	The small dot indicates the tip position. The activated probe ("1") is highlighted in red.
10	Motor referencing	The **Find Reference** button starts the motor referencing procedure. Only once the motors are referenced (green indicator) procedures ("4") can be executed.
11	Offset from standard sample height	The offset from the standard sample height allows adjusting the tip position in Z ("2"). A modification of this value will not move the motor. The **Auto** button adjusts the Z tip position to zero. Note that once the tip is in contact with the sample the **Auto** button allows to adjust the absolute Z origin to the actual sample height for the given sample and tip. Note that there is only one offset from standard sample height value for all probes. Once the offset it set for one probe, for the other probes you can use the tip geometry ("7") to adjust for further offsets between tip position and sample.

Back Home

Curlew Multiprobe Sample Stage Motor Control (optional Module)

This module controls the sample stage piezo motors of the Curlew Multiprobe.

The piezo motors used in the Curlew Multiprobe Sample Stage come with a handset. If the software is running the handset is automatically switched off (indicated on the device).

1	**Axes Numbers**	The Axes Numbers is the preconfigured selection of the two motor channels (do not change!).
2	Move configuration	Piezo motor moves can be configured by the drive **Amplitude**, the drive **Frequency** and the number of **Steps** performed. Note that the step distance is depending on the amplitude. If the amplitude is set below a threshold value the motor will not move. The speed, for steps > 1, is depending on the drive amplitude and the frequency.
3	**Move** ±X, ±Y	The buttons move the sample stage (XY) by the specified number of Steps at the specified Amplitude and Frequency ("1") in the corresponding direction. You can use the cursor keys (arrow keys) to move the sample stage motor from the keyboard (works only when the Sample Stage module is the active window). Note that the Y direction is oriented along the transfer direction.
4	Move to End Stop (4x)	The buttons move the sample stage to the end stop position in X and Y. If the end stop is reached the green indicators light up. To perform the UnmountTip and MountTip procedures (refer to "Curlew Multiprobe Motor Control") it is required that the end stop position in X and Y for the corresponding probe are reached.
5	**Stop**	The Stop button will stop any movement immediately.

Back Home

Coarse Motion (Jeol version)

The Jeol motor controller moves continuously when the digital line for one of the direction is active. Thus, the configuration includes the speed.

1	Move buttons	The move buttons move the motor a defined number of **Steps** in the corresponding direction. The **Stop** button will stop any

		movement immediately. If a manual move button is kept pressed, the motor first runs the number of steps specified, then, after a wait time of 0.5s, it will start moving continuously until the button is released. You can use the cursor keys (arrow keys) and PgUp/PgDn to move the motor from the keyboard (works only when the motor control module is the active window).
2	**Steps**	Set the number of steps and click the **Move** button start moving.
3	**Stop** button	Stop travelling
4	**Go to 0**	Move to the Zero position.
5	Move indicator	Indicates when the motor is executing a move command.
6	**Speed** selector	Speed of the motor when moving continuously.
7	Show advanced configuration	When selected, the advanced configuration settings are displayed.
8	Pulse **Width**	Duration the digital line will be active for a pulse.
9	Pulse **Pause**	Delay after a pulse (digital line inactive) before next pulse is generated.
10	Factory Default	Reset the pulse configuration back to the factory defaults.
11	Invert Z	Invert the move direction of the Z axes.

Version with Position read-out

12	**Positions**	Position read-out
13	**Axis Configuration**	Set axis-specific calibration and offset. The **Auto** button uses the current position as offset.

Back Home

Coarse Motion (Unisoku Version)

The Unisoku motor controller uses two pulsed digital lines for +Z and -Z, respectively.

1	Move buttons	Move steps in +Z or -Z direction. When holding down a button for 500ms, it will start to move continuously (using the pulse parameters **Pulse Width** and **Pulse Delay**) until the button is released.
2	Number of **Steps**	Number of Steps (pulses) to move when one of the direction buttons ("1") is clicked.
3	**Stop** button	Stop moving
4	Show advanced **Configuration**	To display the advanced configuration settings.
5	Pulse width	Duration the digital line will be active for a pulse.
6	Pulse delay	Delay after a pulse (digital line inactive) before next pulse is generated.
7	DIO Port	Port (A or B) to use at the Nanonis connector box.
8	Lines	Lines to use for -Z and +Z direction, respectively. Refer to the SCCB (Signal conditioning and connector box) manual for detailed information on the digital lines and their numbering.

Back Home

Coarse Motion (attocube ANC150 version)

The attocube ANC150 is connected to the host PC via serial port (RS-232).

1	Move buttons	Move the number of steps defined in the **Steps** control in + or - direction of the corresponding axis.
2	**Steps** (X, Y, Z)	Enter the number of single steps to move when a move button is clicked.
3	**Step Counters** (X, Y, Z)	Indicates the number of motor steps done.
4	**Reset**	Resets the motor **Step Counters** (X, Y, Z).
5	**Voltage**	Set the voltage of the corresponding axis in the range 0..70V.
6	**Frequency**	Set the frequency of the corresponding axis in the range 0..8000Hz.
7	Serial port	Select the port (**VISA Resource Name**) the ANC150 is connected to.
8	**Connection Speed**	Select one of the available Speeds.
9	**Reconnect**	Reconnect if the communication got lost.
10	Time to wait (ms)	Time to wait in milliseconds between sending a command to and receiving the answer from the device.
11	**Response**	Shows the communication status the ANC150. For example, if **Time to wait** ("10") is too short, it might show an error as it didn't receive a response to the command sent in the time specified. This doesn't necessarily mean that the ANC didn't perform the command.
12	**Stop**	Stops the motor. Only available when moving.
13	**Axis numbers**	Assigns axis numbers to the directional axes. On the software window, the axes are labelled X,Y,Z while on the ANC they are numbered 1,2,3. Using these controls you can assign each direction the corresponding axis number of the ANC.
14	**Manual Ground**	Manually grounding of all axes immediately.
15	**Automatic ground after Hold-on**	Automatic grounding of all axes after the **Hold-on** time (ms) if there is no movement in progress.

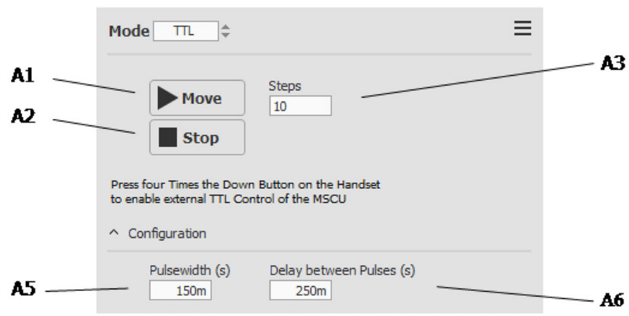
Back Home

Coarse Motion (Omicron Version)

The Omicron motor controller remote controls the Omicron MSCU (micro slide control unit). The MSCU can be controlled by TTL or by Serial Mode. To change between modes use the Mode selector on the upper left of the Omicron Motor Control.

TTL Mode

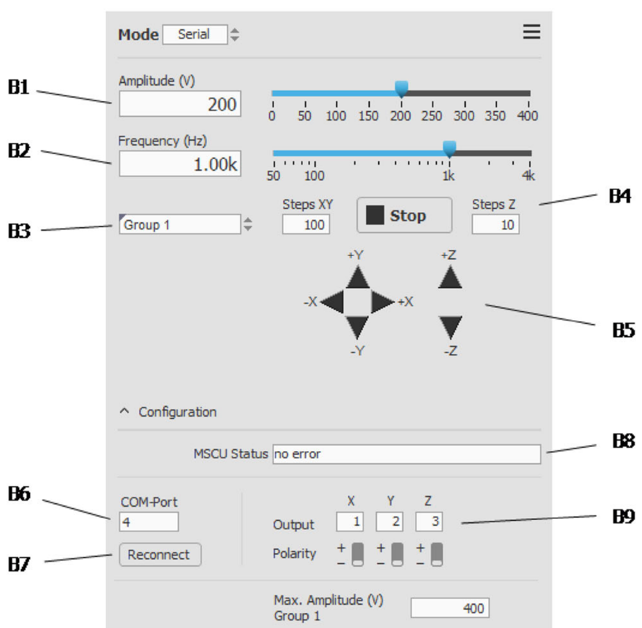
For TTL mode the MSCU handset must be connected. To activate the external TTL input of the MSCU, press four times the down button on the handset. As the external input can only trigger movement in one direction, set it to -Z (Z approach) to be able to use the auto approach.



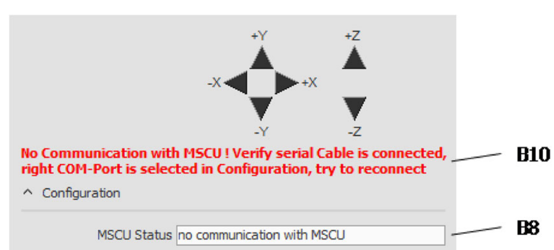
A1	**Move** Button	Start the number of steps defined in the **Steps** control
A2	**Stop** button	Stop travelling
A3	Number of Steps	Set the number of steps and click the **Move** button start moving
A4	Pulse width	Duration the digital line will be active for a pulse
A5	Pulse delay	Delay after a pulse (digital line inactive) before next pulse is generated

Serial Mode

For Serial mode the MSCU must be connected to the host PC with the MSCU serial cable.



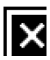
In case of no communication with the MSCU, an error message is shown and the move buttons are disabled.



B1	**Amplitude**	Controls the waveform amplitude in volts. The maximum voltage is 400V.
B2	**Frequency**	Waveform frequency in Hz.
B3	Group selector	Selects which group to move. The Omicron motor control supports 2 groups of 3 axis each and group 3 which has only 2 axis. Use the channel configuration ("B9") to configure which physical output channel to move for each axis in the selected group.

		The group names can be changed, right-click the group selector and select "Edit Group Name" to modify the name of the selected group.
B4	**Steps XY** and **Steps Z**	Number of steps can be set individually for XY and for Z.
B5	Manual move	The buttons move the motor a defined number of steps in the corresponding direction. The Stop button will stop any movement immediately. If a manual move button is kept pressed, the motor first runs the number of steps specified, then, after a wait time of 0.5s, it will start moving continuously until the button is released. You can use the cursor keys (arrow keys) and PgUp/PgDn to move the motor from the keyboard (works only when the motor control module is the active window). When holding down the Shift key when pressing any of these keys it will move continuously until the button is released. Note: The Move and Stop Keyboard Shortcut Assignment is visualized in a separate document.
B6	Serial port number	Select the **COM-Port** the MSCU is connected to.
B7	Reconnect	Reconnect to MSCU: the status of the MSCU is not constantly polled, reconnect if the communication got lost.
B8	MSCU status	Returns the last status report of the MSCU. Tip: If the MSCU is in power down mode, press Stop or one of the direction buttons to power up.
B9	Channel **Configuration**	Configure which physical channel (**Output**) should be active when moving the corresponding axis and which **Polarity** it should have.
B10	No communication message	Returns message when no communication with the MSCU is established. In this case verify the MSCU serial cable is connected, the right serial port is selected and the try to reconnect with the Reconnect button.

or 2 adjacent lines the motor can be positioned at 8 half-steps.



1	**Manual** Steps	Move single steps in +Z or -Z direction. When holding down a button during 500ms, it will start to move continuously until the button is released.
2	**Steps**	Number of half-steps per click on one of the **Manual** steps buttons.
3	**Engage** and Disengage	Engages the tip (auto-approach). Disengages the tip by retracting it **Disengage** Steps.
4	**Stop** button	Stop motor movement
5	Show **Configuration**	When selected, the configuration settings are displayed.
6	**Position, Reset**	**Position**: Indicates the current position of the motor. **Reset**: Resets the motor **Position** counter.
7	**Motor Speed** (rpm)	Sets the motor speed in rotations per minute (rpm).
8	**Pulsewidth**	Time to activate the digital lines for each half-step.
9	**Engage**	**- Z Threshold:** Withdraw tip & drive motor when tip is within specified threshold from piezo range minimum. If set to 100pm, the tip is withdrawn once it enters a range of 100pm above the piezo range minimum. - **Disengage** steps: Number of half-steps to retract the tip when clicking the disengage button.
10	DIO **Port**	Specifies which digital port to use at the SCCB (Signal conditioning and connector box). Can be A or B.
11	**Enable**	Enables or disables the step motor.
12	Digital Lines	Specifies which lines are used subsequently to move the motor towards the sample (approach). Refer to the SCCB (Signal conditioning and connector box) manual for detailed information on the digital lines and their numbering.

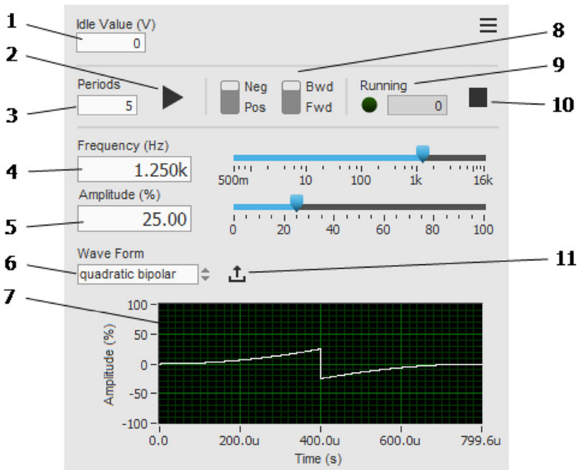
Back Home

Function Generator

The function generator can generate waveforms on the "FAST AO" of the SC5. This can be used for example to drive a piezo motor. Different predefined waveforms are available to generate, or a custom waveform can be loaded from a file.

The FAST AO of the SC5 is unfiltered, unlike the normal AO which are low pass filtered at 40 kHz.

If you need to amplify the generated signal (e.g. to drive a piezo motor), you will need a fast (high-bandwidth) channel in the high voltage amplifier.



1	Idle Value	Sets the output voltage of the FAST AO when the function generator is not running.
2	Go button	Starts the generation of the waveform on the FAST AO.
3	Periods	Number of periods to generate when clicking the **Go** button ("2"). This is the number of steps the motor will move (when used with a piezo motor). Set to -2 for continuous movement (until the **Stop** button ("10") is clicked).
4	Frequency (Hz)	Frequency of the waveform in Hz. When changing the frequency, the time in the graph ("7") will be adjusted accordingly. The frequency can be set in multiples of 0.4768 Hz up to 15.625 kHz.
5	Amplitude (%)	Amplitude of the waveform in percent. 100% corresponds to +/- 10 V at the output of the SC5.
6	Wave form	Select the shape of the waveform to generate. When switching, it'll take some time to download the pattern to the FPGA (indicated through a progress bar).
7	Waveform preview	Preview of the waveform, showing shape, timing, and amplitude.
8	Sign / Direction	**Sign**: Switch between positive and negative sign of the waveform. This will invert the waveform along the y axis. **Direction**: Switch between forward and backward waveform. This will invert the waveform along the x axis (reverse the time).
9	Running	When the function generator is running, the LED will light up and the indicator will display how many periods are left.
10	Stop	Stop function generator.
11	Load custom…	Loads a waveform from a file. Select "custom" in the **Wave Form** selector ("6") to use the custom waveform. The waveform file must have 8192 lines with one value in the range -32768..32767 (16 bit integer, corresponds to -10..10 V at 100% amplitude) per line. The 8192 values form one period of the output signal that will be generated. After successfully loading a file and selecting "custom" in the **Wave Form** selector ("6") you'll see the preview in the graph ("7"). Excerpt of a custom waveform file: -32768 -32752 -32736 -32720 -32704 ...

Back Home

Generic PI controller (optional module)

Controls the signal specified in **Input signal** (it switches to **Demodulated signal** when AC mode is selected) by adjusting the selected User Output channel. Any User Output present in the system can be utilized by the Generic PI Controller. Both the name and SI units of the output channel are set in the appropriate User Output module.

The Generic PI Controller module will adjust the value of the selected User Output to make the Control Signal match its **Set point**. The PI parameters (i.e. proportional gain and time constant) can be adjusted freely.

When AC mode is deactivated, the module will control the specified signal. If AC mode is activated, the selected output will be modulated using the specified settings (**frequency**, **phase**, **amplitude**) and the control signal will be demodulated. In this case, the set point defines the demodulated amplitude, and the controller will adjust the DC value of the output until the demodulated amplitude matches the set point. In AC mode, the **time constant** of the controller should be at least 10 times higher than the modulation period (= 1/mod.frq).

During operation, the Generic PI Controller will take control of the selected User Output if either **Modulation** or **Control Loop** are active. If this case, no user changes can be made to the selected User Output. When **Modulation** and **Control Loop** are both inactive, the User Output module can be operated normally (see User Output –

"Override").

In the screenshot below, a temperature (see **Input signal**) in K is controlled by adjusting the heating power in W.

1	Output Configuration	**Output Signal** : Chooses the user output to be controlled by the PI controller. **Lower Limit** : Sets the lowest available output value for the selected channel (in real-world units) **Upper Limit** : Sets the highest available output value for the selected channel (in real-world units) **Modulation** : If **AC Mode** is activated, we can modulate the output signal **Freq (Hz)** : Frequency (in Hertz) of the output modulation **Ampl. (unit)** : Amplitude (in real-world units) of the output modulation **Phase (deg)** : Phase shift (in degrees) of the output modulation **Auto** : Automatic adjustment of the modulation phase in order to maximize the amplitude of the demodulated signal **-90** : -90 degree phase shift of the output modulation **+90** : +90 degree phase shift of the output modulation
2	Input Configuration	**Input Signal** : Input signal to monitor (in order to align the Input Signal with the Setpoint **AC Mode** : Switches the input configuration to AC component of the incoming signal. When activated, the setpoint is references to the amplitude of the modulation measured on the input signal. **Signal Value** : "Raw" the input signal is used without modification, "Absolute" the absolute value of the input signal is used
3	PI Controller Settings	**Control** Loop : Switches the PI controller on and off **Setpoint** : The value to which the PI controller attempts to align the selected input signal by modifying the selected output signal **Temperature (K)** : The current value of the chosen input signal **Slope** : Controls whether the relationship between output signal and input signal is assumed to be proportional ("Pos.") or anti-proportional ("Neg."). **P-Gain*** & **Time Constant** : The PI parameters (i.e. proportional gain and time constant) can be adjusted freely. The higher the P gain or the lower the time constant is set, the faster the controller reacts to deviations from the control signal to the Set Point. If the P gain is set too high (or the Time constant too low), the controller will start to oscillate. If this happens, adjust the controller parameters. **The Generic PI Controller supports a method where the **upper limit** and **lower limit** values are the only two output values possible (no intermediate values) by setting the P-Gain to "Inf".*

Back Home

Pulse Counter

The pulse counters will display the current frequency of the pulses at the digital input in Hz. This frequency is then available as channel for data acquisition under the name specified in the pulse counter module.

Up to four pulse counters are available. They use the HI-SPEED-DIGITAL IN lines on the RC5.
The counters are running at a sampling rate of 200 MHz, the theoretical maximum detectable pulse frequency is therefore 100 MHz. In practical applications the limit is more like 50 MHz.

CAUTION: Make sure the signal levels match. The RC5 DIO channels work with 3.3V TTL logic, so the amplitude should never exceed 5V with respect to DGND. Higher voltages can cause damage to the hardware.

1	Channel Name	Name of the pulse counter channel as it will appear for data acquisition.

2	Frequency	Displays the current frequency of the pulses at the corresponding digital line.

Back Home

Atom Tracking

The AtomTracking module modulates the xy position of the tip in a circular form and detects the movements in z-direction with a lock-in. Thus the local slope in x and y direction is measured. A feedback controller moves the tip such that it gets locked on a local maximum or minimum. Now, the linear drift can be measured and compensated allowing a drift free measurement afterwards.

1	XY Modulation	- Turn the circular modulation of the tip on or off. The tip will move in a spiral way to avoid jumps when turned on or off, therefore it can take some time until the modulation is fully on or off. When the modulation is turned off, the controller and the drift measurement is turned off as well in case they were on. - **Frequency**: Defines the frequency of the modulation. The unit is Hz, number of rounds per second. Make sure the modulation is not too fast in order that the z-controller can follow and no mechanical resonances will be excited. - **Amplitude**: This is the radius of the circles in the xy-plane. - **Phase**: The reference phase for the lock-in detection. - **+90** / **-90**: Increment and decrement the reference phase by +/- 90 degrees.
2	AtomTracking Controller	Turns the atom tracking controller on or off. When it is turned on it will also turn on the modulation in case it is off.
3	Demod./Control Signal	Signal to demodulate and control. The lock-in will demodulate this signal to calculate the slopes in x and y directions. The controller will try to keep these slopes at zero. The default is to demodulate Z (requires that the Z-controller is on).
4	Switch off Delay (s)	XY position averaging time when switching off the Atom Tracking controller. Higher values result in better reproducibility of the position.
5	Integral	Integral gain of the controller. Needs to be adjusted when the demodulated signal (3) is changed. The unit of the integral gain depends on the demodulated signal.
6	Drift measurement	Starts or stops the drift measurement. During drift measurements the displacement in x and y direction is measured while the time is recorded. By dividing the distance by the time one obtains the linear drift. When the drift measurement is started the values are first reset to zero. To measure drift in z direction the Z-controller must be on.

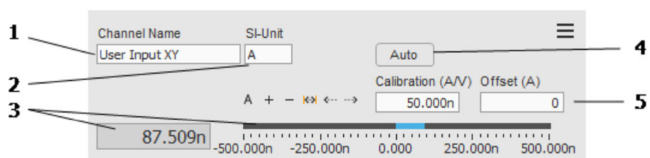
7	Displacement	The travelled distance in x, y and z direction during a drift measurement.
8	Time	The elapsed time during a drift measurement.
9	Apply drift correction	When this button is pressed the linear drift velocities are computed and added to the drift compensation vectors. If the drift compensation was not on it is turned on. After that the drift measurement is turned off and the values reset to zero.
10-11	Drift Compensation	Thus is a duplicate of the controls and indicators of the Piezo Calibration module.
12	Offset	Adds the accumulated drift vector to the scan frame center.
13	Quick Operation Procedure SmarTilt	This button initiates the procedure for a compensation of the sample tilt. This is only possible when demodulating Z (requires that the Z-controller is on). The sample tilt is not indicated in this module, you have to open the Piezo Calibration module and switch to the tab Z-correction. The procedure first sets the amplitude according to the size of the scan frame of the Scan Control module, then adjusts the frequency such that the tip velocity is as specified in the *speed* section of *Scan Control* module, then the slope in x and y is measured and added to the slope compensation of the Piezo Calibration module. After that, the xy modulation is turned off again.
14	Quick Operation Procedure Drift compensation	This button initiates the procedure for a drift compensation: it turns on the modulation, turns on the controller, performs a drift measurement for the time indicated in (15), compensates that drift and finally turns the controller and modulation off again. You can use this button to quickly adjust the drift from time to time between experiments. First you have to make sure that all modulation parameters (1) like frequency, amplitude, phase etc are correctly set. Also the tip should be positioned already close to the local maxima or minima you want to track.
15	Acquisition time	Time to measure the drift for automated drift compensation feature (14).
16	Show/hide graph	Shows or hides the graphical display of the slope/tip path.
17	Graph display	Switches the graph display between Slope / Tip path XY / Tip path Z.
18	Clear graph / Save data	• Clears the tip path. Only available when the graph is in tip path mode. • Saves the data from the graph to file. The format depends on the displayed data.
19	QuickScale buttons	Use the QuickScale buttons to adjust the scale.
20	Slope / Tip path XY / Tip path Z graph	Displays the slope in X & Y direction or the XY tip path or the Z tip path in the graph. The slope is always indicated digitally in the upper right corner of the graph.

Back Home

User definable Input Channel

The input channels not in use by other Nanonis modules can be used as additional input channels to measure arbitrary data.

The signal must be in the range from -10 V to +10 V. The channel can be given a name, and the unit can be specified. To calculate a physical signal out of the input voltage, use calibration & offset.
See the SC5 manual for detailed specifications on the analog inputs.



1	Channel Name	Enter a name for the input channel. This name will appear in other modules such as graphs.
2	Unit	The unit of the signal can be specified here. Enter only SI units without unit prefixes like p (pico) or n (nano).
3	Value indicators	The digital indicator and the slider bar both show the current value of the input channel in physical units. Use the QuickScale buttons to adjust the slider scale.
4	Auto Offset	Click this button for automatic offset correction. When clicking, the current value is considered the offset (assuming the input signal to be 0 at this point).
5	Offset & Calibration	Calibration and offset of the input channel. These values are used to convert the input voltage to a physical signal.

Back Home

User definable Output Channel

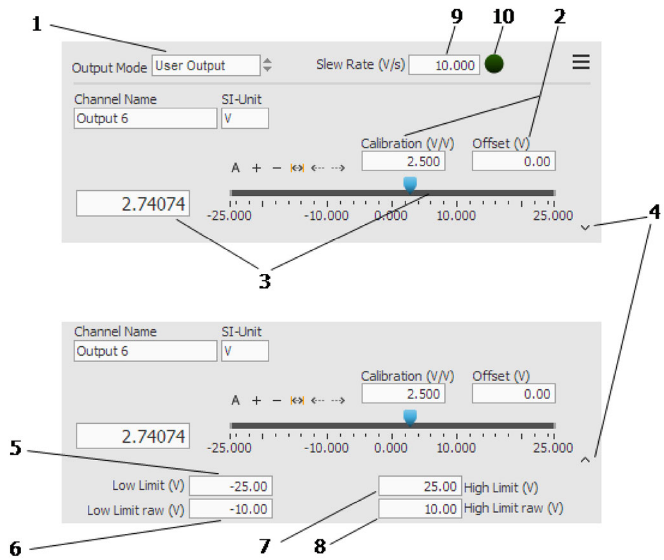
The output channels not in use by other Nanonis modules can be freely configured.

There are four different operation modes: “User output”, “Monitor”, “Calculated Signal” or “Override”.

The output range for the SC5 output hardware is -10V .. +10V.

*Due to differences in the system architecture for **SPM** measurements (Nanonis SPM Control System) and **Transport** measurements (Nanonis Tramea), the user output software modules for these two distinct application areas differ.*

Parameters common to all User Output modes



		Nanonis SPM Control System & Nanonis Tramea
1	Mode switch	Use this selector to set the output modes for this output channel: User output, Monitor or Calc. Signal. The output channel number is visible in the window title bar.
2	Calibration / Offset	Calibration and offset of the output channel. These values are used to convert the output voltage to a physical signal. Throughout the software physical values are displayed.
3	Output value	The digital indicator and the slider can both be used to set the value of the output channel in physical units. Use the QuickScale buttons to adjust the slider scale.
4	Show / Hide Limit settings	Shows or hides the settings for limits (both raw and calibrated) for the user output. When activated, the User Interface shifts upwards to display the settings 5-8 listed below (lower image above). When deactivated, the User Interface shifts back again (upper image above).
5	Low Limit (Calibrated)	This value sets the lowest possible output value for the user output in calibrated units (nA in this example).
6	Low Limit (Raw)	This value sets the lowest possible output value for the user output in Raw units (V). Minimum settable value for this parameter is -10V.
7	High Limit (Calibrated)	This value sets the highest possible output value for the user output in calibrated units (nA in this example)
8	High Limit (Raw)	This value sets the highest possible output value for the user output in Raw units (V). Maximum settable value for this parameter is +10V.
		Nanonis Tramea only
9	Slew Rate (Unit / second)	Sets the maximum allowed rate of change of the physical output of the SC5 (expressed in calibrated units).
10	Slew Rate active	Indicates whether the user output is currently moving to a new value.

User output mode

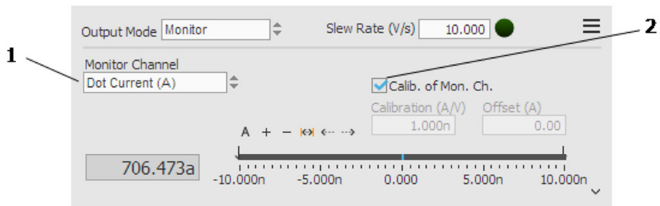
In user output mode the output voltage can be freely set. Any device can be connected to the output connector of the SC5. To convert the output voltage to a physical signal, enter the device calibration and offset in the corresponding fields. The output channel will be available as sweep signal in the Generic sweep module.

1	Channel name	Specify a name for the output channel. This name will appear in other modules such as graphs.
2	SI Unit	Unit of the output channel. Enter only SI units without unit prefixes like p (pico) or n (nano). You can leave in Volts (V) with a calibration of 1 and offset 0 so the physical value is equal to the output voltage.

Use the Generic Sweeper to sweep this (or any other) output.

Monitor output mode

The monitor (output) channel is used to put an arbitrary signal on a physical output so it is accessible on the hardware. The outputs are converted to voltages in the range -10V .. +10V using the specified calibration and offset (default is the monitored channel's calibration). By adjusting the calibration and offset it's even possible to amplify a signal.

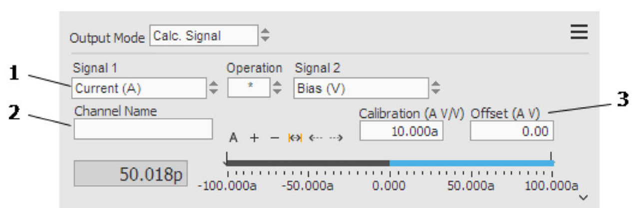


In Monitor mode, the User Output channel will be given the name "MON: monitorchannel", in this example above the channel will be visible in the system as "MON: Current (A)"

1	Monitor Channel	Select the channel to put on the corresponding output (output number is displayed in the window title bar, it is 4 in the picture above).
2	Calib. of Mon. Ch.	When selected the calibration and offset ("3") of the monitored channel are used. When not selected any calibration and offset can be specified. The output voltage is limited to -10V .. +10V, so depending on the calibration and offset values the output might go into saturation.

Calc. Signal mode (Nanonis SPM Control System)

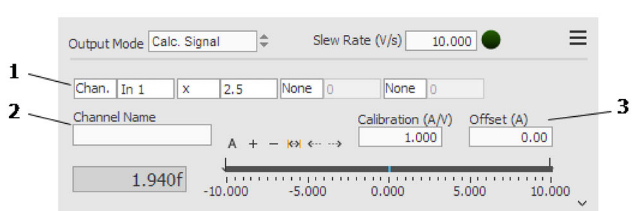
The Calc. Signal mode allows you to calculate a signal based on 2 available signals and basic math operations.



1	Definition of calculated signal	Use these controls to define the calculated signal. The operation between Signal 1 and Signal 2 can be addition (+), subtraction (-), multiplication (*), and division (/). The logarithmic value of Signal 1 can be calculated selecting "log". If "none" is selected, the calculated signal is actually working as monitor output of Signal 1.
2	Channel name	Specify a name for the calculated signal. When left empty, the name is created based on the formula used for the calculated signal.
3	Calibration / Offset	Calibration and offset of the calculated signal. These values are used to convert the physical signal to an output voltage. The default calibration and offset values of the calculated signal are derived from the source signals. When the selected operation is "none", "+", or "-", then the default values are those of Signal 1. When using "*" or "/", the calibrations are multiplied or divided respectively. When using the "log" operation, the default values are calculated computing the logarithm of the calibration of Signal 1. These values can be modified by the user at any time. Note: Be aware that a calculated signal outside the valid physical range of the output will be coerced to it at the corresponding output connector of the SC5.

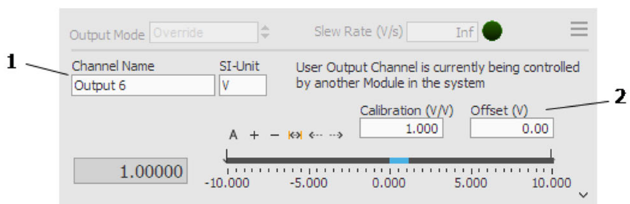
Calc. Signal mode (Nanonis Tramea)

The Calc. Signal mode allows you to calculate a signal based on 2 available signals and basic math operations.



1	Definition of calculated signal	Use these controls to define the calculated signal. For each step of the calculation (of which there are a maximum of four) we can add, subtract, divide or multiply by a constant, an Input or an output channel. Additional functions include absolute value, negation, log and exponent. In this way, complicated relationships between channels can be defined. There is no mathematical operator precedence in operation here; the equations are executed in a strict left-to-right fashion. This is equivalent to defining the calculations as ((((Step 1) Step 2) Step 3) Step 4). The average of Input 1 and Input 2 is defined as "Input 1 + Input 2 / 2". The sum of Input 1 plus half of Input 2 is defined as "Input 2 / 2 + Input 1". The reciprocal of Input 1 is defined as "1 / Input 1".
2	Channel name	Specify a name for the calculated signal. When left empty, the name is created based on the formula used for the calculated signal.
3	Calibration / Offset	Calibration and offset of the calculated signal. These values are used to convert the physical signal to an output voltage. The default calibration and offset values of the calculated signal are derived from the source signals. When the selected operation is "none", "+", or "-", then the default values are those of Signal 1. When using "*" or "/", the calibrations are multiplied or divided respectively. When using the "log" operation, the default values are calculated computing the logarithm of the calibration of Signal 1. These values can be modified by the user at any time. Note: Be aware that a calculated signal outside the valid physical range of the output will be coerced to it at the corresponding output connector of the SC5.

Override mode



When a channel is in Override mode, this means that another module has claimed control of the user output (such as Generic PI Controller). The user has much reduced ability to change the parameters of the channel in this mode. Only the name, unit and calibration may be changed. Due to possible operation limitations, slew rate and output value are unavailable. In order to put the user output back into normal operation, the module which has taken control must release the channel.
Please note that programming interface functions also have no permission to access certain parameters if the user output is in override mode. Errors may be returned as a result.

1	Channel Name	Specify a name for the output channel. This name will appear in other modules such as graphs.
2	Calibration / Offset	Calibration and offset of the output channel. These values are used to convert the output voltage to a physical signal. Throughout the software physical values are displayed.

Back Home

Generic Sweep

The Generic Sweep module is used to measure several channels while sweeping a user-defined channel.

The data recorded are stored into one of four buffers (Plot 1-4) which are displayed in the two graphs. You can save the buffers either manually or automatically after the sweep is done.

The sweep is controlled from the host computer (*i.e.* not from the real-time controller), therefore limiting the acquisition speed to about 50 pt/s.

The Sweep Signal is ramped point by point from the start limit to the end limit using the Max. Slew Rate and measuring the selected signals during Period Time at every point before ramping to the next point.
The detailed step by step function is as follows:

The current value of the Output to be swept is recorded (Reset Value).

Set the Sweep Signal target value to the start sweep limit: the Sweep Signal will sweep to the start limit at Max. Slew Rate (Units/s).

Wait Initial Settling time (s).

Acquire and average the Sweep Signal & the signals selected in the Channels list during Period Time (s).

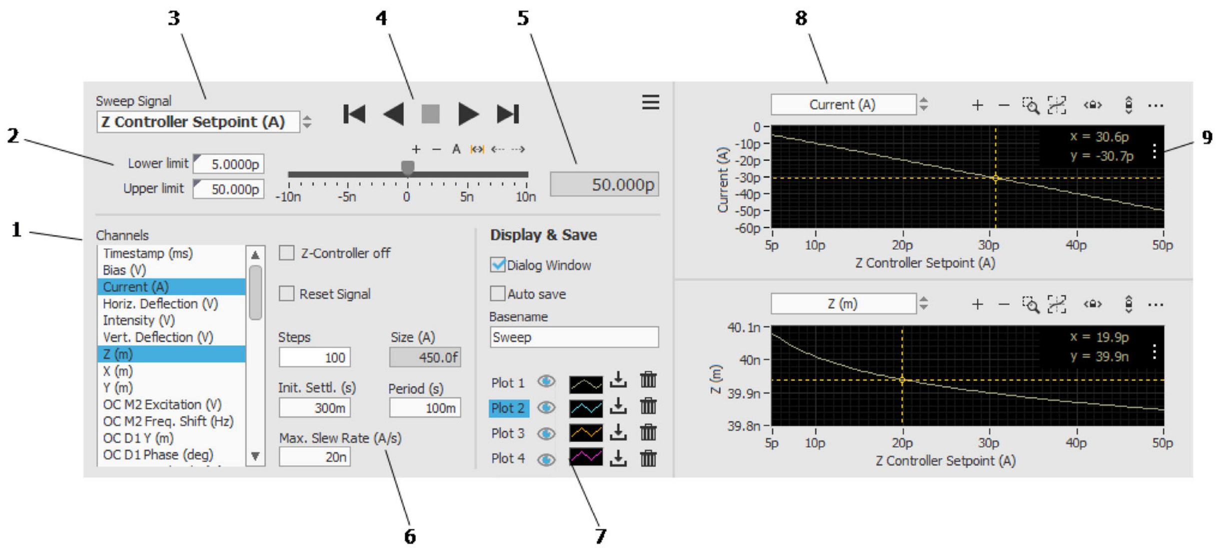
Set the Sweep Signal target value to the next point at Max. Slew Rate (Units/s).

Repeat steps 4) and 5) until the last point of the sweep.

7) If Reset Signal is checked, we set the Sweep Signal to the Reset Value recorded in 1).

The Animations Period and the Measurements Period are set to 20ms in the TCP receiver while measuring.

Note: Generic graph handling and numeric control handling are explained in the user interface section.



1	Channels	Select the channels you want to record. Tip: you can select more than one channel by pressing the Ctrl-key while selecting the channels with the mouse.
2	Limits	Define the **Lower** and **Upper** limits of the **sweep signal**. The **Lower** value must be smaller than the **Upper** one. Otherwise, the fields will blink and it won't be possible to start a sweep.
3	Signal to sweep	Select the signal you want to sweep.
4	Control	Start the sweep, from **Lower** limit to **Upper** limit. Start the sweep, from **Upper** limit to **Lower** limit. When starting a sweep, if the signal value is different from the starting value, the signal will first ramp to that point at the rate defined by **Max. slew rate**. Stop the sweep. Ramp to the **Lower** or **Upper** limit at the rate defined by **Max. slew rate**.
5	Sweep signal value	The digital control and the red slider can be used to manually set the signal value. They are coupled with the controls on their own module. The yellow line represents the sweep range as defined by the **Limits**.
6	Acquisition parameters	**Reset Signal**: When selected, the signal is set to its initial value (*i.e.* value before starting the sweep) when the sweep is done. **Z-Controller off** (SPM only): When selected, the Z-controller is switched off during the ramps and for the duration of the sweep. It is turned on afterward (if previously on). Before turning off the controller, the **Switch off Delay** and **TipLift** values (in the Z-controller module) are taken into account. Thus, to get a reproducible Z position, it is recommended to set a non-zero **Switch off Delay**. **Steps**: Number of measurement points to record **Size**: define the step size. **Settling Time (s)**: Time to wait just before starting the sweep. **Period (s)**: Defines the measurement speed. Higher values result in more accurate data because of the oversampling. Its lower limit and increment are defined by the Measurements Period (s) parameter in the TCP receiver. **Max. Slew Rate (X/s)**: Maximum rate at which the **sweep signal** changes when ramping to the starting point. If the **switch** is set to **Size** but the sweep range is such that the number of steps is not an integer, then the **Number** field will turn orange. This warns that the last step of the sweep will be smaller than the value specified by the user.
7	Save options & Plot	**Dialog**: when selected, a dialog window will open before saving data. **Auto**: when selected, the data are automatically saved after each sweep. **Basename**: defines the file basename when saving sweeps. The file names will be "*Basename*NNN.dat" where NNN is an automatically increasing number. Tip: special tags can be used in the Basename to include additional timing information: %a (abbreviated weekday name), %b (abbreviated month name), %c (locale-specific date/time), %d (day of month), %H (hour, 24-hour clock), %I (hour, 12-hour clock), %m (month number), %M (minute), %p (a.m./p.m. flag), %S (second), %x (locale-specific date), %X (locale-specific time), %y (year within century), %Y (year including century) An optional # modifier before the format code letter (e.g. %#d) removes the leading zeros. Additionally, the tag %R used in the basename removes the index number 1. If more images are saved afterwards with the same basename, the %R tag won't have any effect (i.e. if %R is used in the basename and there is no file with the same basename saved in the session directory, then the saved filename won't include the index number). Example: %y%m%d_MySample **Plot**: select to which plot (buffer) the next sweep will be saved to. : show/hide the plot curve on the graphs. : save the data to ASCII file (.dat).

		· : clear the plot.
8	Signal to display	Choose the signal you want to display in the graph.
9	Cursor	Display the cursor coordinates. The cursor can be hidden/shown by right-clicking on the graph and selecting Cursor>Show. The cursor can be moved from one curve to the other by dragging its crosshair. Note: Generic graph handling including cursor handling is explained in the user interface section.

Back Home

Frequency output for Scala controller

The *Scala* controller has a +/- 10V input that corresponds to a **frequency shift** of -732..+732Hz. In AFM mode, the *Scala* controller will control this input to 0.

This module lets you define the **Setpoint** in Hz and it will apply the corresponding voltage on output 4. It subtracts the **Setpoint** ("2") from the current frequency shift and converts it to the correct range, i.e. the voltage of output 4 is calculated as follows:
U = (Frq.Shift - Setpoint)/(-73.2).

The module also offers the SafeTip feature. Specify a condition on which the tip should be retracted. If the condition is met, the output voltage is set to +/- 10 V (sign is configurable). This will lead to a tip retraction from the *Scala* z-controller.

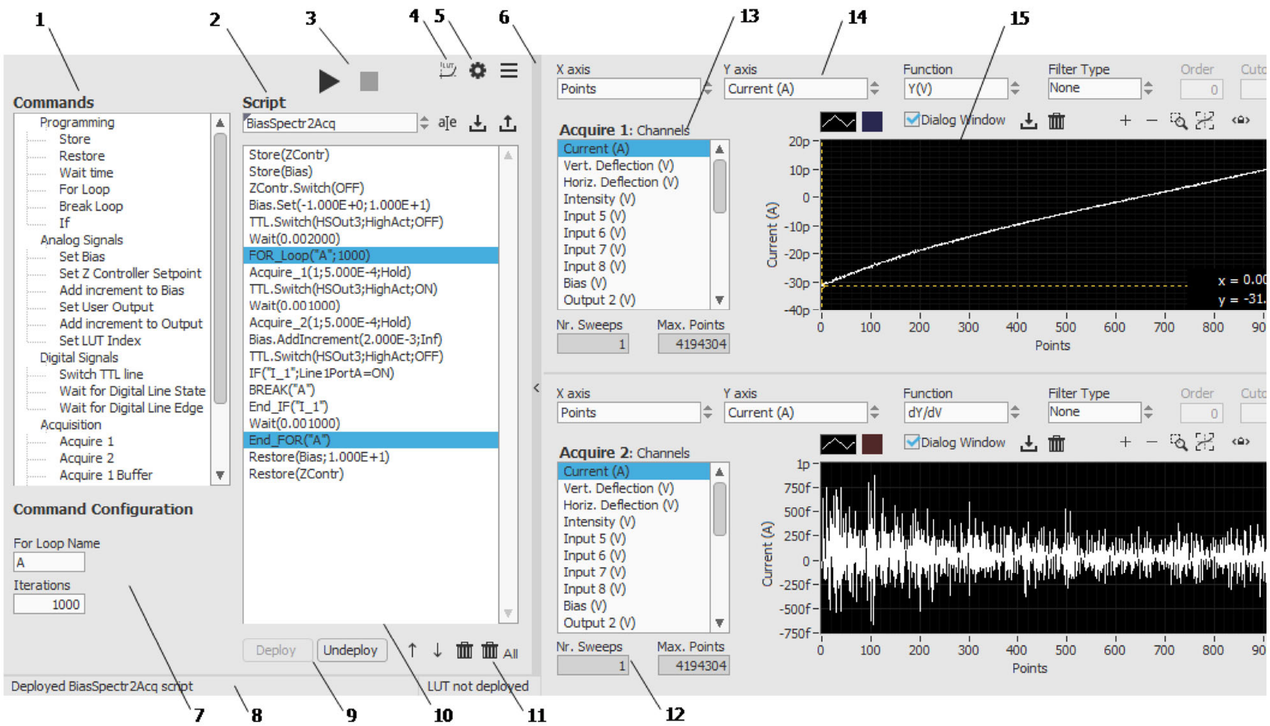
1	Calibration	Set the calibration of the **Frequency Shift**. The default value corresponds to the *Omicron Scala* calibration.
2	Setpoint (Hz)	Set point for the **Frequency Shift** in Hz.
3	Frequency Shift	Displays the current frequency shift as in the oscillation control module.
4	Output voltage	Voltage on output 4. The text above describes the calculation.
5	SafeTip on/off	Switch the SafeTip feature on/off.
6	Show/hide SafeTip	Click the arrow to show/hide the SafeTip configuration
7	Reset SafeTip	Only visible if a SafeTip event occurred. Click this button to reset the output to the normal value. On a SafeTip event, the output will remain at +/- 10V until this button is clicked or the SafeTip mechanism is switched off.
8	SafeTip event indicator	Lights up if the SafeTip condition is met. Use the **Reset** button ("7") to set the output back to the normal value.
9-12	SafeTip condition	The SafeTip condition is set up with the controls 9-12. If the specified **SafeTip Signal** ("9"), selectable whether **absolute** or **bipolar** ("10"), exceeds or falls below ("11") the specified threshold ("12"), a SafeTip event is triggered and the output voltage is set to the value specified in "13".
13	SafeTip output value	If a SafeTip event occurs, the output voltage is set to + or - 10V. Set the sign here so the z-controller of the *Scala* will retract the tip.
14	SafeTip signal value	Shows the current value of the selected SafeTip signal ("8").

Back Home

Script Tool

The Script Tool allows to create scripts containing a set of commands (up to 255 per script) which can be deployed to the real time system and execute at real time speed.

It is possible to configure up to 20 scripts, and half of them can be used in the Multipass configuration module, so they can also run at the beginning of a new pass. See the help of the Multipass module for more details.
All scripts can be deployed to the real time system and they can stay deployed in the real time system during the entire execution of the Nanonis software, but **only one script is allowed to run or be deployed at a time**. This is very important for safety reasons, as two scripts running simultaneously could interfere with each other and produce undesired results.



1	List of Commands	Displays the list of available commands, and it varies depending on the licensed modules. Drag the desired command and drop it into the **Script Editor** or double click it to automatically add it at the current position of the script. The commands are grouped by category (Programming, Analog Signals, Digital Signals, Acquisition, and Control). The following list shows the complete list of commands: - **Store**: it stores the current value of Bias/Z-Setpoint/User Output X or the current status of the Z-Controller/Kelvin Controller/PLL output/PLL2 output. - **Restore**: it applies the stored value to Bias/Z-Setpoint/User Output X or the stored status of the Z-Controller/Kelvin Controller/PLL output/PLL2 output. - **Wait time**: it waits the specified amount of seconds. - **For Loop**: it creates a For Loop with the selected number of iterations. Any script command can be used within a loop, even nested loops. - **Break Loop**: it forces the script to break the loop (without waiting until the total number of iterations is executed) and go to the line after End_FOR. This command cannot be used outside a For Loop. - **If**: if the condition is met (any signal can be used in the condition), the code within the If command is executed. Otherwise, the next command to execute corresponds to the line after End_IF. - **Set Bias**: it applies the specified Bias value at the specified slew rate (V/s). A single value can be applied or a Look Up Table can be used to set programmatically the next value from the LUT. Have a look at the Script-LUT Editor to know how to configure the LUT. - **Set LUT Index**: it sets the index to use from the deployed LUT. If not specifically set, the index is automatically increased by 1 whenever used in Set Bias or Set User Output commands during the execution of a script and among scripts of the same Scan Grid/Subgrid. When running a different script (and outside a grid/subgrid), the LUT index starts from 0 by default (unless Set LUT Index is used to set a different index). - **Set Z-Controller Setpoint**: it applies the specified Z-Setpoint value at the specified slew rate (increment/s). - **Add increment to Bias**: it adds the specified increment to the Bias value at the specified slew rate (V/s). - **Set User Output**: it applies a value to the selected User Output at the specified slew rate (units/s). A single value can be applied or a Look Up Table can be used to set programmatically the next value from the LUT. Have a look at the Script-LUT Editor to know how to configure the LUT. - **Add increment to Output**: it adds the specified increment to the selected User Output value at the specified slew rate (units/s). - **Switch TTL Line**: it switches on/off the configured High Speed TTL output with the selected polarity. - **Wait for Digital Line State**: it waits for the selected state (on/off) of a TTL line (port A or B). It will stop waiting after the timeout. - **Wait for Digital Line Edge**: it waits for the selected edge (rising/falling) of a TTL line (port A or B). It will stop waiting after the timeout. - **Acquire 1**: it acquires the specified number of points of the channels selected in the **Acquire 1: Channels list**, and each point is averaged during the selected integration time. It stores the acquired data in the 1st Buffer of the Script module. It is possible to wait until the acquisition finishes or just continue to the next script line. The acquired data will be displayed in the top graph of the acquisition section. By default, all data is displayed as a single sweep in the same plot in the graph. In order to assign the data from different

		Acquire 1 commands to different sweeps, use **Acquire 1 Buffer** command. The maximum number of data points which are allowed to acquire per script is 4.194.304. If this is exceeded, the remaining data won't be acquired. - **Acquire 2**: it acquires the specified number of points of the channels selected in the **Acquire 2: Channels list**, and each point is averaged during the selected integration time. It stores the acquired data in the 2nd Buffer of the Script module. It is possible to wait until the acquisition finishes or just continue to the next script line. The acquired data will be displayed in the bottom graph of the acquisition section. By default, all data is displayed as a single sweep in the same plot in the graph. In order to assign the data from different Acquire 2 commands to different sweeps, use **Acquire 2 Buffer** command. The maximum number of data points which are allowed to acquire per script is 4.194.304. If this is exceeded, the remaining data won't be acquired. - **Acquire 1 Buffer**: this command takes care of organizing the 1^{st} Buffer data in different sweeps along the script, so that each sweep is displayed as a different plot in the top graph of the acquisition section. Using this command at a certain script line decides in which sweep the next acquisition data from the 1^{st} Buffer (the data that will be acquired in the next Acquire 1 command) will be. There are 2 options for this command: **Set Buffer to Sweep N°,** and **Increment Sweep N°.** The first one allows setting the next acquisition data to a specific sweep number (starting from sweep n°1), whereas the second option increases the sweep number by one. If no Acquire 1 Buffer command is used along the script, all acquired data will be assigned to the same sweep. This command cannot be used inside an If command. The number of sweeps configured in the script assigned to the 1^{st} Buffer is displayed in the graph section below the list of channels to acquire. The maximum number of points allowed per sweep is displayed right next to it. - **Acquire 2 Buffer**: this command takes care of organizing the 2nd Buffer data in different sweeps along the script, so that each sweep is displayed as a different plot in the bottom graph of the acquisition section. Its description is equal to Acquire 1 Buffer command's, but applied to the 2nd Buffer. This command cannot be used inside an If command. The number of sweeps configured in the script assigned to the 2nd Buffer is displayed in the graph section below the list of channels to acquire. The maximum number of points allowed per sweep is displayed right next to it. - **Switch Z-Controller**: it switches the Z-Controller on/off. - **Switch Kelvin controller**: it switches the Kelvin controller on/off. Switching on the Kelvin controller switches on the Kelvin modulation. - **Switch Kelvin modulation**: it switches the Kelvin modulation on/off. Switching off the Kelvin modulation switches off the Kelvin controller. - **Switch PLL output**: it switches the oscillation controller AC output on/off. When switching off the AC output, the amplitude controller is switched off. - **Switch PLL2 output**: it switches the oscillation controller 2 AC output on/off. When switching off the AC output, the amplitude controller is switched off. - **Start Experiment**: it starts Bias/Z-Spectroscopy (the module should already be configured before running the script, else the script cannot be deployed and the script line appears in red as warning). It is possible to wait until the experiment finishes or just continue to the next script line.
2	Script selector	Displays the list of scripts. Right-click the script selector in order to edit its name. Load script. Save script. These buttons allow saving the selected script to disk, or loading a different one replacing the currently selected. By default the scripts are saved in "<All users application data>\Nanonis\Certificate <certificate no>\Scripts" folder (in Windows XP: "C:\Documents and Settings\All Users\Application Data\Nanonis\Certificate <certificate no>\Scripts"). When saving to Session, not only the selected script but all of them (if they have commands) are saved in the session settings file. When loading from Session, the current scripts in the Script Tool are replaced by those from the session settings file, and automatically deployed to the real time system.
3	Execution Buttons	Starts the execution of the selected script. Stops the execution of the running script.
4	Open LUT Editor	Opens the Script-LUT Editor to create look up tables.
5	Script Options	Opens the Script Options window where the global parameters (those applied to all commands and scripts) can be set.
6	Show/Hide Acquisition Section	This button expands or collapses the window so that the acquisition section is visible or not visible respectively.
7	Command Configuration	Displays the configuration of the command selected in the **Script Editor**. It varies depending on the selected command.
8	Script Status Bar	When no script is running, it shows the status of the currently selected script (deployed or undeployed), and the background color is white. When a script is running, it shows the name of the running script, and the background color is green. At this time, no other script can be deployed or executed until the running script stops.
9	Deploy/Undeploy	**Deploy**: Sends the commands of the currently selected script to the real time system. The longer the list of commands, the longer it will take to deploy the script. Once deployed, the script can be executed (run). **Undeploy**: Deletes the currently selected script from the real time system.
10	Script Editor	Shows as many lines as commands added to the script (up to 255) from the **List of Commands**. The script can be manually modified by selecting a line and changing the configuration of the selected command in the **Command Configuration** section, or by using the **Edit Buttons** to delete lines or re-ordering the lines within the script.
11	Edit Buttons	Deletes the selected line.

		All Deletes all lines from the script. ↑ ↓ Move the selected line up or down within the script.
12	Number of Sweeps	The number of sweeps configured in the script assigned to the each buffer is displayed in the graph section below the list of channels to acquire. The maximum number of points allowed per sweep is displayed right next to it.
13	Acquired Channels List	Displays the selected channels to acquire during the script execution, when using the Acquire 1 command (channels list at the top half) and the Acquire 2 command (channels list at the bottom half).
14	Acquisition Display	It is possible to set any of the acquired channels as X axis or Y axis, apply filters (Gaussian, Butterworth,…) or functions like dY/dV.
15	Acquisition Graph	The top and bottom graphs show the different sweeps as distributed by the Acquire 1 Buffer and Acquire 2 Buffer commands respectively. Saves the data stored in the corresponding buffer (Buffer 1 or Buffer 2). Each plot is saved into a different .dat file by default, but all plots from Acquire 1 and Acquire 2 buffers from the same script could be saved into the same file, if configured like that in the Script Options. When the flag called **Dialog Window** is selected, a dialog window will open before saving data. Deletes the data from the graph display.

Back Home

Data Logger

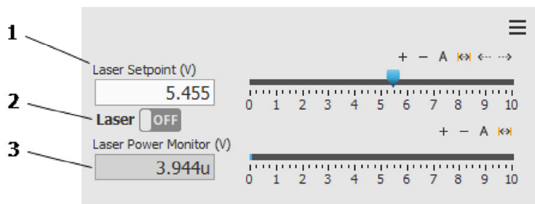
This module allows the user to select a set of channels, start their acquisition at a specific averaging rate, and save them into file for as long as the user sets the acquisition duration. The maximum sampling rate is given by the **Signals Period (s)** set in the TCP Receiver, but additional averaging is available.

1	Channels	Select the channels to acquire and save.
2	Control Buttons	Starts/Stops the acquisition.
3	Time Units	Sets the units of the **Acq. Duration** and **Acq. Time** parameters, where s=seconds, m=minutes, and h=hours.
4	Acquisition Mode	The mode sets the acquisition duration. If **Timed**, the selected channels are acquired during the **Acq. Duration** time or until the user presses the Stop button. If **Continuous**, the selected channels are acquired continuously until the user presses the Stop button. The acquired data are saved every time the averaged samples buffer reaches 25.000 samples and when the acquisition stops.
5	Acquisition Indicators	**Saved Points**: Number of points (averaged samples) already saved into file. This parameter updates while running the acquisition. **Acq. Time**: Time (units defined by **Time Units**) already passed since the acquisition started. **Sampl. Rate**: Sampling rate defined by the **Signals Period (s)** set in the TCP Receiver module and the **Averaging** value.
6	Averaging	Set how many data samples (received from the real-time system) are averaged for one data point saved into file. By increasing this value, the noise might decrease, and fewer points per seconds are recorded.
7	Save Options	**Basename**: defines the file basename when saving averaged samples. The file names will be "BasenameNNN.dat" where NNN is an automatically increasing number. **File #**: index of the next saved file. **Comment**: text you want to include in the file header. Tip: special tags can be used in the Basename to include additional timing information: %a (abbreviated weekday name), %b (abbreviated month name), %c (locale-specific date/time), %d (day of month), %H (hour, 24-hour clock), %I (hour, 12-hour clock), %m (month number), %M (minute), %p (a.m./p.m. flag), %S (second), %x (locale-specific date), %X (locale-specific time), %y (year within century), %Y (year including century) An optional # modifier before the format code letter (e.g. %#d) removes the leading zeros. Additionally, the tag %R used in the basename removes the index number 1. If more files are saved afterwards with the same basename, the %R tag won't have any effect (i.e. if %R is used in the basename and there is no file with the same basename saved in the session directory, then the saved filename won't include the index number). Example: %Y%m%d_MyFile

Back Home

Laser control

The laser control module can control a laser diode driver for Omicron AFM.

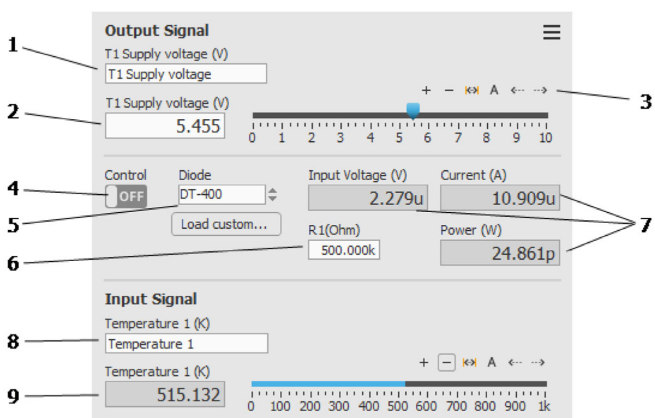


1	Laser setpoint	Setpoint of the laser diode driver in volts.
2	Laser on/off	Switch the laser on or off.
3	Laser power	Displays the laser power.

Back Home

Temperature Measurement

The temperature measurement modules can measure temperature in K from Lakeshore sensors and supply them a controlled current of 10 uA.



1	Output name	User-defined name for the output voltage used to supply current to the temperature sensors.
2	Output voltage	When control is off, the user can adjust the output voltage with the slider, and when control is on, the output voltage is automatically calculated to supply 10 uA to the sensors.
3	QuickScale	Use these buttons to adjust the scale of the sliders.
4	Control on/off	Switch the controlled output voltage on or off.

5	Diode type	Select the temperature diode in use. This will load the corresponding lookup table to translate the voltage read into a temperature.
		If none of the options work for your diode, you can load a custom lookup table from a text file. The file must contain two data columns in ASCII format, the first column are the temperature values (in K), the second column the corresponding voltages (in V). The two columns must be separated by spaces or tab(s). There can be a text header or a third column, both of which are ignored.
		Excerpt from a valid file:
		`1.400   1.64429`
		`1.500   1.64299`
		`1.600   1.64157`
		`1.700   1.64003`
		`1.800   1.63837`
		…

6	R1	User defined resistance that is connected in series with the temperature sensor.
7	Input values	Display the measured voltage (V) and the calculated current (A) and power (W).
8	Input name	User-defined name for the input temperature signal.
9	Input temperature	Display the calculated temperature based on the predefined lookup table of the sensor.

Back Home

Potential Compensation

This module calculates the voltage needed to compensate for an electrostatic potential in the vertical deflection signal.

1	Compensation Output name	User-defined name for the output voltage used to compensate the electrostatic potential.
2	Compensation Output value control	The user can manually adjust the output voltage with the slider. (Not possible during automatic potential compensation procedure.) The digital indicator and the slider bar both show the current value of the input channel in physical units. Use the QuickScale buttons to adjust the slider scale.
3	Auto Compensate	Start automatic potential compensation procedure. When the **Auto Compensate** procedure is started, the output voltage is automatically calculated through the following procedure: 1. Set output voltage to –V **limit (V)** 2. Start adding **Increment (V)** to the output voltage until the vertical deflection gets out from saturation 3. Once the vertical deflection is not saturated, start subtracting **Gain***deflection from the output voltage until the vertical deflection becomes zero (i.e. it is below 10mV).
4	Compensation parameters	- **V Limit (V)**: Defines the limits of the output voltage range used to start the calculation of the potential compensation. - **Increment (V)**: Defines the voltage increment added continuously to –V limit until the vertical deflection gets out from saturation (i.e. vertical deflection is below 5V) - **Gain**: Factor to multiply the deflection by when calculating the potential compensation in the non-saturation range.
5	Deflection Input	User-defined name (in the Beam Deflection module) for the vertical deflection. The unit of the signal can be specified in the **SI-Unit** field. Enter only SI units without unit prefixes like p (pico) or n (nano).
6	Deflection configuration	- **Gain**: Select the gain of the deflection preamplifier (1, 10, 100, 1000). - **Calibration / Offset**: Calibration and offset of the deflection input channel. These values are used to convert the input voltage to a physical signal. The values for calibration and offset are saved individually for every preamplifier gain. - **Auto**: Click this button for automatic deflection input offset correction. When clicking, the current value is considered the offset (assuming the input signal is 0 at this point).
9	Deflection Input value indicator	Displays the value of the vertical deflection.

Back Home

Data File Viewer

The Data File Viewer displays Nanonis ASCII Data File Format (dat) acquired, for example, with a Sweep module like the Generic Sweep module.

Note: Generic graph handling and numeric control handling are explained in the user interface section.

1	Browse settings	Click on the Browse button to select a file or a folder. Selecting a file will open it in the graph and set the **Filter pattern** to match its base name so that the File list contains all the image of that series. Selecting a folder will set the Filter pattern to "*" so that the File list contains all the files present in that folder. The Filter pattern follows the Windows matching rules: use the question mark character (?) to match any single character and the asterisk character (*) to match any sequence of one or more characters. Note: The selected path will appear in the window title bar. Tip: Both Filter Pattern and File allow dragging & dropping a folder/file into them.
2	**File**	Select which File to display. The list contains the files in the selected folder that match the Filter pattern. Use the up/down arrows to quickly browse through the list.
3	**Attributes** from Plot	Displays Attributes like experiment name, date and other (user-defined) key-value-pairs saved with the file.
4	**Auto Shift** **Treat fwd & bwd as one**	Shifts the selected files in the graphs automatically one up respectively down on a **File** selector change. Shows the fwd and bwd data of a sweep as one plot in the graphs.
5	**Data Filtering**	**Filter type**: select a filter to smooth the displayed data. Extra parameters are available for some filters only: **Order**: filter order of a dynamic filter or width (in number of points) for the gaussian filter. **Cutoff frq**: cutoff frequency for dynamic filters. This assumes the acquired data have a sampling frequency of 1 Hz, regardless of the sweep timing. This parameter is very important when filtering as it defines how "strong" the filter is.
6	Signal selector **Function**	Choose with signal to show in the graph. Select the function to be applied in the graph.
7	Zoom and Pan	Buttons for the graph:

		- Zoom in/out - Marquee Selection Zoom - Auto Fit Scales Drop down menu for more Scale and Graph options. Note: Generic graph handling is explained in the user interface section.
8	X-Axis selector	Choose which Signal to show in the X-Axis of the graph.
9	Cursor	Display the cursor coordinates. The cursors can be hidden/shown by right-clicking on the graph and selecting Cursor>Show. The color indicates on which curve the cursor is snapped to. Tips: the cursor can be moved from one curve to the other by dragging its crosshair, if Cursor>Stick to one curve is deselected in the right-click menu. It is also possible to make it jump to the other curve by right-clicking on the graph and selecting Cursor>Jump to Other Curve. If you drag the cursor using its vertical line, it won't jump to the other curve. The **Band RMS** cursor type is only valid for Spectrum. It shows the rms value in the frequency band defined between both cursors with filter and function applied. Note: Generic graph handling including cursor handling is explained in the user interface section.
10	**Same X-Axis Range**	Keeps the X-Axis of both graphs synchronized.

Back Home

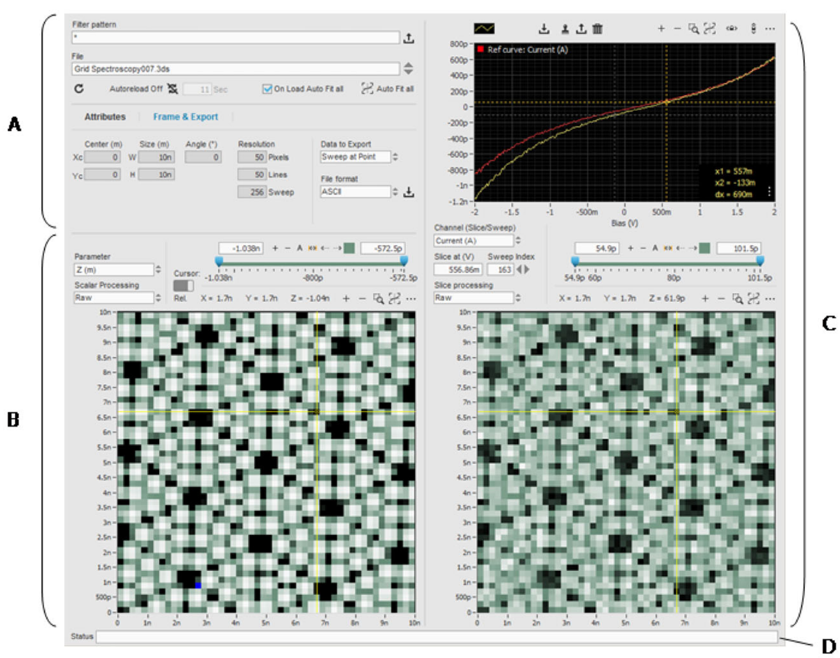
Binary File Inspector

The Binary File Inspector displays 3-dimensional binary data sets (.3ds) acquired, for example, with the Grid mode in the scan control module. It displays the topography, the sweep curve at a single point and a slice through the data at a specific value. For example, if you acquired bias spectroscopy on a grid, it allows you to display the current (at each point) at a given bias voltage.

You can even open a file that is currently written to (the currently running experiment), to update the graphs just click on the Reload button or use the Auto Reload option. The data can be exported in several formats.

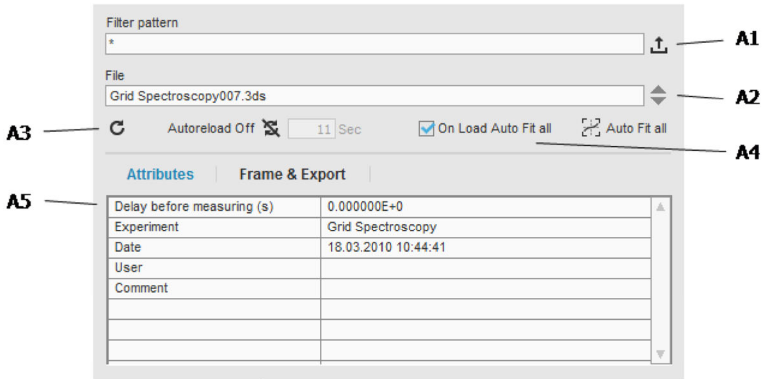
Note: Generic graph handling and numeric control handling are explained in the user interface section.

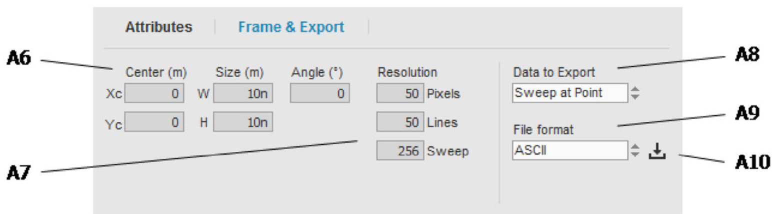
Overview



A	File information & export	Selection of the file to open / display of information of that file / export of data
B	Topography section	Displays the topography (Z signal) or another parameter on the grid.
C	Sweep & Slice	- Upper graph: sweep data at the selected point - Lower graph: slice through the data at a specified value
D	Status bar	Displays error or other status messages.

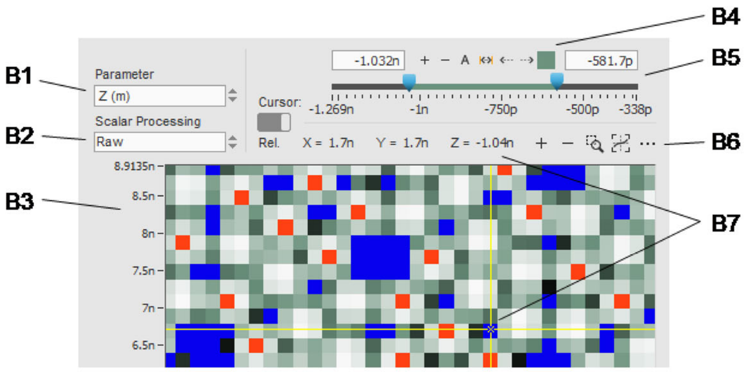
A - File Information & Export





A1	Browse settings	Click on the Browse button to select a file or a folder. Selecting a file will open it in the graph and set the **Filter pattern** to match its base name so that the File list contains all the image of that series. Selecting a folder will set the Filter pattern to "*" so that the File list contains all the files present in that folder. The Filter pattern follows the Windows matching rules: use the question mark character (?) to match any single character and the asterisk character (*) to match any sequence of one or more characters. Note: The selected path will appear in the window title bar. Tip: Both Filter Pattern and File allow dragging & dropping a folder/file into them.
A2	**File**	Select which File to display. The list contains the files in the selected folder that match the Filter pattern. Use the up/down arrows to quickly browse through the list.
A3	Reload	If you opened a file that is currently written to (the data file of a running experiment), click Reload every time you want to update the graphs. You can also check Auto Reload to automatically update graphs at a user-defined interval.
A4	**Auto Fit all**	Click **Auto Fit all** to auto fit all Graphs (Sweep, Slice and Topo Graph) including their Z-Axes. Select **On Load Auto Fit all** to auto fit all when loading a new File (not for Reload).
A5	**Attributes**	Displays Attributes like experiment name, date and other (user-defined) key-value-pairs saved with the file.
A6	Acquisition grid	Dimensions of the grid in physical units, i.e. **Center** position, **Size** with width/height in meters and rotation **Angle** in degrees.
A7	Grid **Resolution**	Grid resolution in number of points (Pixels/Lines) and points of the sweep measurement performed at each grid point.
A8	**Data to Export**	Choose which data to export. Options are: - **Parameter (Topo)**: left lower graph - **Slice**: lower right graph - **Sweep**: upper right graph
A9	Export **File Format**	Choose file format of **Data to Export**. Options are: - **png or bmp**: exports the graph as displayed, including scales (png by default) - **png or bmp - image only** (not available for sweep): exports the data as an image (png by default) with *Grid Dim X* x *Grid Dim Y* pixels. No scale will be visible in the exported - image. - **sxm (scan file)** (not available for sweep): exports the data as a scan data file (.sxm, like the files created by the Scan control module). - **ASCII**: exports the data in ASCII format. For the sweep data, this will be a file with a header and several tab-separated data columns (a column corresponds to one acquired signal in one direction). For the parameter (topography) or slice data the header will be the same as for the sxm files, but with tab-separated ASCII data. - **ASCII (all)**: exports all the sweep data in ASCII format at once.
A10	Save (Export)	Exports the data selected in "A8" in the format specified in "A9".

B - Topography Section



B1	Parameter	Choose which Parameter should be displayed in the graph ("B3"). Usually this is Z (m), so the graph displays the topography data.
B2	Processing	Processing of the data (like in Scan monitors).
B3	Graph	Graph displaying the selected parameter (usually topography).
B4	Color	Select the color in the middle of the color ramp. The ramp will be black-selected color-white. Out of range colors are blue and red.
B5	Color range	Adjusts the range of the selected parameter over which the color ramp is distributed. The two digital controls replicate the selection in the slider. Use the QuickScale buttons above the slider to modify the slider range.
B6	Zoom and drop-down menu	Buttons for the graph: - Zoom in/out - Marquee Selection Zoom - Auto Fit Scales Drop down menu for more Scale and Graph options. Note: Generic graph handling is explained in the user interface section.
B7	Cursor	Move the cursor using the mouse or the cursor controls above the graph. The cursor position defines which sweep curve is displayed in the sweep graph ("C3"). The cursor positions of the topography graph and the slice graph are synchronized. The cursor switch Rel./ Abs. allows to change the cursor coordinates between relative (grid size) and absolute (piezo X/Y-positions). Tip: Move mouse over a graph and use keyboard Ctrl + cursor keys Up / Down / Left / Right to move the graph's cursor.

C - Sweep & Slice

C1	Save and Background functions	- Save: save the present sweep curve to an ASCII file (.dat format). - Paste: paste the present sweep curve to the graph's background. - Load: load a saved curve and paste to the graph's background (reference curve). - Clear: delete the background curve.
C2	Background Reference curve	Sweep of the loaded or pasted curve. This curve can be used as a reference to compare the present sweep to a previously acquired one. This is useful to compare noise levels and peaks amplitude. Tip: the color of this curve can be changed by clicking on the small box next to the curve legend (top left corner of the graph).
C3	Sweep graph	Displays the sweep curve at the point specified by the cursors in the topography and slice graphs ("B8"/"C11").
C4	Slice reference value and Sweep index	Defines the value at which the slice is generated. For example, if a bias spectroscopy measurement (acquiring current) was done at each point of the grid, you can select at which bias voltage you want to generate the slice. You will get a graph displaying the current values at the specified bias voltage for each point of the grid. To change the reference value and browse slices in the slice graph below ("C7"), move the cursor in the sweep graph ("C3"), change the Sweep Index value or change the value of the Slice at control. Tips: Move mouse over a graph and use keyboard Ctrl + cursor keys Up / Down / Left / Right to move the X-position of the sweep graph cursor. To modify the value of the Sweep Index control or the Slice at control, use its increment buttons or set the prompt in the control and use the cursor keys. The actual increment value of the Slice at control is shown in its tip strip. Note: Slice at control is converted to an indicator if the file was recorded in Multi line segment mode. (More details about Multi line segment mode in the Bias spectroscopy documentation)
C5	Channel	Select the channel to display in the sweep graph ("C3") and the slice graph ("C7").
C6	Slice processing	Processing of the slice data (like in Scan monitors).
C7	Slice graph	Displays the slice data of the channel selected in "C5" at the reference value set in "C4".
C8	Zoom and drop-down menu	Buttons for the graph: - Zoom in/out - Marquee Selection Zoom - Auto Fit Scales Drop down menu for more Scale and Graph options. Note: Generic graph handling is explained in the user interface section.
C9	Cursor and cursor indicator of sweep graph	Displays the sweep graph cursor coordinates. The cursor and the cursor indicator can be hidden/shown by right-clicking on the graph and selecting Cursor>Show. The color indicates on which curve the cursor is snapped to. Tip: For free cursor dragging uncheck the option Cursor>Stick to one curve in the right-click menu. It is possible to make the cursor jump to the other curve by selecting Cursor>Jump to other Curve.
C10	Color range	Adjusts the range of the selected channel over which the color ramp is distributed. The two digital controls to the left replicate the selection in the slider. Use the QuickScale buttons above the slider to modify the slider range. To select the color, use the color box to the right of the slider.
C11	Cursor of slice graph	Move the cursor using the mouse or the cursor controls above the graph. The cursor position defines which sweep curve is displayed in the sweep graph ("C3"). The cursor positions of the topography graph and the slice graph are synchronized. Tip: Move mouse over a graph and use keyboard Ctrl + cursor keys Up / Down / Left / Right to move the graph's cursor.

Back Home

Long Term Spectrum Viewer

The Long Term Spectrum Viewer displays the data saved with the Long Term Spectrum analyzer.
With the **Filter Pattern** capabilities it is possible to precisely define which files within the selected folder you want to access.

1	Browse settings	Click on the Browse button to select a file or a folder. Selecting a file will open it in the graph and set the **Filter pattern** to match its base name so that the File list contains all the image of that series. Selecting a folder will set the Filter pattern to "*" so that the File list contains all the files present in that folder. The Filter pattern follows the Windows matching rules: use the question mark character (?) to match any single character and the asterisk character (*) to match any sequence of one or more characters. Note: The selected path will appear in the window title bar. Tip: Both Filter Pattern and File allow dragging & dropping a folder/file into them.
2	**Spectrum file**	Select which file to display. The list contains the spectrum files in the selected folder that match the Filter pattern. Use the up/down arrows to quickly browse through the list.
3	Header Information	Display the information saved in the header of the file.
4	Signal Information	Display the **Channel** name, number of **Sample Points**, and frequency resolution **df(Hz)** of the displayed signal.
5	**Z Axis**	- **Base Color**: Define the color of the intensity graph. - Color ramp: Show the correspondence between the color and the PSD amplitude. - Color range: Define the range limits of the color ramp.
6	Export	- **Export all**: Export the displayed spectral data to an ASCII file. - **Export single**: Export the selected single spectra to a .dat file. The single spectra must be selected with the vertical cursor on the intensity graph, and its index is displayed next to the Export single button.

Back Home