Nanonics NWS

User Guide

NANONICS IMAGING LTD.

May 2012
Draft 8

Restrictions

RESTRICTION ON THE USE, DUPLICATION, OR DISCLOSURE OF PROPRIETARY INFORMATION. This document contains proprietary information which is the sole property of Nanonics Imaging Ltd. This document is submitted to the recipient for his or her use only.

Table of Contents

Introduction

NWS (Nano Workshop) is software developed by Nanonics for use with its range of MultiView and CryoView scanners. NWS has functions that are relevant before you carry out a scan; it enables you to monitor a scan while it is taking place and when the scan has finished you can view the scanned image. You can also analyze a wide variety of data that was collected during the scan.

The NWS functionality is described in this manual and it includes the following features:

- Control the stepper motor.
- Adjust the physical position of the sample on the scanner.
- Define the origin of triggers.
- Control the speed and resolution of the scan.
- Calibrate the scanner and carry out linearization.
- Define the size of the area to be scanned.
- Control the scan direction.
- Dynamically adjust the setpoint.
- View the image as it is “drawn” during the scan.
- View dynamic data about the scan.
- View trace and retrace graphs of a number of channels as the scan is taking place.
- Save the results of the scan in a number of formats.

Installation

Installing NWS consists of the following steps:

- Installing the data translation (DT) drivers.
- Installing the DT cards in the computer.
- Installing NWS.

Installing the Data Translation Drivers

>> ***To install the data translation drivers***

1. Insert the Data Acquisition Omni CD.
2. The Welcome screen appears:

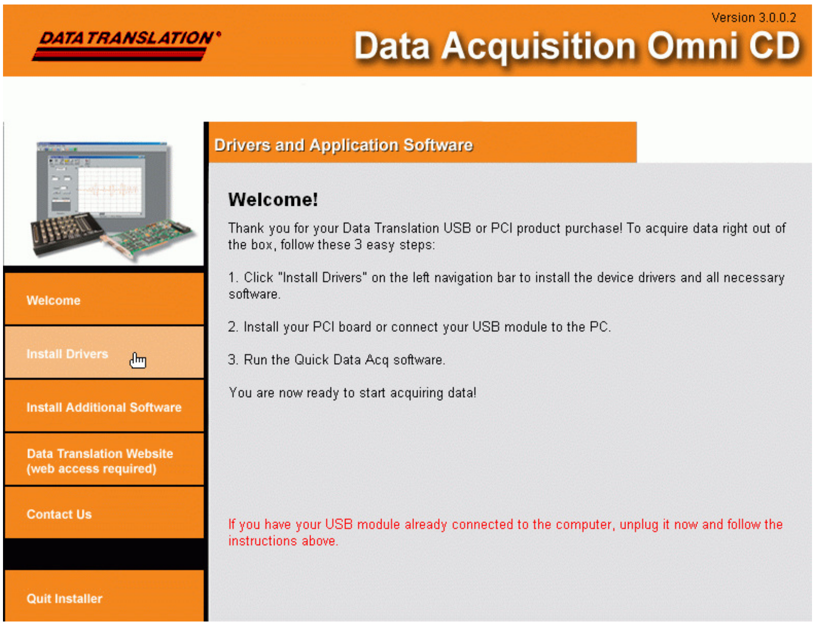


Figure 1: The Data Acquisition Welcome screen

- If the Welcome screen does not open automatically, double click setup.exe on the CD.

3. Click **Install Drivers**.
4. Click **Install Now**.
5. Click **Next** three times.

   The installation starts.
6. Click **Finish**.
7. Click **Quit Installer**.

Installing the Data Translation Cards

NOTE: If you purchased a computer from Nanonics as part of the system then the DT cards are pre-installed. Continue with "Installing the NWS Software" below.

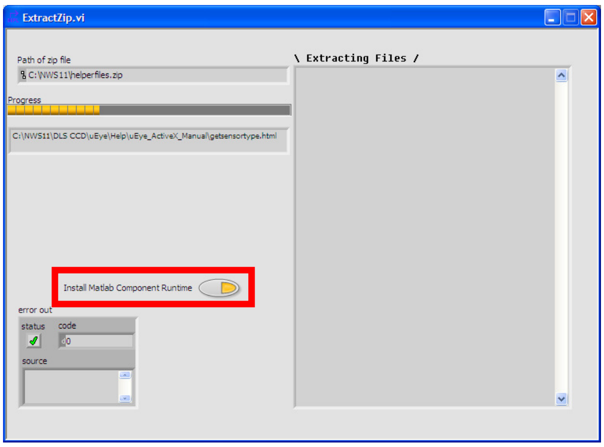
>> ***To install the DT cards***

1. Turn off the computer and disconnect the power.
2. Open the computer and install the DT 335 and DT 3016 cards in vacant slots. Make sure they are fitted securely.
3. Close the computer and restart it.
4. The computer identifies the cards and automatically installs them. Follow the on-screen prompts.

Installing the NWS Software

>> ***To install NWS***

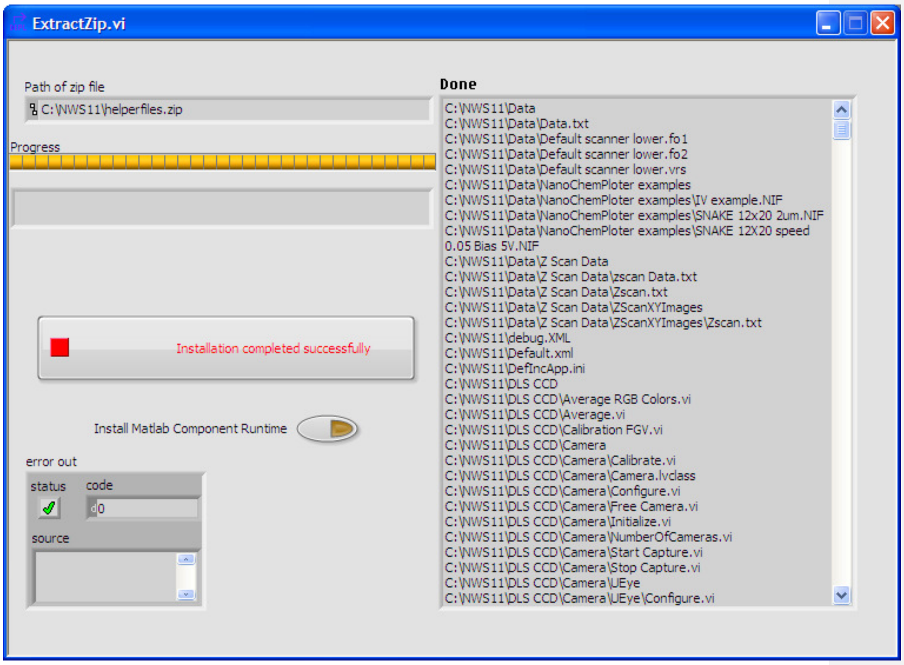
1. In the NWS folder with the installation files, double click setup.exe.
2. Click **Next** three times. The installation starts.
3. When installation is completed click next to install External Components.
4. Click on Install Matlab Component Runtime to install Matlab components. It is needed when the software is installed first time on the computer. If you have previous versions of NWS Software, the Matlab components are installed already on the computer.



NOTE: If Win7 OS is run on your computer, turn off the User Account Security level before Matlab installation

Comment [A1]: Should be checked

5. When installation of External components is completed click on "Installation completed successfully" button.



6. In order to complete the installation, reboot your computer.

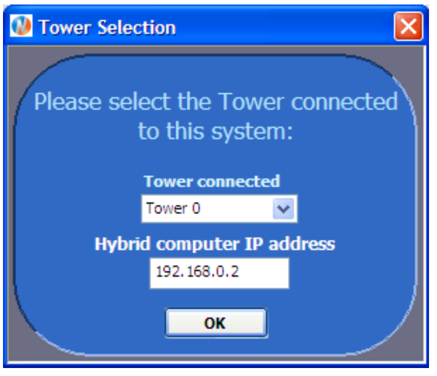
Opening the NWS Software first time

Click on **Start-> All Programs-> NWS11** program.

The Welcome screen appears:

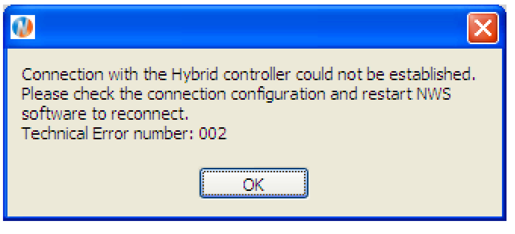
Choose "Hybrid System (Digital)" configuration and click START button.

The Tower Selection window appears.



The IP address of the Hybrid computer should be 192.168.0.2. Click Ok button.

If there is no communication between 2 computers the following message appears:



In this case check the connection between the computers (see Appendix 4)

The NWS Software

NWS has two main views:

- ***Scopes:*** Displays 2D images of the sample during a scan, as well as graphs based on the scanning data. The Scopes view is described below.
- ***Actions & Settings:*** Enables you to enter settings for the scanner. The Actions & Settings view is described on page 22.

Each view has an upper panel and a lower panel.

\>> ***To choose a view***

- Click the **Actions & Settings** or **Scopes** tab in the top left corner.

The Scopes View

The Scopes view is shown below:

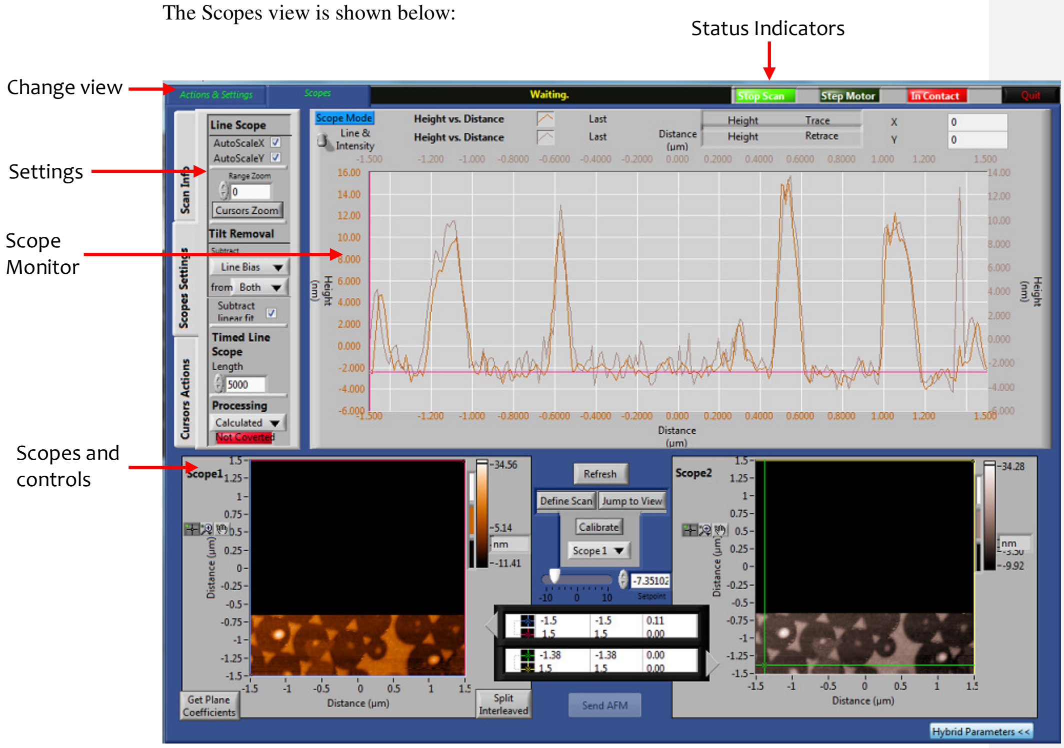


Figure 2: The Scopes view

The Status Indicators

Figure 3: The status indicators

The Status Indicators appear at the top of the screen and they give information about the status of the scanner.

Indicator	Description
Waiting, Scanning, etc.	The current status of the system. When no scan is taking place “Waiting” is displayed.
Scan	Click **Scan** to start; click **Stop Scan** to end a scan. When you click **Scan** to restart, the scan starts from the beginning and not from where it stopped. See also the Start button in the Scan Window on page 28.
Step Motor	When bright green, the stepper motor is in motion. Press the indicator to stop the movement caused by the stepper motor. The usual method for stopping the stepper motor is by pressing the Approach or Retract button for a second time in the SPM Control tab. See page 33.
In Contact	When red, the tip is in contact with the sample. Press the indicator to perform retract.
Quit	Click **Quit** exiting NWS Software.

Scope Monitor

The Scope Monitor is displayed below. It appears in the upper panel.

Figure 4: The scope monitor

The scope monitor displays the traces from two channels simultaneously. The names of the channels being displayed are shown above the monitor. The channel monitored by the blue trace appears on the top line and the channel monitored by the red trace is underneath. The scale of the blue trace is to the left and the scale of the red trace is to the right.

The monitor display can easily be changed by clicking on any of the labels and making a choice from the menu that opens.

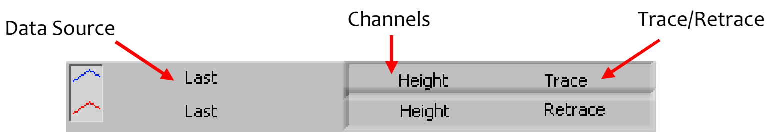


Figure 5: Changing the scope monitor display

Data Source

Clicking the data source ("Last" in the example above) opens a menu with the options below. Each graph can be set separately.

Option	Description
Last	The last line scan obtained during the scan.
Blue Cursor	The profile of the scan image measured along the blue cursor in the scan window.
Red Cursor	The profile of the scan image measured along the red cursor in the scan window.
Cursor A to Cursor B	The cross-section between two points defined by the cursors in the scan window
Timed Line Scope	Displays the points over the timed interval that appears in the **Timed Line Scope** field in the **Scope Settings** tab. See page 14. The number is the number of points to display. If the number in the timed line scope is -1, it shows all the scan points made using this option and it ignores the trace/retrace setting.
Ch1 Vs Ch2	Displays a plot of one channel against another.

Channels

Clicking the channels (both are "Height" in the example above) opens a menu with a list of the channels that are currently acquired. Each graph can be set separately. The list of channels is also displayed in the Input Channels settings (p. 26).

Trace

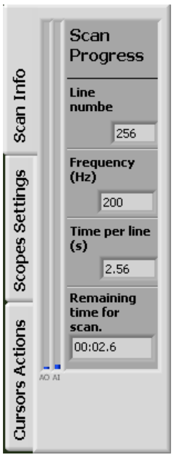
Click and choose whether to display the Trace or Retrace line.

Scope Settings

There are three tabs in the Scopes view that enable you to view or set parameters that relate to the way the data in the Scope Monitor is displayed.

Scan Info

The Scan Info tab provides information about the line currently displayed in the Scopes. These parameters are not relevant when external triggers (see page 27) are being used during the scan.



Field	Description
Line number	The number of the line which is currently being drawn.
Frequency	The frequency with which the scan is recording points.
Time per line	The time taken to scan each line.
Remaining time for scan (minutes)	The estimated time remaining for the scan.

Figure 6: The Scan Info tab

Scope Settings

The Scopes Settings tab allows enables you to set a number of parameters for the scan.

Figure 7: The Scope Settings tab

Field	Description
Auto Scale X Auto Scale Y	When these are checked the scale in the graph updates automatically according to the graph's values
Range Zoom	See next page for explanation.
Tilt Removal (for the Scope windows)	These settings apply different filters to the line scans during the scan. This is seen in the *lower* scope displays. The drop-down lists enable you to subtract values in one or both of the scopes. Line Bias, Line Linear Fit, Line Parabola Fit: These correct biases in each line of the scan individually. Plane: This takes into account the plane of the entire sample scan. **Tip:** When the scan in the scope appears flat and featureless, try subtracting a tilt removal parameter in order to see the sample features.
Subtract Linear Fit (for the Scope Monitor)	When the lines in the graphs in the Scope Monitor are not generally horizontal, check **Subtract Linear Fit** to make the display more horizontal.
Timed Line Scope	Set the number of points displayed in each timed line scan. When the number of points exceeds the selected number then the older points are discarded and the new points are shown. To show all points enter -1.
Processing	When **Calculated** is selected from the drop-down list, the data for each channel is processed according to the formula listed in the Input Channel option. See page 26. If you select "Processing: Calculated" when the data is not converted, it will show calibrated data on the line scopes and in the intensity map for the Height channel only. After the data is converted, selecting "Processing: Calculated" will show all calibrated data on all channels in the scope windows. Selecting "Processing: Calculated" also enables you to save to the WSxM calibrated format – see page 37.

Field	Description
Converted/ Not Converted	Click to switch between Converted and Not Converted. When **Converted** is selected, the values are converted from their voltage values to their calibrated values according to the input channel formulae so they can be shown in the scope windows. This means that the y units in the Scope Monitor change from Volts to the actual units of the channel that is displayed in the y axis. This option is used for saving calculated data in ASCII format. **NOTE: Don't use this option during the scan!**

Range Zoom

The range zoom is useful when the y values of the two graphs are far apart. By entering a value in the Range Zoom field you can alter the values of the y axis and display both graphs and compare them.

Range Zoom Value	Range of Y Axis
a < 0	(–a, a)
a > 0	(0, a)
a = 0	Range zoom is inactive

>> ***To cancel the range zoom***

- Check and uncheck **Auto Scale X** and **Auto Scale Y**.

Cursor Zoom: Zoom in to the area enclosed by the graph's cursor lines.

>> ***To zoom into an area on the graph***

1. Use the cursor lines to define a rectangular area.
2. Click **Cursor Zoom**.

   The graph zooms into that area.

- To clear the zoom, check **Auto Scale X** and **Auto Scale Y**.

Cursor Actions

The Cursor Actions options are explained in the Nanonindentation User Guide.

The Scan Windows

The scan windows display 2D images of the sample based on the channels selected in the Scope Monitor. They appear in the lower panel.

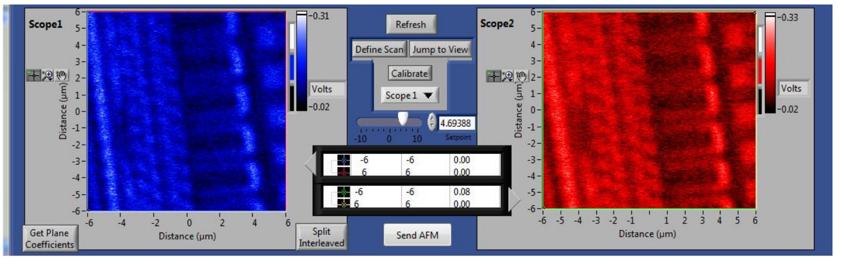


Figure 8: The scan windows

In order to set the scan parameters select the desired scope using the scope selector in the center of the panel Scope 1. Scope 1 is on the left and Scope 2 is on the right. Set the scan margins in one of two ways: (1) Drag the cursor lines to the desired location or (2) Manually enter the parameters in the scan parameter box. The box for Scope 1 is in the upper part of the number grid, with the left arrow and the box for Scope 2 is on the lower part: .

When you have defined the scan area, click **Define Scan**.

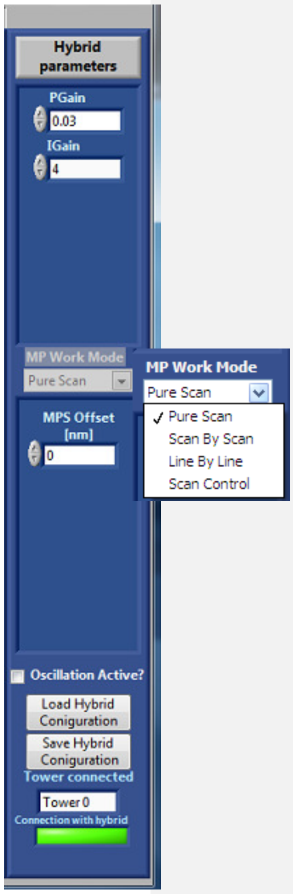
The scan in each scope is in the colors that appear in the color bar at the top right of each scope. Click the different areas of the color bar to change the colors. Changing the color of the scan also changes the color of the graph in the Scope Monitor.

The value of the setpoint is displayed between the two scopes. You can adjust it using the slider or by entering a value in the numerical field and pressing the Tab key. You can also set the setpoint in the SPM Control tab – see page 33.

Click **Refresh** to refresh the image in the scope windows.

To the right of each scan window, next to the color bar, is the unit box. During a scan this is always Volts, as the data is unprocessed. An exception to this is when “Height” has been selected in the Scope Monitor. In this case, when “Calculated” is selected in the Processing area of the Scopes Setting tab, the units are nm.

If “Calculated” is selected in the Processing area of the Scopes Settings tab (see p. 14), and “Converted” is displayed beneath, the data will be processed upon completion of the scan and the Scope window will show the data with the correct units. It is recommended to select “Converted” only when the scan has finished.



Hybrid Parameters

The Hybrid Parameters are appeared at the right. It gives possibility to change PGain and Igain parameters during scan.

MP Work Mode.

1. **Pure Scan**. Regular scanning mode
2. **Scan by scan**. Multipass scanning mode by doing first scan when tip is in contact and second scan when tip is out of contact
3. **Line by line**. Multipass scanning mode by doing one line in contact and the same line when tip is out of contact.
4. **Scan Control**. Not implemented yet
5. **MPS Offset** [nm]. The height of the second pass
6. **Oscillation Active?** Activation of Oscillation on the second pass

7. **Load Hybrid Configuration/Save Hybrid configuration buttons.** Possibility to load previously saved Hybrid parameters or to save it
8. **Connection with Hybrid.** If the indicator is green the computers are connected. Otherwise there is no connection between computers

Linearization & Calibration

The scanner is always supplied after it has been linearized and calibrated, and it is not usually necessary to carry out these actions again. However, if at some point you wish to carry out linearization and calibration, follow the instructions in this section.

Linearization: This is recommended for the x-y axes, as it includes calibration of the x-y axes as well.

Calibration: This is recommended for the z axis only.

Linearization

Linearization is carried out by scanning the calibration grid, and then carrying out a number of adjustments. If the calibration grid displays the rectangle artifact after it has been scanned then linearization is required. The rectangle artifact has occurred when a series of rectangles start off with square dimensions, and as the scan progresses the squares are "squashed" into rectangles, as shown below.

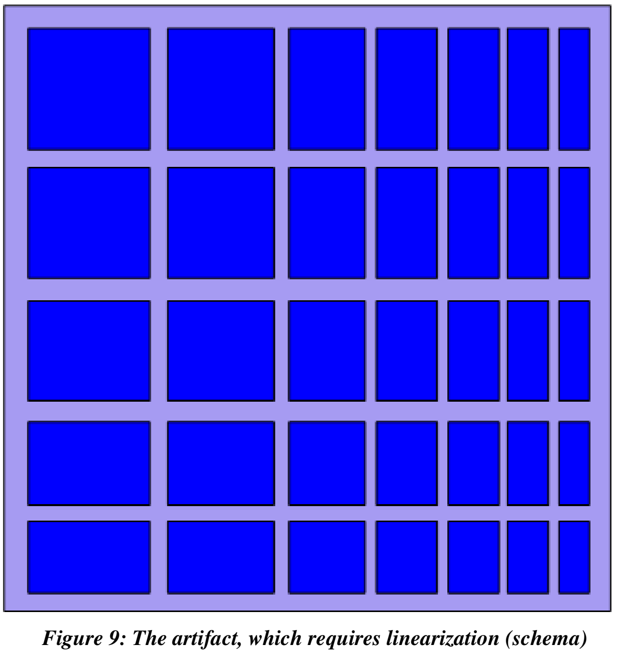


Figure 9: The artifact, which requires linearization (schema)

This is a sign that the scanner is speeding up as the scan progresses. The same effect is shown below as it appears in the Scopes window:

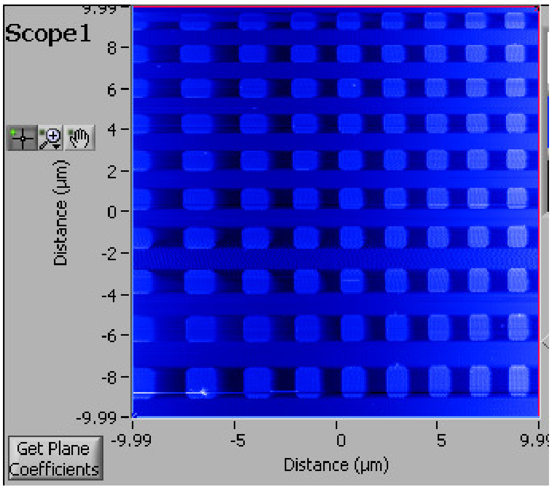


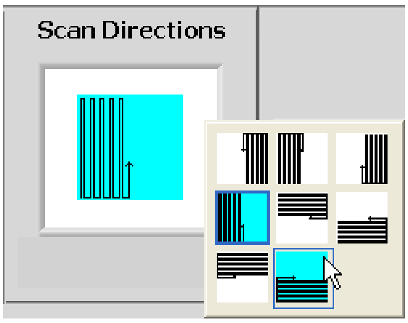
Figure 10: Scan of a sample showing that linearization is required

In actuality all the squares in the sample are identical but in the scan it can be seen that they become smaller. To correct this artifact, carry out linearization.

IMPORTANT: Linearization and calibration are relevant for a specific scanner only. If your system has more than one scanner, e.g., MultiView 2000 or MultiView4000, or if you are using the NWS software to control two or more different systems, then note that each scanner has to be linearized and calibrated separately.

>> ***To scan the calibration grid***

1. Choose the **Scan Window** tab.
2. Choose "Direct" from the **Formula** drop-down list.
3. Enter a **Window Size** of 20 x 20 µm.
4. In the **Scan Directions** area, choose either "top-left to bottom-right" or "bottom-right to top-left", as shown on the right. Note which direction you choose, as you will repeat this process using the other scan direction.



5. Click **Scan**. The Scan window will appear.

>> ***To carry out linearization***

1. When the scan has finished, use the cursor lines in Scope 1 to enclose all the complete squares and rectangles as shown below:

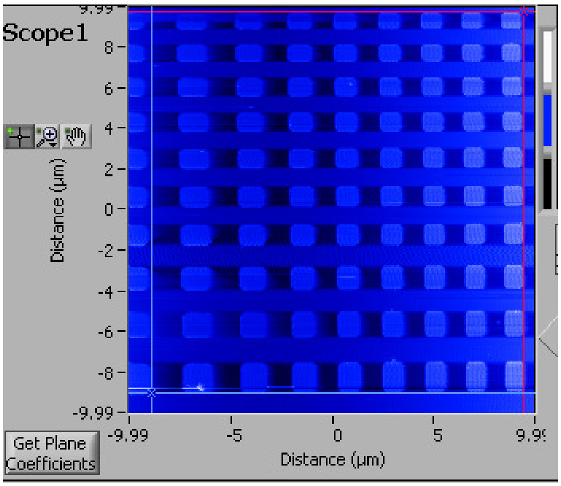


Figure 11: Enclosing the complete squares

2. Click **Calibrate**.

3. Click the **Linearization** tab.

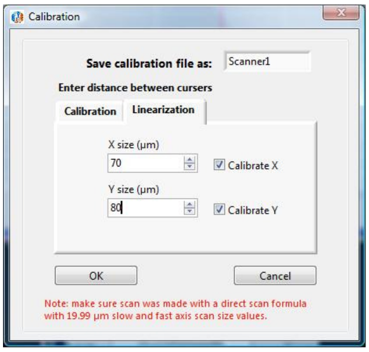
4. Enter a name for the linearization in the **Save calibration as** field, e.g., "Scanner1". Do **not** choose a name that is reserved for other uses, such as "Default", "Direct", "NanoWrite", "Sensors", "Softlin", etc.

NOTE: When you have more than one scanner, e.g. MultiView 2000/4000, then each linearization needs to be saved separately. Therefore it is recommended to include the terms "upper scanner" or "lower scanner" in the file name in order to differentiate.

5. Estimate the length between the cursors by calculating the number of the rectangles between them. The periodicity of calibration grid is usually 10 microns.

6. Enter the values, in microns, in the **X size** and **Y size** fields. The x-axis is the way it appears in the Scope window.

7. Click **OK**.



8. The linearization file is saved and the Linearization Simulation window opens, as explained below.

Checking the Linearization & Calibration

After clicking OK in the Linearization tab, the Linearization Simulation window opens automatically. This enables you to check the linearization.

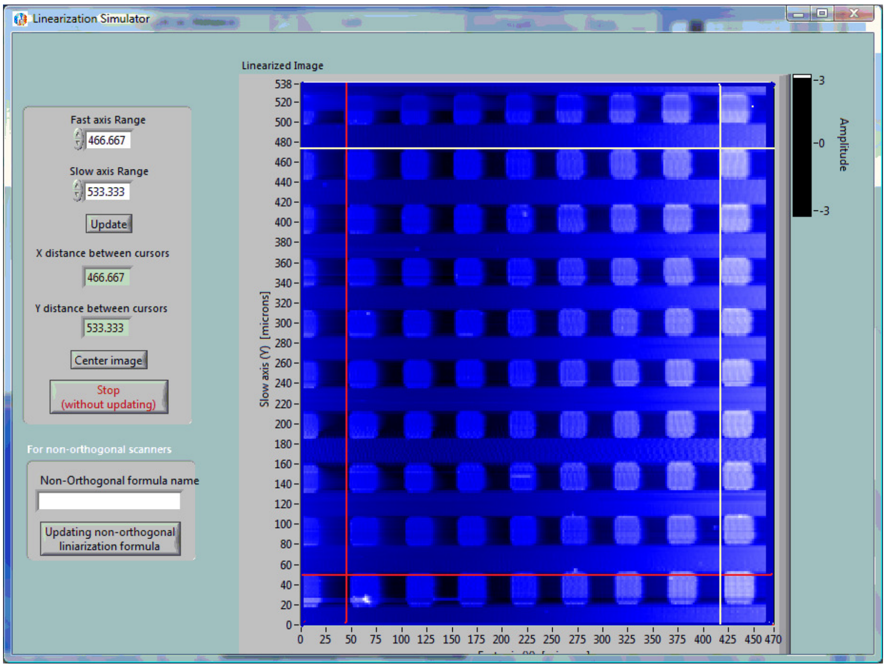


Figure 12: The Linearization Simulation window

The scanned image has had the linearization settings applied to it, so the rectangle artifact should not be discernible. The window has the following fields and buttons. All distances are in microns.

Field/Button	Description
Fast Axis range	The range of the x axis.
Slow Axis range	The range of the y axis.
Update	After checking the linearization click **Update** if you made adjustments.
X distance between cursors	The distance between two vertical cursors
Y distance between cursors	The distance between two vhorizontal cursors
Stop without update	Exit this window without making any changes.
Non-Orthogonal formula name	Use this option when the scanner looks non – orthogonal after linearization process (the squares look like rhombs)

>> ***To check the linearization***

1. Place the vertical cursors and horizontal ones to enclose all the complete squares, as shown below:

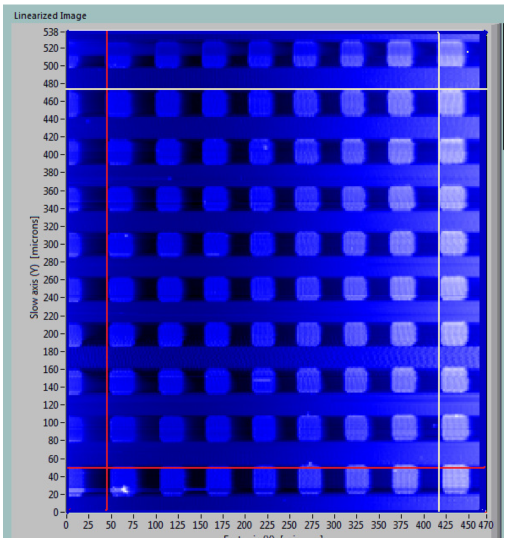


Figure 13: Checking the linearization

2. The distance between the two vertical lines appears in the "X distance between cursors" field. Compare this distance with the actual known distance.

3. If the difference is small (less than half a micron), this could be due to the fact that the cursors are not placed exactly. If the difference is not negligible, adjust the value in the **Fast Axis range** field.

4. The distance between the two horizontal lines appears in the "Y distance between cursors" field. Compare this distance with the actual known distance.

5. If the difference is small (less than half a micron), this could be due to the fact that the cursors are not placed exactly. If the difference is not negligible, adjust the value in the **Slow Axis range** field.

6. If you made changes, click **Update** and then click **Replace** in the confirmation window that opens.

7. If no adjustments were necessary, click **Stop without update** and continue with the next procedure below.

>> ***To linearize for other scan directions***

In Step 4 of "To scan the calibration grid" on page 15 above, you chose a scan direction. Now, repeat the entire linearization process for the other scan direction.

The linearization files for different scan directions have different file names.

Testing the Linearization

When you have completed the linearization process, scan the calibration grid for a final time in order to check that the scan is correct and that the rectangle artifact has been corrected. In the **Formula** drop-down list in the **Scan** tab, choose one of the linearization files that you created.

Z Calibration

Linearization includes calibration of the x and y axes so in order to calibrate those axes it is recommended to carry out linearization. Calibration is recommended for the z axis.

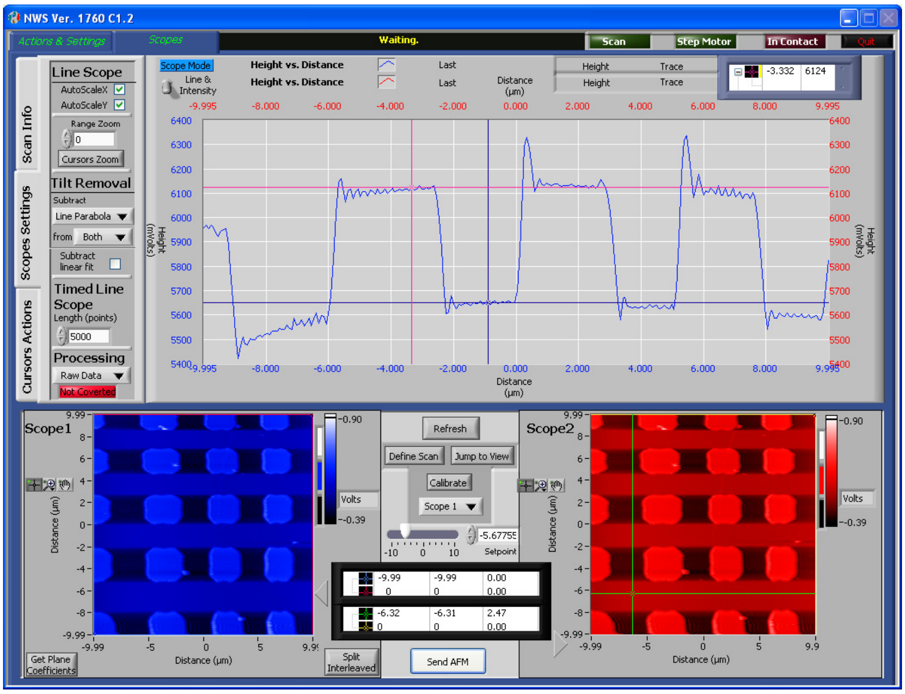
>> ***To calibrate the scanner in Z direction***

1. Scan the calibration grid using the following settings: **Formula:** Direct; **Scan Direction:** “top-left to bottom-right” or “bottom-right to top-left”.

2. When you have two–three rows of squares you can stop the scan.

3. In the Scope Monitor, change one of the data sources to “Blue Cursor”.

4. In the Scan Window, move the blue cursor to an area of known



topography.

5. In the Scope Monitor use the cursors to choose a minimum point and maximum point on the graph. When using the standard calibration grid this distance is 110-119nm (depends on what is written on the Calibration Grid box)

6. Check the **Processing** button is in **Raw Data** position

7. Click **Calibrate**. The following window opens:

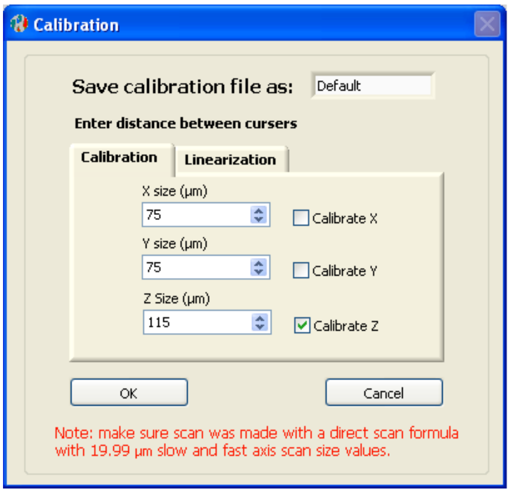


Figure 14: The calibration window

8. Enter 115 in the **Z Size** field.

9. Check **Calibrate Z** only.

10. In the **Save calibration as** field, enter the file name that you chose for the linearization.

NOTE: Do not choose "Direct", "Sensors" or "Softlin" as these are system names.

---

11. Click **OK**.

You can check the calibration by repeating the scan and checking the distance between the minimum point and maximum point as described above.

The Actions & Settings View

The Actions & Settings view is shown below:

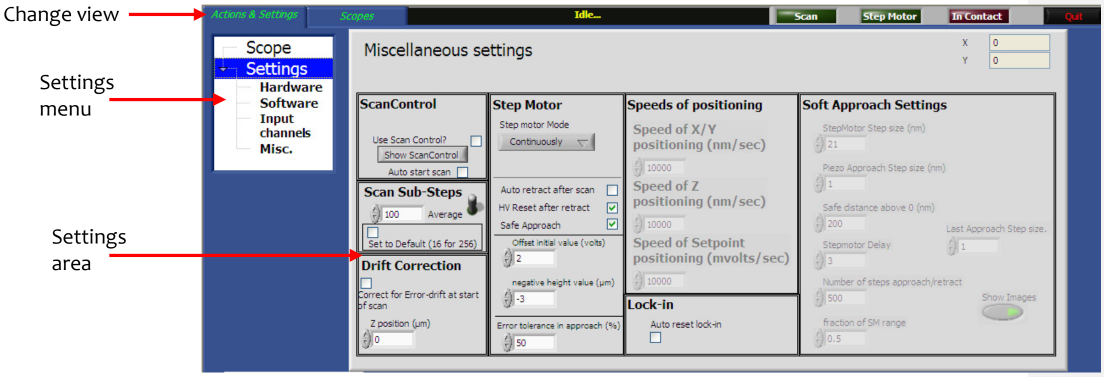


Figure 15: The Actions & Settings screen

NOTE: The upper panel displayed above shows the Settings menu and area. When you click **Scope** in the Settings menu you can display the Scope monitor (see page 11).

There are four options in the Settings menu:

- Hardware (see Appendix 1)
- Software
- Input channels
- Miscellaneous settings.

Software Settings

The Software Settings screen is shown below:

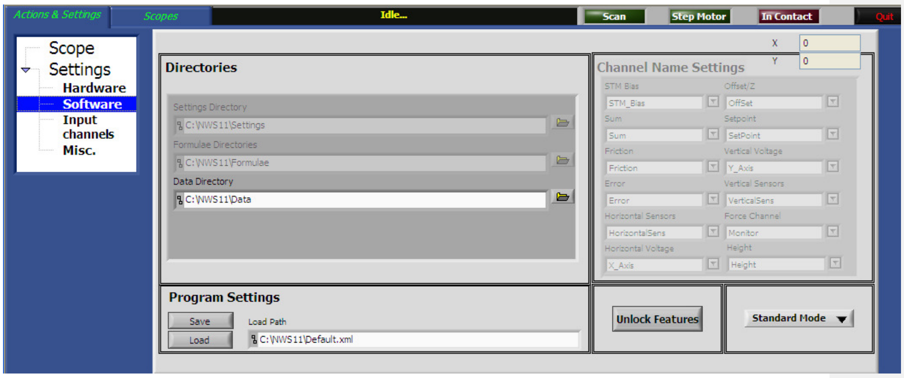


Figure 16: The Software settings

The Software settings are described in the table below.

NOTE: Use this feature only if two or more users share the computer and they wish to save their data in different folders.

Field	Description
Directories	The location of a number of NWS folders is displayed. To change the Data folder, click the yellow folder icon, choose a different folder and click **Open**. **Note**: Change the Data folder only if two or more users share the computer and each wants to save their data in a separate folder. Each time the users work with NWS they will have to change the folder.
Program Settings	Save the current scan settings (not images), or load settings saved in the past. • Click **Save** to save the current settings to a different file. • Click **Load** to load the settings file that appears in the **Load Path**. • Click the ⇅ graphic to designate whether or not the load path is valid. • Edit the **Load Path** to specify a different path or file. Saving the program settings enables you to easily work in different modes for different tasks, without changing all the program parameters manually.
Unlock Features	For the use of Nanonics technicians only. It enables the extended features of NWS, such as NanoPlot, Sensor controller type, etc. to be activated.

Field	Description
User Mode	User Mode is **Standard Mode.** The **Service Mode** is for use of Nanonics staff only.

Input Channel Settings

The Input Channel screen is shown below:

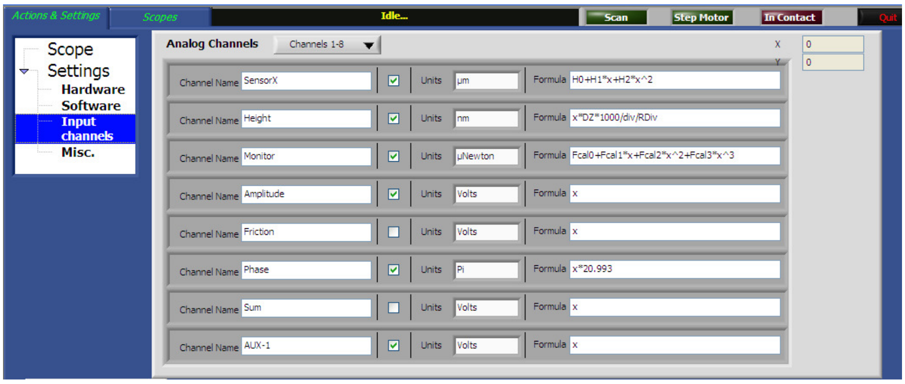


Figure 17: The input channels

The Input Channels show the channel names, the units and the formula for deriving the channel's value. The check mark indicates that the channel is active and will be displayed. There are 16 possible channels and an additional eight "virtual" channels. Use the drop-down list at the top of the screen to view the additional channels. The channel names can be chosen in the Scopes screen.

Miscellaneous Settings

The Miscellaneous Settings screen is shown below:

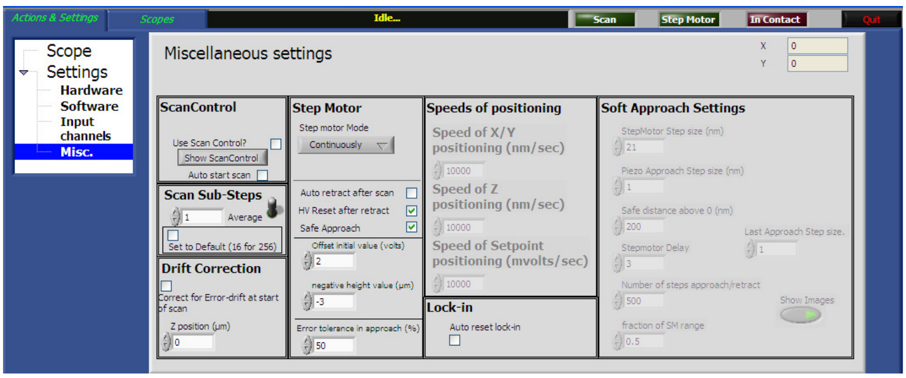


Figure 18: Miscellaneous settings

The Miscellaneous settings control various aspects of the scan and are described in the table below:

Field	**Description**
Use Scan Control?	Check this box if the scan controlling external device is needed (for example, AFM-Raman scans)
Auto start scan	Check this box to start Scan Control with the previous parameters.
Scan Sub-steps	The number of sub-steps to move between one point and the next. The recommended value is 100. The higher the number of sub-steps, the slower the maximum speed of the scan (if the large number of channel is acquired simultaneously)
Step Motor	The stepper motor has three modes: **Continuously:** When pressing **Retract** or **Approach** in the **SPM Control** tab, the motor operates continuously until you press the button again to stop. **By time:** The motor operates for the number of seconds defined here. **By steps:** The motor operates for the number of steps defined here. When working in this mode the value of the step motor speed ("Speed" field, p. 33) is ignored.
Auto retract after scan	When a scan has finished, the stepper motor is automatically retracted. This is useful for an unsupervised scan, such as a scan that will take a long period of time or a scan that runs overnight.
HV reset after retract	When tip is retracted from the surface, the scanner returns to its zero position.
Auto-reset Lock-in	When hardware is switched OFF and ON and Software is in working mode, check this box for right Lock-in work
Error tolerance in approach (%)	This value defines permitted deviation of Error signal during approach. The default value is 50%. For MV1000 head this value should be increased to 500% to allow smooth approaching.
The next options are for use of Nanonics staff only	
Safe Approach	**Offset initial value:** The recommended offset before approach. This should always be positive. **Negative height value:** A value used in the approach algorithm.

Field	Description
Speed of XY Positioning	Maximum positioning speed of the X and Y piezo. At the beginning of a scan the tip is in a central position. The value in this field determines how fast the tip moves to the starting point for a scan. When the scan finishes, the tip returns to a central position.
Speed of Z Positioning	The maximum Z speed (movement of the tip up and down). This value is also connected to the Offset that appears in the SPM Control tab. It is not connected to the initial position for a scan.
Speed of Setpoint Positioning	The speed at which the Setpoint is changed.

The Scan Settings

The Scan Settings appear in the lower panel. There are four tabs:

- Scan window
- Positioning & Inertial Motion
- SPM Control
- Files.

Scan Window

Use the Scan Window tab to set parameters for the scan.

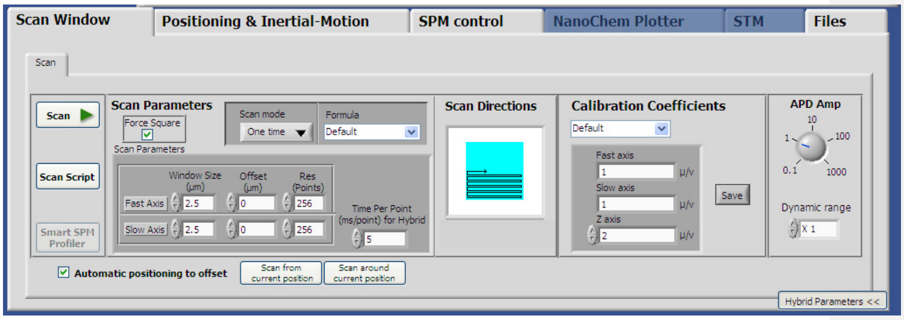


Figure 19: The Scan Window tab

The fields and buttons are described below:

Field	Description
Start	Start the scan.
Scan Script	Start the script for scanning. Provides the possibility to run several scans with different parameters, one after another. See Appendix 3 for explanation

Field	Description
Z Axis Scanning (optional)	Start scanning in Z direction

Scan Parameters

The scan parameters are described below:

Field	Description
Window size	The scan area in microns. This area is represented by the scope windows. See page 30 for an explanation of the fast axis and the slow axis.
Force square	Force the window size to be square. When you alter a value for the fast axis or slow axis, the other axis automatically has the same value.
Offset	You can offset the default center of the scan by adjusting the fast or slow axis.
Res	The number of points per line that are scanned. This determines both the resolution and the speed of the scan.
Time per point	The rate at which each point is sampled.
Scan mode	Set whether the scan is performed once or continually. If the continuous option is chosen the image is not saved at the end of each scan. In Continuous mode the scan is performed with the scan point first going normally, then with the point of the original scan going backward and then forward again and so on until you change the scan mode to "One Time".
Automatic Positioning to Offset	When this checkbox is checked the Offset values are automatically applied to HV PiezoDriver before scanning.
Start from current position	Starting scan from the point od current tip position without moving the tip to the initial position.
Scan around current position	When **Positioning** tab is used for tip positioning, this button automatically apply the current position of the tip to Offset field
Formula	See below.

The Scan Formula

The data that is outputted during the scan is calculated using the scan formula. User defined formulae can be set by adjusting the calibration and/or linearization parameters – see page 17 and following. The following formulae are pre-defined.

Formula	**Explanation**
Default	The scanned image takes into account the default calibration settings.
Sensors	The scan uses hardware linearization and is processed according to the input of the hardware sensors. See Appendix 1
Direct	Choose this option when calibrating the scanner, as unprocessed signals are produced. When using this mode the window size is in Volts (max. 19.99 V).
NanoWrite	The scan passes each line just once, without a retrace scan. This option is used mostly for lithography.

Scan Directions

Click on the image to define the scan direction and the fast/slow axis. The scan direction can be top-to-bottom, left-to-right, etc.

The *fast axis* is the axis along which the line scans are taken and the slow axis is the other axis.

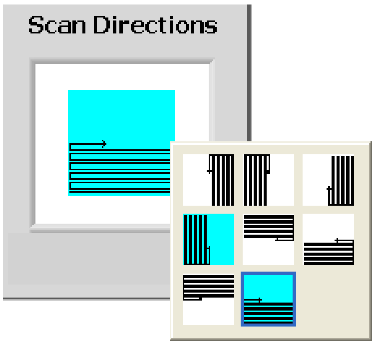


Figure 20: The scan direction

The image that is shown is the current choice. Click the image to open the menu. The turquoise images are the common choices.

The origin shows the starting location of the scan and the arrow shows in which direction the scan is going. For example, the option that is selected above is bottom left-to-top right. The fast axis is horizontal and the slow axis is vertical.

Calibration Coefficients

The frame contains fields with parameters of calibration in X, Y and Z directions. The units are micrometers per Volt. Each scanner has its own file for calibration.

APD Amp

When NSOM channel is acquired, the settings in APD Amp field should be chosen correspondently with the position of knobs in **APD Counter & Power Supply box,** in order to have right calibration of NSOM channel.

Positioning and Inertial Motion

Use the Positioning and Inertial Motion tab to control the rough positioning of the sample.

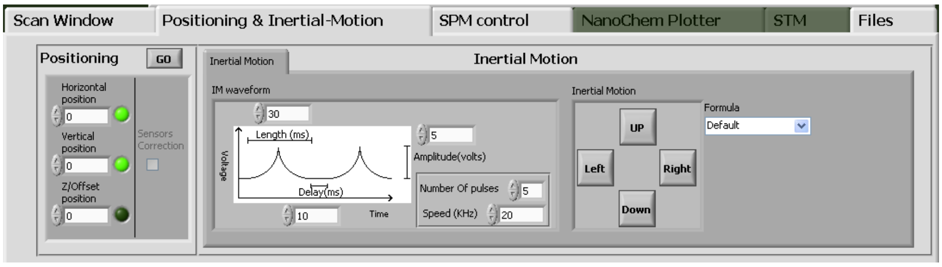


Figure 21: The Positioning & Internal Motion tab

Positioning

Positioning is used to set the position of the scanner (x, y and z) within the scan range. The units are microns.

Define the desired position in the **Positioning** area and click **Go**. Using hardware sensors the position can be kept and monitored to compensate for drift effects and so on.

Inertial Motion

Inertial Motion is used for the rough positioning of the sample over distances that are larger than the scan range, i.e., 2 mm – 4 mm. The sample holder rests on three sapphire balls and short pulses from the piezos “nudge” the sample, similar to the way in which one might tap an object to move it into a precise position.

The shape of the pulse applied to the piezos is shown in the diagram and you can set the parameters of the pulse in this tab.

Define the number of pulses and their speed in the **IM Waveform** area and click the Inertial Motion direction buttons to the right.

SPM Control

The SPM control tab enables you to control the scan itself.

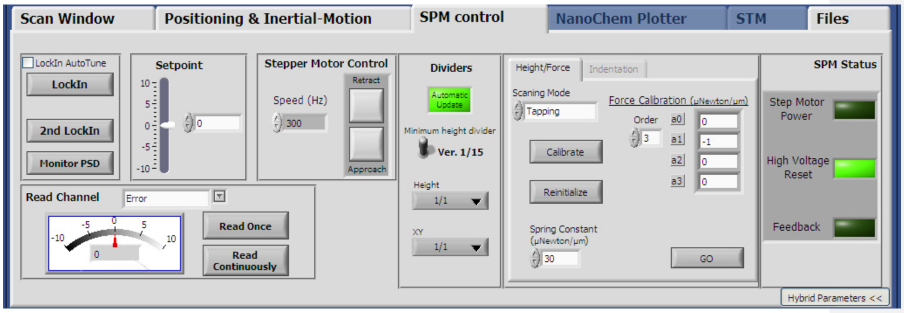
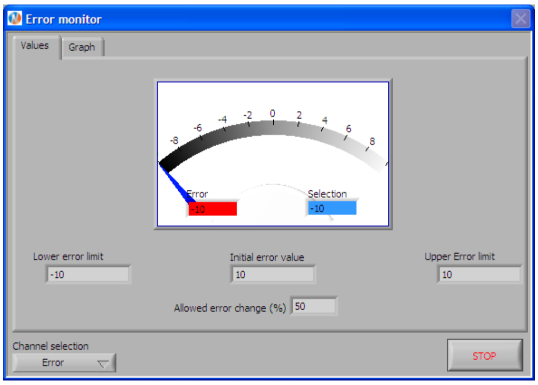


Figure 22: The SPM Control tab

The SPM Control tab includes a number of areas and they are explained below.

Area	**Description**
Stepper Motor Control	Control the speed of the stepper motor and click the **Approach** and **Retract** buttons to move the tip towards and away from the sample. When on of the buttons are clicked the **Error Monitor** window appears and show current Error value. If the Error value is changed more than **allowed Error change (%)** one during **Approach** the approaching is stopped and Error message appeared. In this case Error value should be checked again. 
Speed	The speed of the stepper motor in Hz.
LockIn	Open the LockIn module. See page 39 below.
2nd LockIn	Used for Kelvin probe measurements
PSD Monitor	Open the PSD Monitor. See page 35 below.

Area	Description
Setpoint	Set the setpoint using the slider or the numerical field. Press the Tab key if you type the value. The setpoint can also be set in the area between the scope windows – see page 16.
Height/Force Diagram	Click **Go** to produce a trace and retrace of the force upon the probe as it moves relative to the sample. This calibrates the force from the deflection of the laser beam on the PSD's quadrants. See the Nanonindentation User Guide for more details on this feature.
SPM Status	The indicators show the status of the stepper motor, the High Voltage supply and Feedback. Click the green buttons to change the status. Bright green: active; dark green: inactive.
Read Channel	Select a channel from the drop-down list and click **Read** to view the current value.
Dividers	The value that is displayed matches the setting on the High Voltage divider. When **Automatic Update** is selected, changing the value on the High Voltage driver changes the value here.
Minimum height divider switch	Use **Ver.1/23** position when the minimum height divider of HV PiezoDriver is 1/23. Use **Ver.1/15** position when the minimum height divider of HV PiezoDriver is 1/15.

PSD Monitor

The PSD Monitor is used to align the laser for beam bounce feedback and to define the setpoint used during the scan.

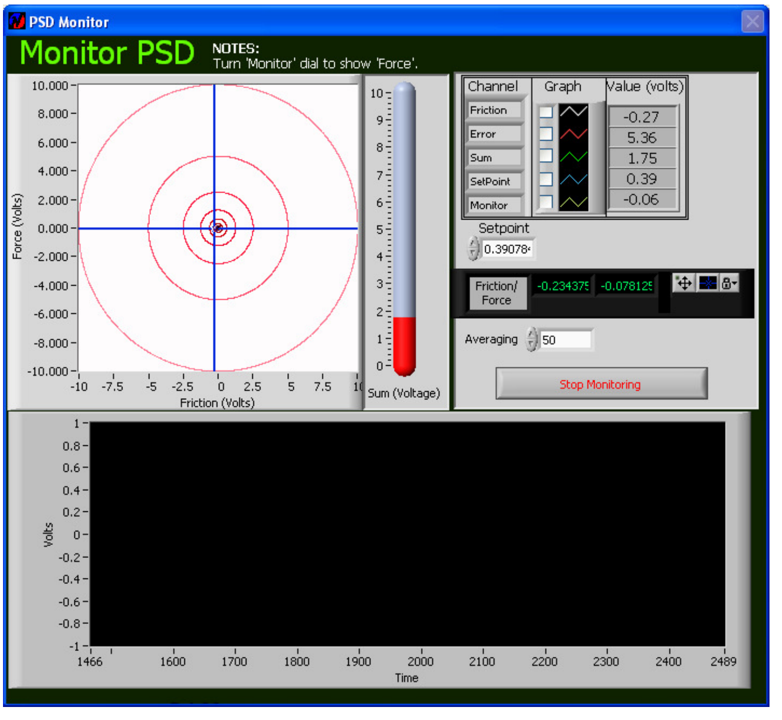


Figure 23: The PSD Monitor

>> ***To open the PSD Monitor***

- In the **SPM Control** tab, click **PSD Monitor**. The window above opens.

The Target shows the position of the center of the deflected beam which is reflected onto the PSD. The cursor should be centered when the Error and the Friction signals are both set to zero.

The Sum monitor to the right of the target shows the total signal received by all four quadrants of the PSD. For good alignment the Sum should be at a maximum.

The values of five channels are displayed to the right of the Sum monitor. They can be displayed on the graph by checking the relevant check boxes. The values are averages based on the last 50 points. In the **Averaging** field you can change the number of measurement points on which the averaging is based.

The setpoint is also displayed and can be set here as well.

The Friction/Force values are the coordinates of the crosshairs on the monitor.

>> ***To close the PSD Monitor***

- Click **Stop Monitoring**.

The File Tab

The File tab enables you to save data about the scan.

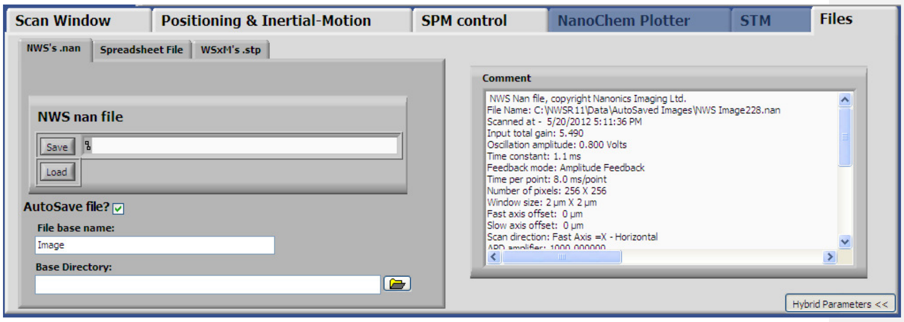


Figure 24: The File tab

The files can be saved in three formats:

1. Nanonics NWS: file extension .nan
2. Spreadsheet file: file extension .txt
3. WSxM format: file extension .stp

>> ***To save a nan file***

1. In the **NWS** tab the scan parameters information is saved in the **Comment** area.
2. The files are automatically saved in …\Data\Autosaved Images folder.
   a. In order to change name type the desired name in "File Base name"
   b. In order to change Base directory, click on the **Browse** icon near Base directory field and choose the desired directory.
3. In order to save file manually:
   a. Click **Save**
   b. In the window that opens, choose a location and enter the name for the file
   c. Click **OK**

   The path and file name appear next to the Save button.

NOTE: If you wish to save again, without replacing the file that appears next to the Save button, click ⇅ and choose "Not a path".

>> ***To load a nan file into the scope windows***

1. Click **Load**.
2. Locate the file and click **OK**.

   The data will now appear in the scopes.

>> ***To save a spreadsheet file***

The spreadsheet file is a line graph text file in ASCII format (tab delimited) which can be opened in a spreadsheet.

1. Click the **Spreadsheet file** tab.
2. Enter any notes about the scan in the **Comment** area.
3. In the drop-down list select “Raw Data” to save the data in Volts; select “Calculated Values” to save the data in the channel’s units.
4. Click **Save**.
5. In the window that opens, choose a location and enter the name for the file.
6. Click **OK**.

>> ***To save a WSxM file***

1. Click the **WSxM** tab.

   The following window opens:

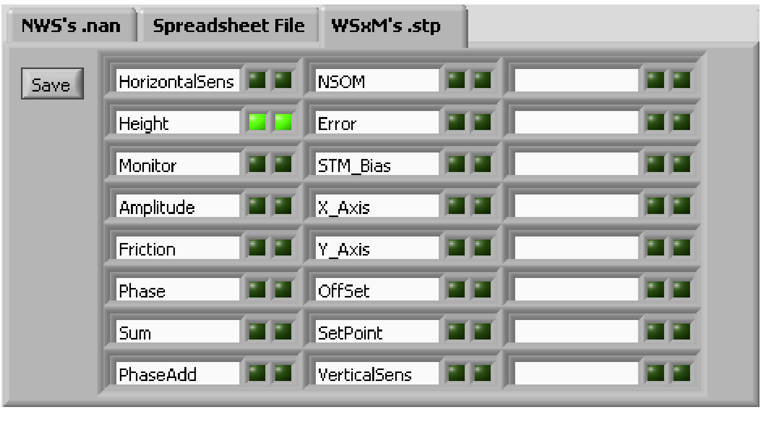


Figure 25: Saving a WSxM file

The name of a channel appears. Additional channels may be displayed.

To the right of each channel are two green lights. When the left hand light is selected (bright green), the trace data is saved; when the right hand light is selected, the retrace data is saved

When Multipass option is used both Feedback image and Multipass images are saved

2. Enter any notes about the scan in the **Comment** area.
3. Click **Save**.
4. In the window that opens, choose a location and enter the name for the file.
5. Click **OK**.

   One file for each trace/retrace option that has been selected is saved, according to the number of channels. Therefore, if there are four channels and you have selected six trace/retrace options, then six files will be the saved. The file names are made up of the name you entered in Step 4 above, the name of the channel and the letter “T” or “R” for trace and retrace.

The files are saved as either raw data or calculated data according to the “Processing” value selected.

The Lock-In Controller

The Lock-In Control screen appears below:

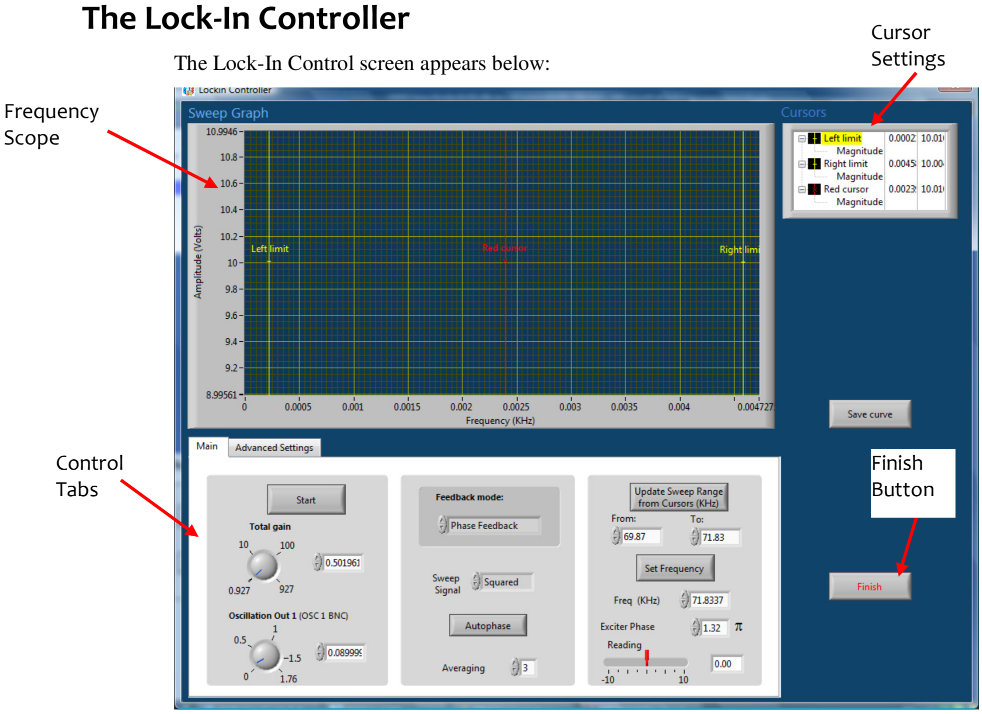


Figure 26: The LockIn Controller

The LockIn Controller has the following tabs:

- Main
- Advanced Settings

The Main tab

The Sweep tab is used to obtain a resonance curve and to set the excitation frequency.

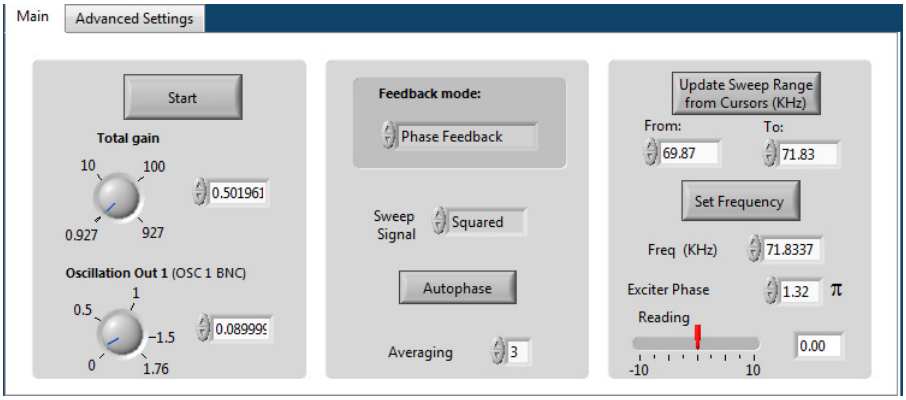


Figure 27: The Main tab

- The **Oscillation Out 1** is used to set the oscillation amplitude. The value is changed from 0 to 1.76V
- **Total Gain** is used to amplify the signal coming from the probe.
- There are two **Feedback modes:** Phase Feedback and Amplitude Feedback modes. Select the mode you are currently using for the feedback.
- The signal measured during the sweep appears in the **Sweep Signal** field. For amplitude feedback set **Sweep Signal** to "Magnitude"; for phase feedback set to **Sweep Signal** to "Squared".
- Set the number of measurements to be made (and averaged) at each point in the **Averaging** field.
- The phase of the signal can be determined automatically by clicking **Autophase** button. In this case the return signal at the AC IN terminal of the controller will be kept in phase with one of the internally generated signals.
- The frequency range can be set either by entering values in the **Lowest Frequency** and **Highest Frequency** fields, or by dragging the yellow cursors to the required positions on the Frequency Scope and then clicking **Update Sweep Range from Cursors**.
- Click **Start** to start the sweep.

The Advanced Settings Tab

The Advanced Settings tab appears below.

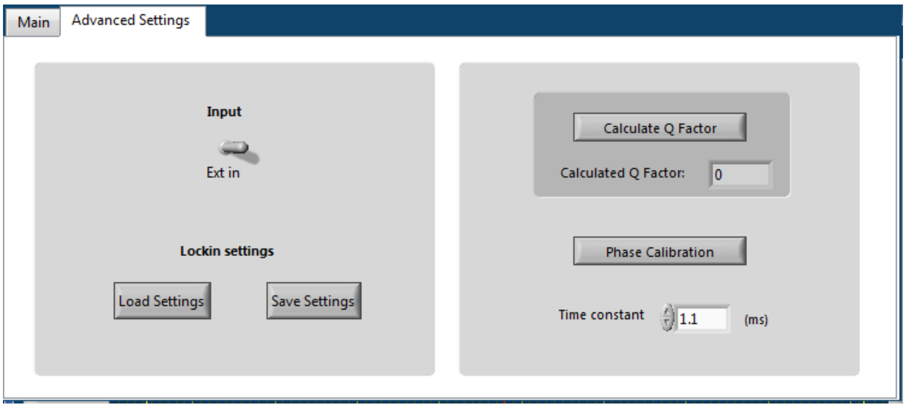
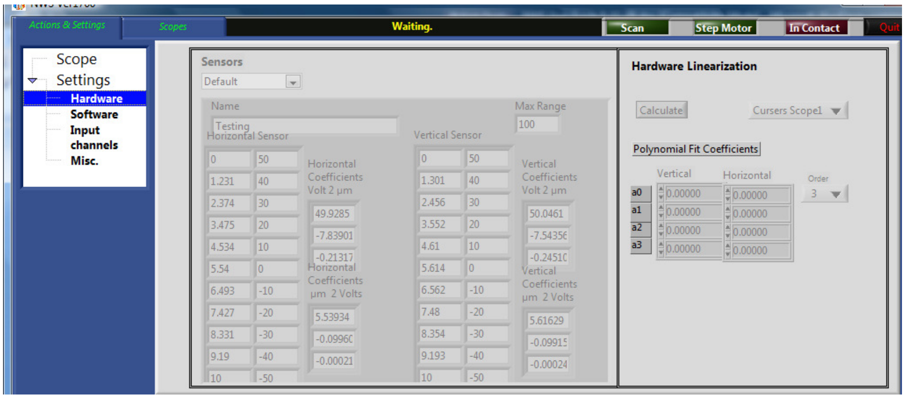


Figure 28: The Advanced Settings tab

- There are two inputs on the rear panel of SPM controller: **Ext In** and **AC In.** Users who have MultiView1000 system should put the knob in **AC In** position. Users, who have all other systems should put the knob in **Ext In** position
- **Calculate Q Factor** button allows calculating Q factor for tips.
- **Phase Calibration** allows calibrating the phase for phase imaging.
- **Time constant** determines the integration time of the Lock-in.

Appendix 1

Hardware Settings



The Hardware Settings screen is shown below:

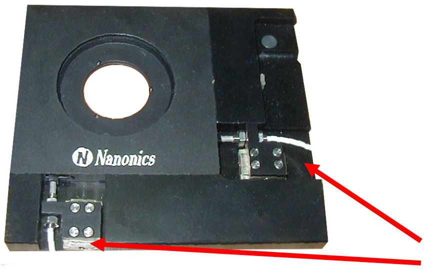
Figure 29: The Hardware settings

The Hardware settings are only for scanner with hardware sensors attached. Many of the options are for the use of Nanonics technicians only.

Field	Description
Hardware Linearization	This option allows polynomial fitting of data coming from the sensors.
Sensors	Read-only data from a file that is relevant when using hardware sensors. When the sensors are changed, a new data file is required.

Hardware Linearization

NOTE: This option is only available on systems with sensors.



Use this option to adjust the hardware sensors when changing the scan range. NWS calculates new coefficients when the scan parameters are changed (range, axes, etc.) so that the sensors will adjust correctly.

>> ***To set the output voltage to 5 V***

6. Click the **SPM Control** tab.
7. From "Read channel" list choose **SensorX** channel
8. Check "Continuously" option and click on **Read** button.
9. Now turn the screws with a screwdriver until the value in the Sensor X channel reaches 5 V.
10. Change the channel to **SensorY** channel
11. Now turn the screws with a screwdriver until the value in the Sensor Y channel reaches 5 V.

IMPORTANT: Take care not to break the inductive sensors.

>> ***To calculate new coefficients***

12. From the **Formula** drop-down list, choose the calibration that is relevant for this scanner.
13. Choose desirable Window size, Time per point and resolution.
14. Click **Start**. At this stage, the tip does not need to be in contact with the sample.
15. When the scan has finished, move all the bottom and left cursors in Scope 1 a short distance to "crop" the edge of the scanned image.
16. Click **Calculate** in the Hardware Linearization area. Ensure that "Cursors Scope 1" is selected.

>> ***To carry out a scan using the new coefficients***

17. Bring the tip into contact by clicking **Approach** in the **SPM Control** tab.

    From the **Formula** drop-down list choose "Sensors".

18. You can now start a scan.

Appendix 2

Quick Lock-in setup for Amplitude and Phase feedback

Feedback mode: Amplitude feedback

1. Adjust Feedback Mode to **Amplitude Feedback.**

2. Sweep signal: **Magnitude.**

    a. Sweep over the range of 20-50kHz for MV2000/4000
    b. Sweep over range of 20-120kHz for MV1000

3. Use the yellow cursors and 'update value from cursors' button to zoom on the peak.

4. **Total gain:**

    a. For EXT In input use the value of ~10. For softer samples increase total gain, but only as far as noises don't appear during scanning.
    b. For AC in Input use the value of ~1

5. **Ocs. Amp** - should be adjusted so total signal of amplitude would be ~8V

6. The red cursor automatically sets to maximum.

7. Press: 'Set frequency to pointer value'

8. Press Autophase (optional, Allows obtaining the phase image simultaneously)

9. Finish

10. Set the error value to be ~0.8-1.6 (~10-20% from the amplitude value) by adjusting the set point. Check the value of the error signal by clicking on READ. As mentioned the value of error 0.8-1.6 is the value when the probe is freely oscillates, the value of the error signal in contact should be zero!

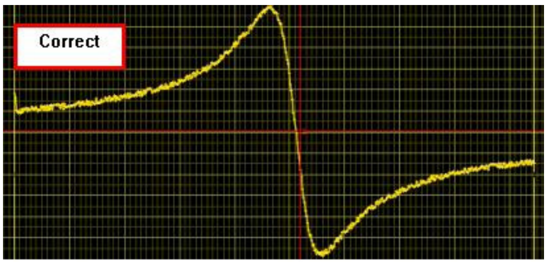
Note: For softer approach use lower values of set point.

11. Press Approach button.

Feedback mode: Phase feedback

Phase feedback works in a similar way to amplitude feedback, only that the feedback is based on the changes of the phase between the oscillation signal from the probe and the reference signal of the oscillator in the lock-in. Phase feedback was found to be much more sensitive than the amplitude feedback. In this mode of operation there is less energy dissipation between the probe and the sample. The amplitude of the probe is essentially stays the same. It is recommended to use phase feedback rather than amplitude feedback with the AFM probes and for soft materials.

1. Change Feedback mode to **Phase feedback**
2. Sweep signal: **Magnitude**. Sweep over the range of 20-50kHz
3. Use the yellow cursers and 'update value from cursers' button to zoom on the peak.
4. **Total gain:** use the value of ~10. For softer samples increase total gain, but only as far as noises don't appear during scanning
5. **Ocs. Amp** - should be adjusted so total signal of amplitude would be ~8V
6. The red cursor automatically sets to maximum.
7. Press: 'Set frequency to pointer value'
8. Press Autophase (optional, Allows obtaining the phase image simultaneously)
9. Change the sweep signal to: **Squared** and press **Start.**
10. Put the red cursor on the center of the curve:

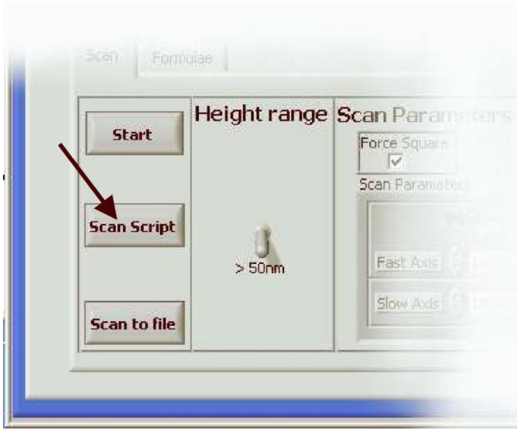


11. Press: '**Set frequency to pointer value**'
12. Finish
13. Set the error value to be ~0.3 by adjusting the set point. Check the value of the error signal by clicking on READ. During scanning you can increase set point (making error higher than 0.3) for having more pressure on the probe.
14. Click **Approach** button

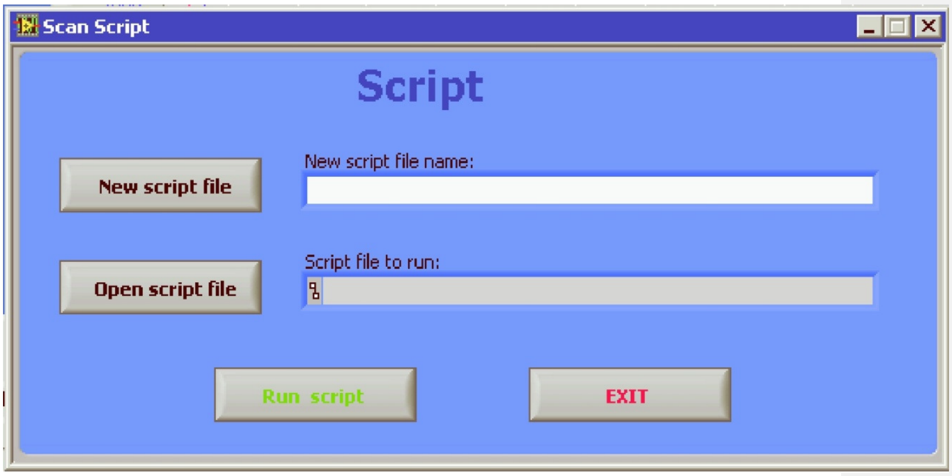
Appendix 3

Scan Script option: to run a script with specifications for multiple scans

New feature that implements a build script file for multiple various scans, with dissimilar parameters for each scan.



Pressing the “Scan Script” button will open the following window:



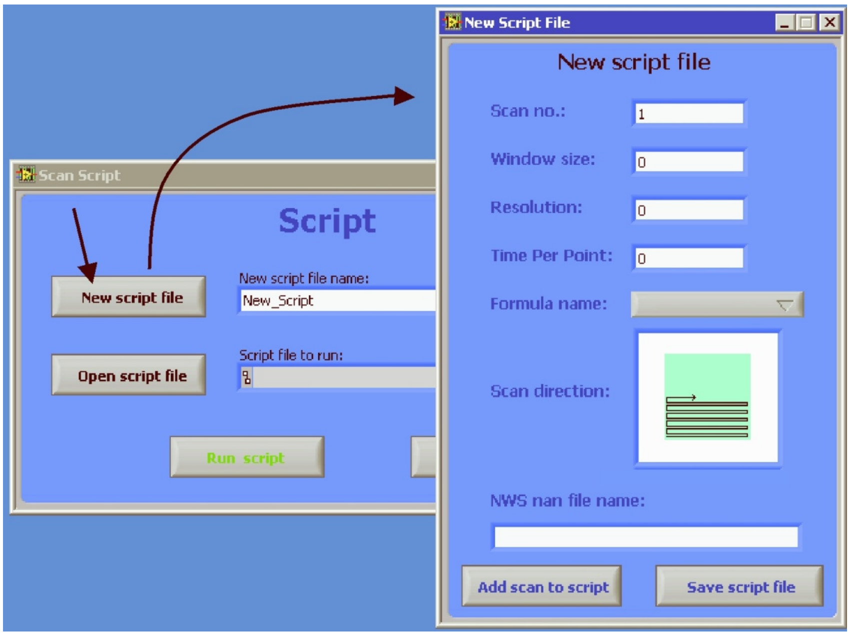
The “Open script file” button allows opening an existing script file.

The “Run script” button will implement the opened script file.

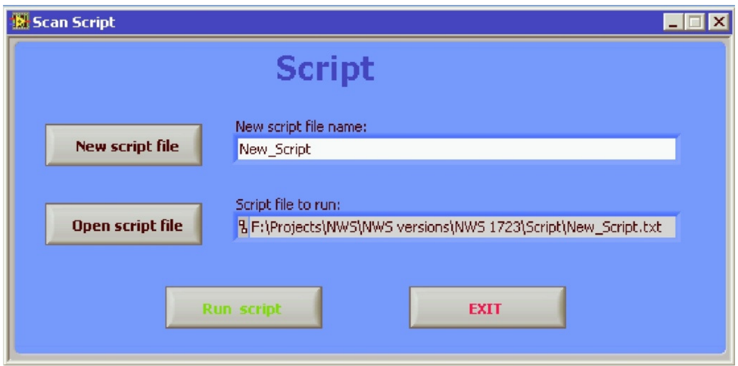
To build a new script file follow the next steps:

1. Fill the “new script file name:” field with the desirable script file name. If you forget to fill it, a new window will appear after pressing the “New script file” button, asking for a new script file name.

2. Press the “New script file” button. Now, a new window will appear:



3. In this window, you can choose the window size, resolution, time per point, formula, scan direction and the NWS nan file name for each scan. The NWS nan file name can be automatically set as Scan_%no.%_Window_size_%no.%.nan in the “Script Scans” directory, for each scan respectively.
4. Press the “Add scan to script” button to add a new scan to the script file.
5. When finished, press the “Save script file” to save the script file, exit this window and returning to the scan script window.
6. Now, the new script file will appear in the “Script file to run:” field (As if we opened it):



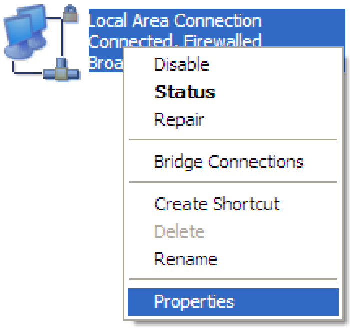
7. Press the “Run script” to implement the script file or “Exit” to cancel

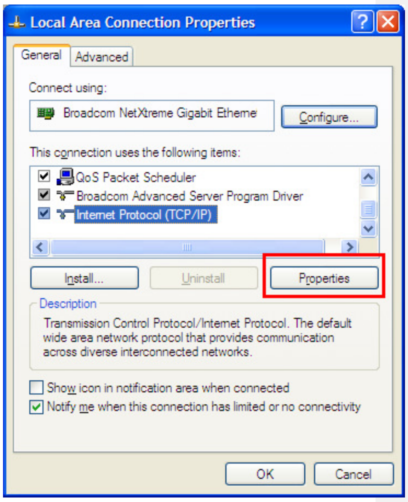
Appendix 4

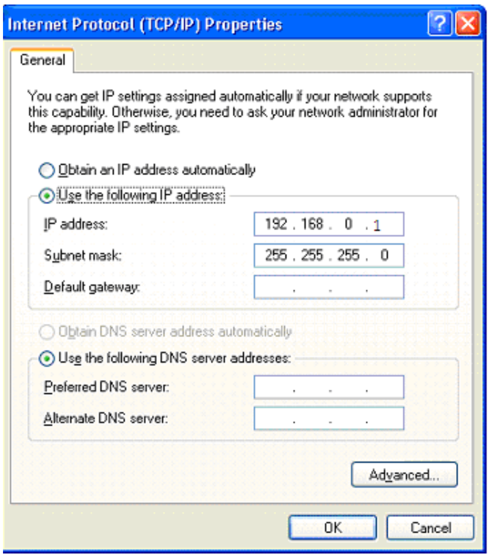
Connection between 2 computers.

For XP Users

1. Connect both computers with Lan Cable
2. Open **Network Connections**
3. Right-click on Local Area connection
4. Open the **Properties** window
5. Click on the **Internet Protocol (TCP/IP)** and click on Properties.
6. In the opened window fill in the IP address: 192.168.0.1, the Subnet mask will be filled in automatically
7. Click Ok to close the window.







For Win7 users

1. Connect both computers with Lan cable
2. Open Control *Panel -> Network->Network connections*
3. Right click on Local Area Connection and choose **Properties** window**.**
4. Click on the **Internet Protocol Version 4 (TCP/IPv4)** and click on Properties.
5. In the opened window fill in the IP address: 192.168.0.1, the Subnet mask will be filled in automatically
6. Click Ok to close the window.