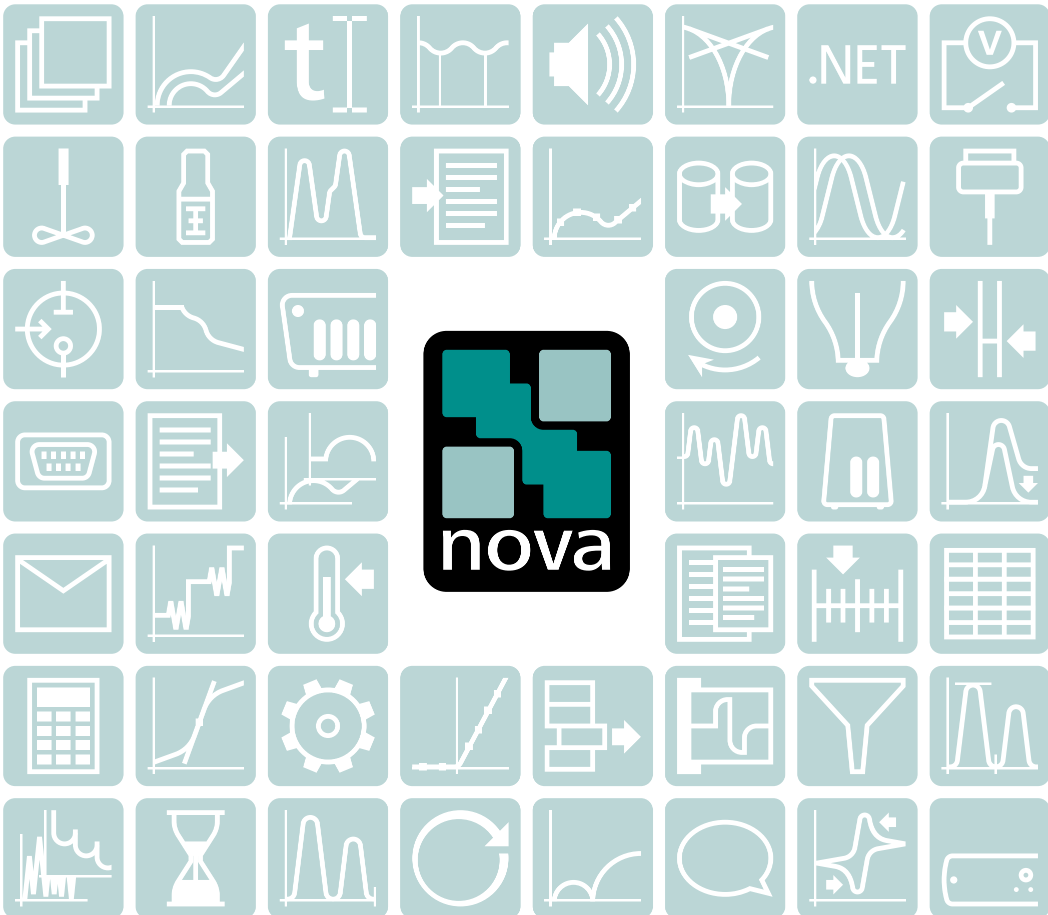
NOVA



User Manual

2.1.2 / May, 2017

Metrohm
Autolab B.V.

Metrohm Autolab B.V.
Kanaalweg 29/G
3526 KM, Utrecht
The Netherlands
+31302893154
autolab@metrohm.com
www.metrohm-autolab.com

NOVA

2.1.2

User Manual

2.1.2 /
May, 2017

MVB/JVD

Metrohm Autolab Teachware
Metrohm Autolab B.V.
3526 KM, Utrecht

Although all the information given in this documentation has been checked with great care, errors cannot be entirely excluded. Should you notice any mistakes please send us your comments using the address given above or at autolab@metrohm.com.

Table of contents

1 NOVA installation

This chapter describes how to install the NOVA software on the host computer and to how connect Autolab and external devices to the host computer.

The NOVA installation package is supplied on CD-ROM or USB support provided with the Autolab instrument. It can also be downloaded from the Metrohm Autolab webpage.

NOTE

Leave the Autolab disconnected from the computer when installing NOVA for the first time.

1.1 Software compatibility

NOVA requires Windows 7 or later as operating systems in order to run properly. NOVA can be installed on 32 bit and 64 bit versions of Windows.

NOTE

Previous versions of Windows are not supported.

The minimum and recommended specifications are reported in *Table 1* and *Table 2*, respectively.

Table 1 Overview of the minimum specifications for NOVA

CPU	1 GHz or faster 32-bit (x86) or 64-bit (x64) processor
RAM	2 GB RAM
HD	20 GB available hard disc space
GPU	DirectX 9.0c compliant display adapter with 64 MB RAM

Table 2 Overview of the recommended specifications for NOVA

CPU	Intel Core i5 or equivalent AMD processor
RAM	8 GB RAM

HD	128 GB available hard disc space
GPU	DirectX 9.0c compliant display adapter with 1 GB RAM

NOTE

The installation of the .NET 4.5 framework is required in order to install NOVA. If the .NET framework is already installed, the installation wizard will directly install NOVA on the computer. If the .NET framework is missing or of the correct version is not found on the computer, you will be prompted to install the .NET framework before the installation of NOVA can be completed.

1.2 Hardware compatibility

NOVA provides support for all Autolab instruments with a USB interface (internal or external), **except** the following legacy instruments:

- µAutolab (type 1)
- PSTAT10

Furthermore, NOVA provides support for all Autolab extension modules, **except** the following legacy modules:

- FRA modules (first generation)
- ADC124
- DAC124
- DAC168

1.3 Software installation

Double click the **nova-setup.exe** executable provided on the installation CD-ROM or USB support or downloaded from the Metrohm Autolab website to start the installation of the NOVA software.

An installation Wizard will be started *(see Figure 1, page 3)*.

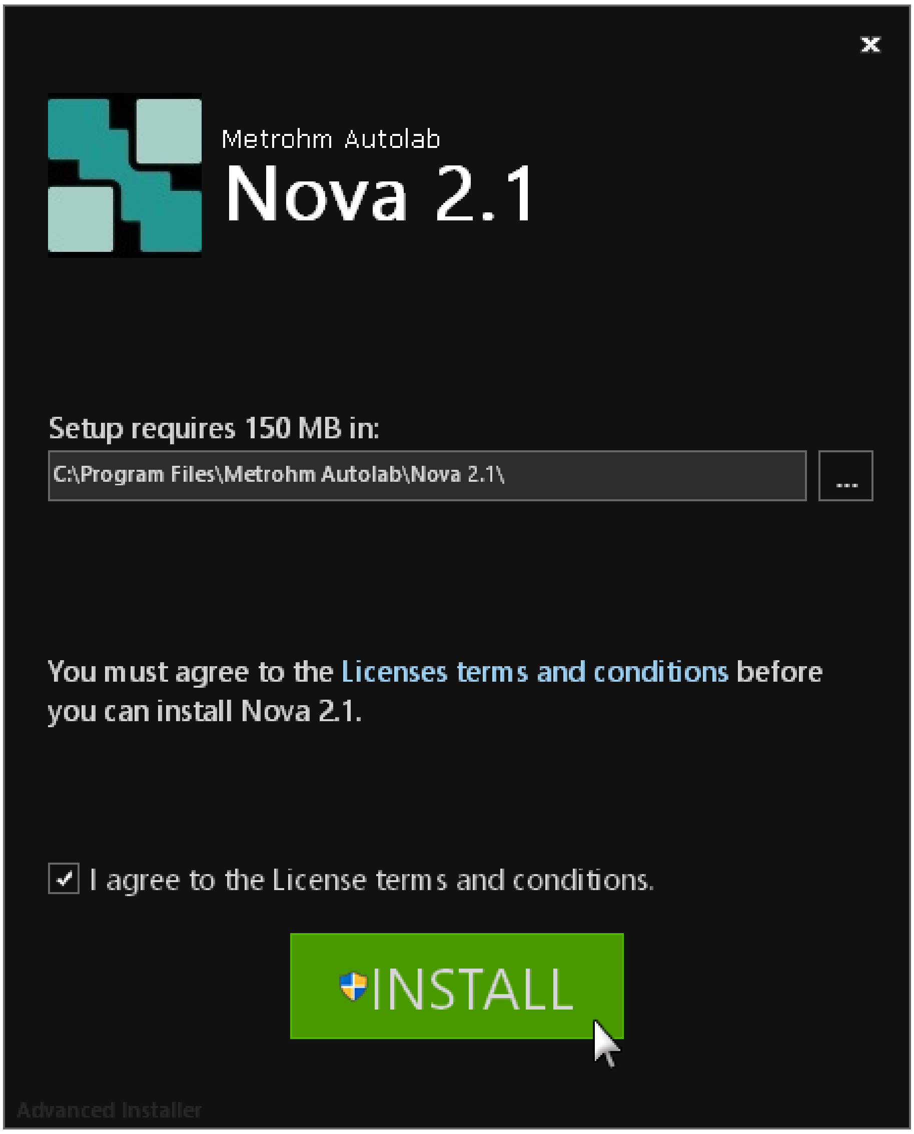


Figure 1 The installation wizard

Click the **INSTALL** button to start the installation. The files will be copied on the computer. If needed, the installation folder can be changed using the installation wizard.

When prompted to do so, please click the **Install** button provided in the Metrohm Autolab Driver installation window *(see Figure 2, page 4)*.

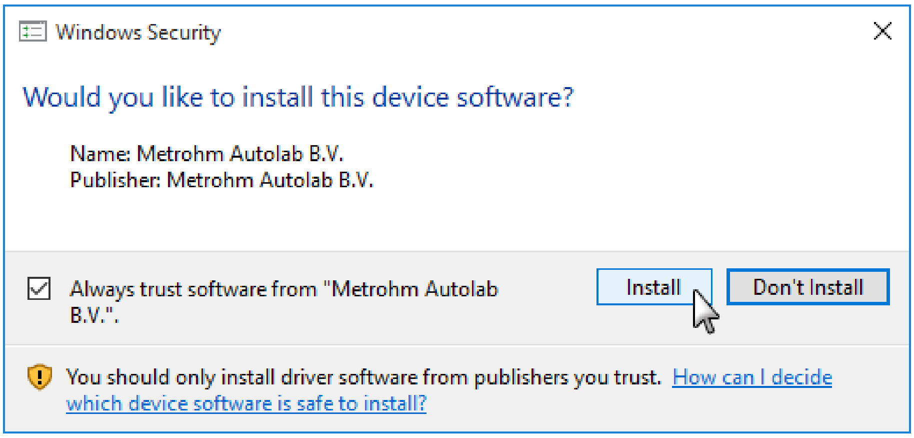


Figure 2 Install the Metrohm Autolab device drivers

NOTE

Make sure that the Always trust software from Metrohm Autolab B.V. check box is ticked.

When prompted to do so, please click the **Install** button provided in the Avantes Driver installation window *(see Figure 3, page 4)*.

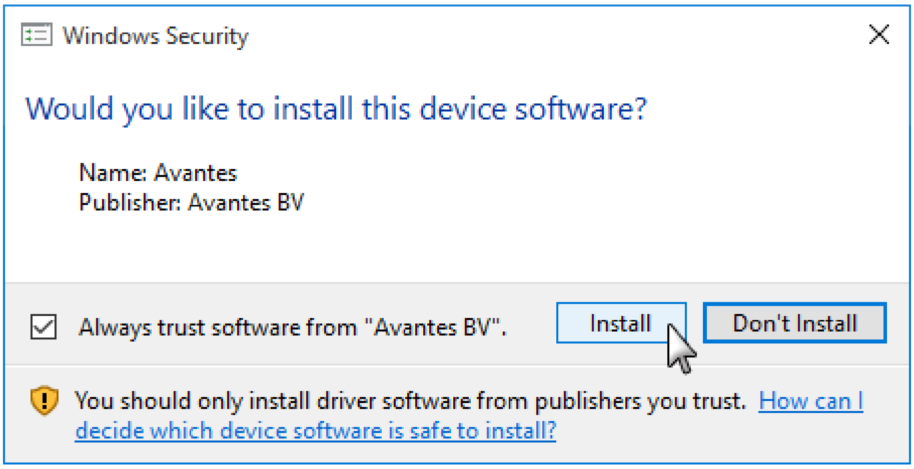


Figure 3 Install the Avantes device drivers

NOTE

Make sure that the Always trust software from Avantes BV check box is ticked.

The installer will indicate when the installation is completed, as shown in *Figure 4*.

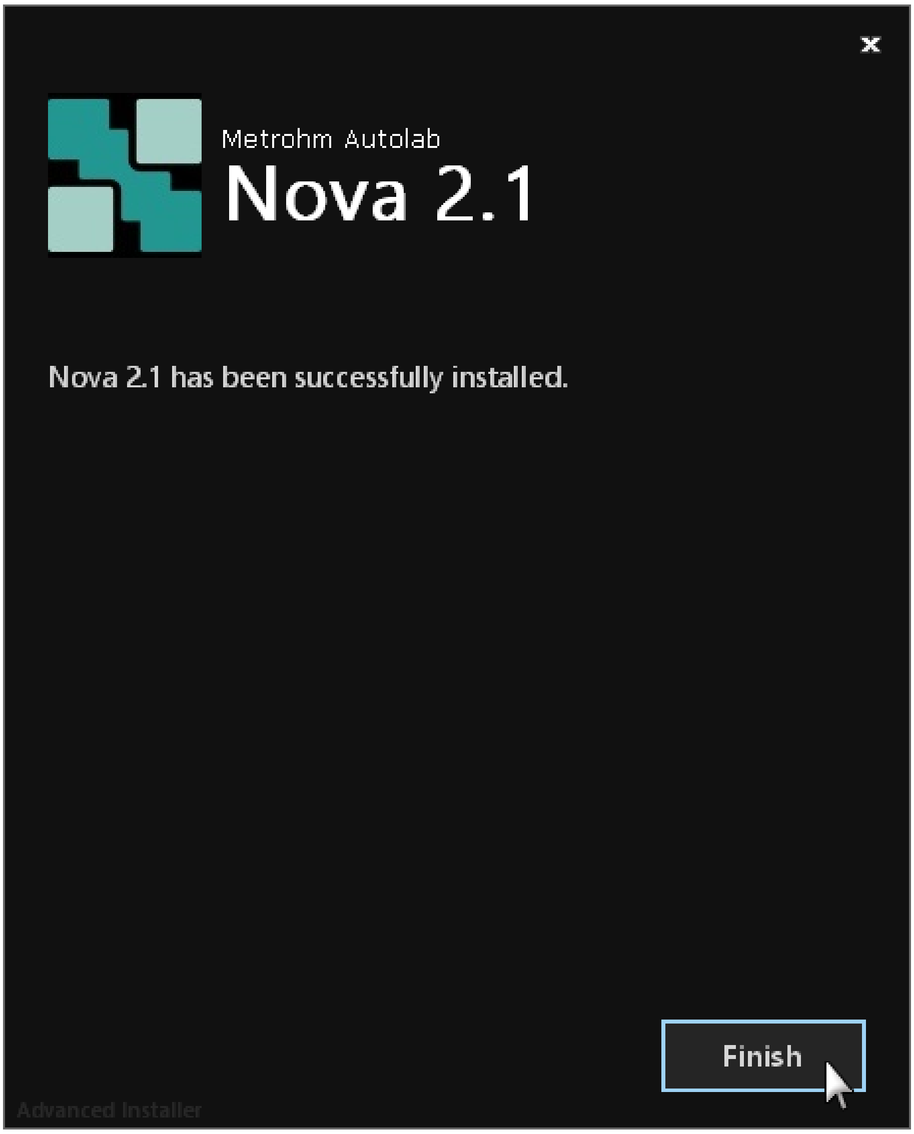


Figure 4 The installation is complete

1.4 External devices

The following additional external devices can be connected to the host computer:

- **Metrohm liquid handling devices:** these devices can be used to handling liquid samples and to automate the handling thereof.
- **Spectrophotometers:** the Autolab or the supported Avantes spectrophotometers can be used to perform spectroelectrochemical measurements in combination with the Autolab potentiostat/galvanostat.
- **Autolab RHD Microcell HC:** this device can be used to perform temperature-controlled measurements.

1.4.1 Metrohm Devices support

NOVA provides support for a selection of Metrohm liquid handling and automation devices.

The following devices are supported through a **USB** connection to the host computer:

- Metrohm 814 USB Sample Processor
- Metrohm 815 Robotic USB Sample Processor
- Metrohm 846 Dosing Interface
- Metrohm 858 Professional Sample Processor

The following devices are supported through a **MSB** connection to one of the **USB** controlled devices listed above:

- Metrohm 800 Dosino
- Metrohm 801 Magnetic Stirrer
- Metrohm 803 Titration Stand with Stirrer and Pump
- Metrohm 804 Titration Stand
- Metrohm 6.2148.010 MSB Remote Box

The following devices are supported through a **specific** connection to one of the **USB** controlled devices listed above:

- Metrohm 802 Rod Stirrer
- Metrohm 741 Magnetic Stirrer
- Metrohm 786 Swing Head
- Metrohm 843 Pump Station
- Metrohm 823 Membrane Pump
- Metrohm 772 Peristaltic Pump

NOTE

At least one Metrohm device connected through **USB** is required in order to control the supported devices.

NOTE

The supported Metrohm devices can be used with or without the Autolab connected to the computer.

1.4.2 Metrohm Devices installation

Connecting a **USB** controlled Metrohm device to the host computer will trigger the installation of the instrument *(see Figure 5, page 7)*.

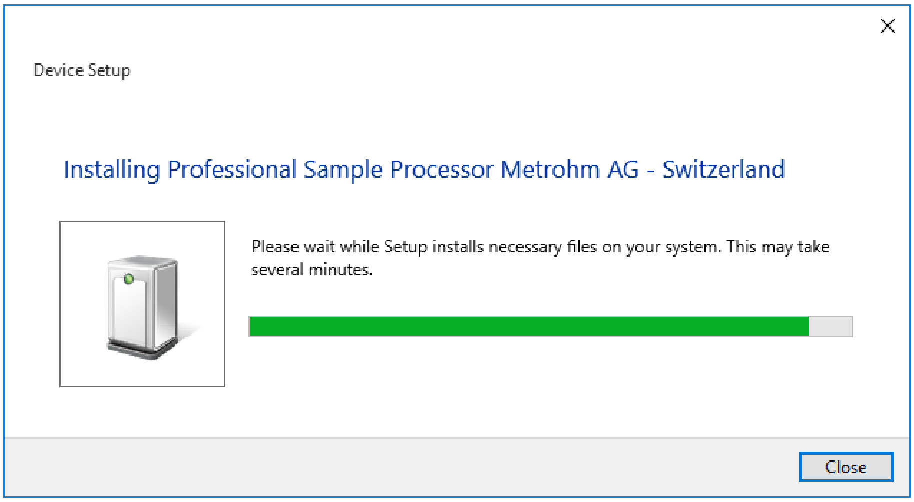


Figure 5 The Metrohm Device Driver installer

The installation will complete automatically.

NOTE

In order to control the supported Metrohm Sample Processors, an additional Windows component must be present on the computer. The controls for the Metrohm Sample processors use the **Microsoft msxml6.0.dll** library for the configuration files (XML file format). This component may not be preinstalled on every Microsoft operating system. Please ensure the availability of this dll on the operating. If this package is missing, please download the installation package from the Microsoft website.

1.4.3 Spectrophotometer support

NOVA provides support for Autolab spectrophotometers.

The following Autolab spectrophotometer are supported through a **USB** connection to the host computer:

- Autolab Spectrophotometer UA
- Autolab Spectrophotometer UB

Additionally, compatible Avantes spectrophotometers are also supported when connected to the host computer through a **USB** connection. The following devices are supported:

- AvaSpec ULS2048-USB2

- AvaSpec ULS3648-USB 2

NOTE

The supported spectrophotometers can be used with or without the Autolab connected to the computer.

1.4.4 Spectrophotometer installation

Connecting a USB controlled Metrohm Autolab Spectrophotometer (or a compatible Avantes Spectrophotometer) to the host computer will trigger the installation of the instrument *(see Figure 6, page 8)*.

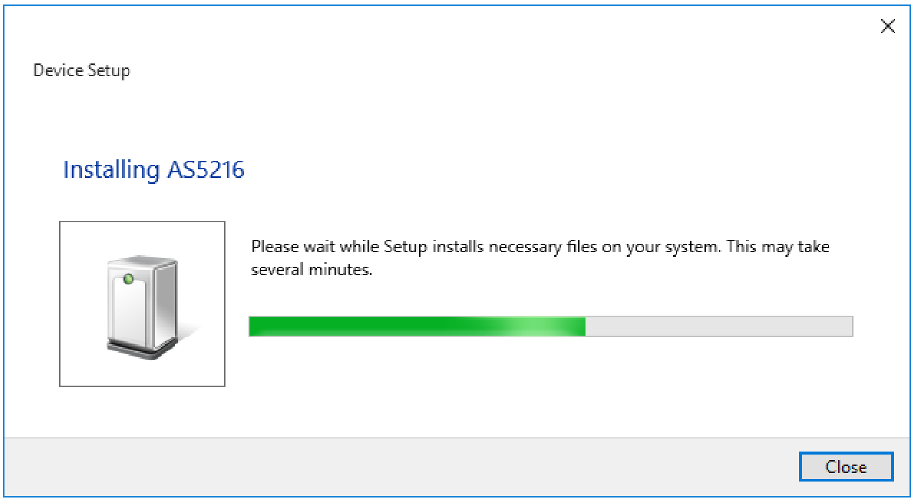


Figure 6 The Spectrophotometer installer

The installation will complete automatically.

1.4.5 Autolab RHD Microcell HC support

NOVA provides support for the complete Autolab RHD Microcell HC product range.

NOTE

The supported Autolab RHD Microcell HC devices can be used with or without the Autolab connected to the computer.

1.4.6 Autolab RHD Microcell HC installation

No driver is required to control the Autolab RHD Microcell HC. When this type of device is connected to the host computer, it is immediately recognized by NOVA and listed in the **Instruments** panel.

NOTE

The Autolab RHD Microcell HC is only detected by NOVA if a stage is connected to the controller with a cell mounted on the stage.

NOTE

The Autolab RHD Microcell HC is connected to the host computer through a serial port. If no serial port is present on the computer, a USB to Serial port adapter can be installed. The drivers required for this adapter are not included in the installation package of NOVA and need to be installed separately.

1.5 Powering the instrument

In order to use the instrument, it must be connected to the mains using the mains connection socket, located on the back plane of the instrument. Before connecting the instrument to the mains make sure that the mains output voltage matches the value indicated on the main voltage indicator, located above the connector *(see Figure 7, page 9)*.

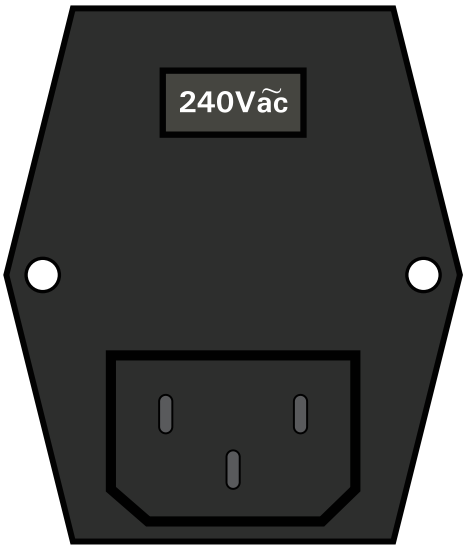


Figure 7 The required mains voltage is indicated above the connector

CAUTION

Connecting Autolab instrumentation to the wrong mains voltage may result in damage to the instrument.

1.6 Autolab hardware installation

After NOVA has been installed, connect the Autolab instrument to the computer using an available USB port. Switch on the instrument or plug the power supply in the external USB interface.

This will trigger the installation of the instrument *(see Figure 8, page 10)*.

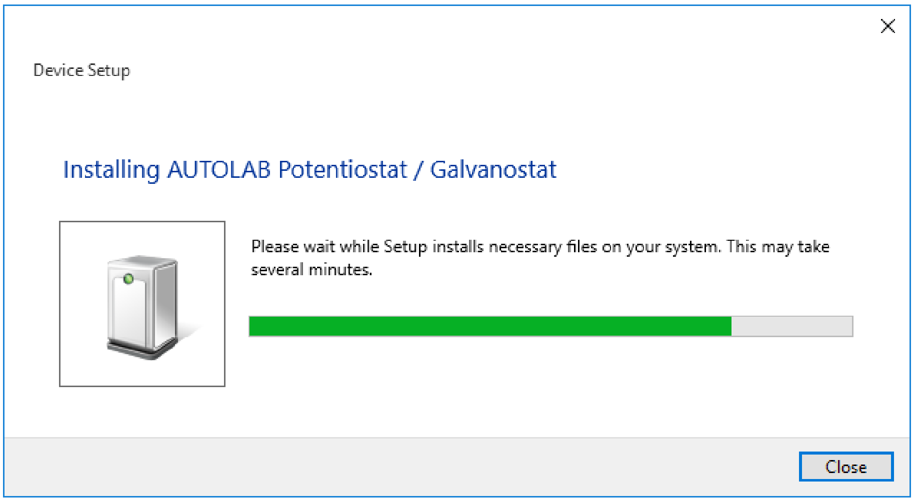


Figure 8 The Autolab installer

The installation will complete automatically.

NOTE

The **Nova only** driver is used during this installation process.

If needed, it is possible to adjust the driver used to control the Autolab potentiostat/galvanostat. Start the Autolab Driver manager application by using the shortcut provided in the Start menu (All Programs – Autolab – Tools – Driver manager). This will start the Driver Manager application *(see Figure 9, page 11)*.

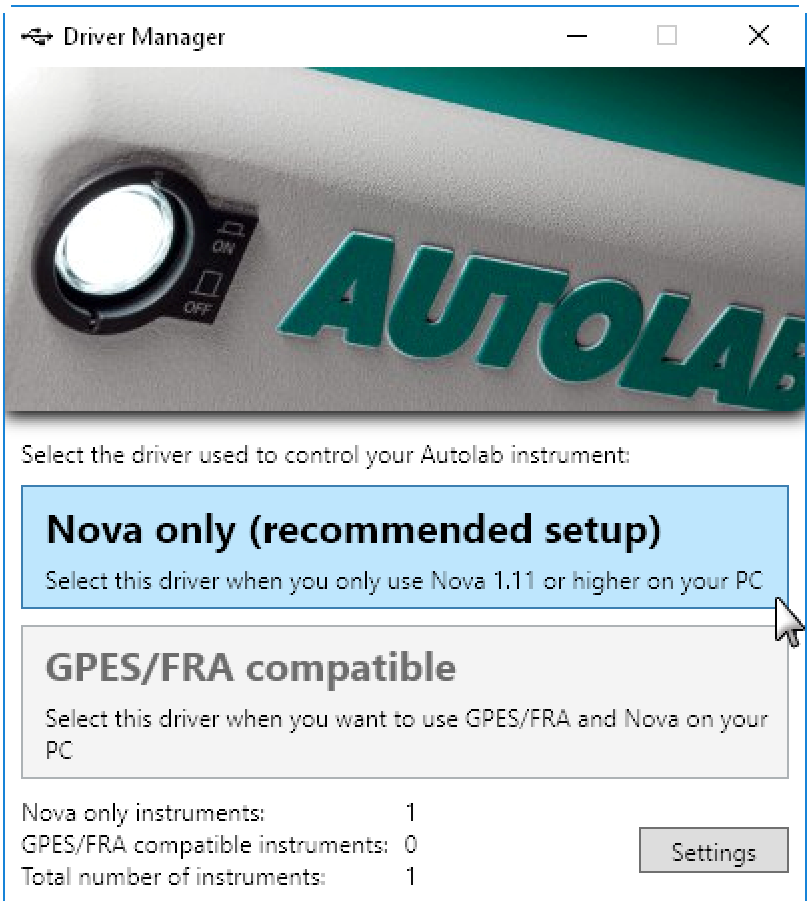


Figure 9 The Driver Manager application

The Driver Manager can be used at any time to select the driver to use to control the Autolab. Two drivers are available:

- **Nova only (recommended setup):** this is the latest driver for the Autolab, allowing up to 127 instruments to be connected to the host computer. This driver is compatible with 32 bit and 64 bit versions of Windows.
- **Legacy driver:** this is an older driver version which can be used in combination with the GPES or FRA software. No further developments are planned for this driver. The maximum number of devices connected to the host computer is 8. Data transfer may be slower than with the NOVA only driver. This driver is **only** compatible with 32 bit versions of Windows.

NOTE

The **Nova only** driver will not work with previous versions of NOVA (version 1.10 and older). In order to use previous versions of NOVA, it is necessary to start the Driver Manager application provided with the previous version and select one of the available drivers provided with this previous version (please refer to the **Getting Started** manual of the previous version of NOVA for more information).

Click the installation button for the required driver to change the device driver and follow the instructions on screen. The selected driver will be installed for all connected Autolab instruments. New instruments connected to the host computer will be configured using the selected driver.

NOTE

The **Driver Manager** application can be used to change the device driver at any time.

1.7 Software license

The Autolab NOVA software, and all its components, provided in conjunction with the Metrohm Autolab potentiostat/galvanostat instruments is copyrighted and owned by Metrohm Autolab.

The software is provided as a **Free Licensed Closed-Source** product with limited warranty. The software can be installed on any computer without specific authorization from Metrohm Autolab.

Metrohm Autolab retains the copyright to the software. You may neither modify nor remove references to confidentiality, proprietary notices or copyright notices. Modifications of the software in part or as a whole is not permitted.

Metrohm Autolab warrants that the software, when operated properly, is suitable for the specified use with the electrochemical instrumentation from Metrohm Autolab or compatible external instrumentation.

Metrohm Autolab is exempt from further warranty or liability. Metrohm Autolab is neither liable for third-party damages or consequential damage not for loss of data, loss of profits or operating interruptions, etc.

1.8 Intended use

All Metrohm Autolab products are designed for electrochemical research and development within the normal environment of a laboratory. The instrumentation shall therefore only be used for this purpose and within the specified environmental conditions. All other uses fall out of the scope of the instrumentation and may lead to voiding of any warranty.

1.9 Options

The application options can be defined by selecting the *Options* from the **Edit** menu. A window will be displayed, showing two different sections *(see Figure 10, page 13)*.

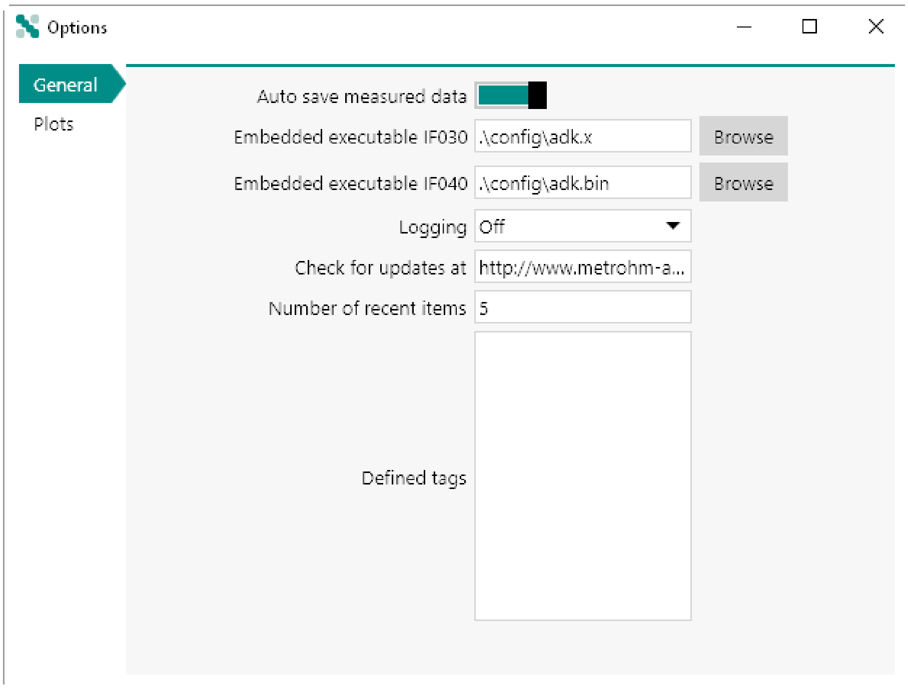


Figure 10 The application Options window

The following properties are available in the **General** section *(see Figure 10, page 13)*:

- **Auto save measured data:** specifies if measured data should be saved automatically at the end of each measurement, using the provided toggle. This option is on by default.
- **Embedded executable IF030:** specifies the path to the embedded application for instrument fitted with the **IF030** controller, using the provided Browse button. This is a system property, do **not** change this unless instructed by Metrohm Autolab.

- **Embedded executable IF040:** specifies the path to the embedded application for instrument fitted with the **IF040** controller, using the provided Browse button. This is a system property, do **not** change this unless instructed by Metrohm Autolab.
- **Logging:** specifies if error logging should be used and which level of logging should be used, if applicable, using the provided drop-down list. This is a system property, do **not** change this unless instructed by Metrohm Autolab.
- **Check for updates at:** specifies the URL for version checks of NOVA. This is a system property, do **not** change this unless instructed by Metrohm Autolab.
- **Number of recent items:** defines the number of recent items shown in the **Recent items** panel of the dashboard. The default value is 5. Please refer to *Chapter 4.2*, for more information on the recent items.
- **Defined tags:** provides a list of tags used in NOVA. This list is empty by default and is automatically populated by user-defined tags through the tagging feature of NOVA. If needed, tags can be removed or added to this list directly.

NOTE

More information on the use of tags can be found in *Chapter 6.8*.

CAUTION

Modifying the system properties shown in the **General** section can interfere with the operation of the instrument. Do **not** change these properties unless instructed by Metrohm Autolab.

The **Plots** section displays the default plot options used in NOVA for all plots *(see Figure 11, page 15)*.

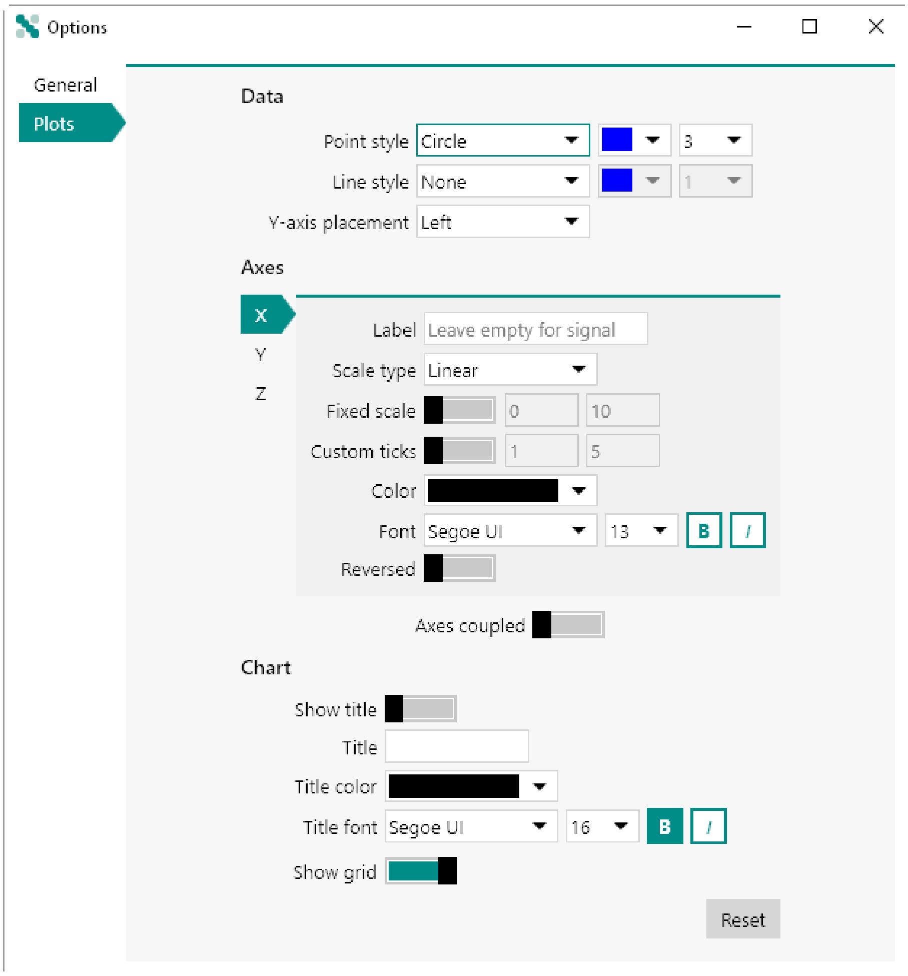


Figure 11 The plot options

In this section all the default options for plots can be specified. Clicking the Reset button will reset all the options to the factory default values.

NOTE

Please refer to *Chapter 9.5.3* for more information on the Plot options.

2 Conventions

Throughout NOVA and all Metrohm Autolab products, the conventions detailed in this chapter are used.

2.1 Scientific conventions

The following scientific conventions are used:

- All units are specified in the International System of Units (SI), unless otherwise specified. See *B. N. Taylor, A. Thomson, The International System of Units (SI), NIST Special Publication 330, 2008 Edition* for more information.
- Electrochemical values like potential and current are indicated according to the International Union of Pure and Applied Chemistry (IUPAC) convention. Positive currents and (over)potentials are associated with oxidation processes. Negative currents and (over)potentials are associated with reduction processes. See *A. D. McNaught, A. Wilkinson, IUPAC, Compendium of Chemical Terminology: IUPAC Recommendations, Blackwell Science: Oxford, England; Malden, MA, USA, 1997* for more information.

2.2 Software conventions

The following standard interaction conventions are used in NOVA:

- A right-handed mouse where the left button is used for selecting items and the right button may open context-related menus is assumed.
- Quickly pressing and releasing the mouse button is called ‘Clicking’. A click of the left mouse button on a menu option, a button, an input item on the screen, will result in an action.
- Quickly pressing and releasing the right mouse button is called ‘Right-clicking’. A click of the right mouse button on a suitable location on the screen opens a context-sensitive menu, if applicable.
- By clicking and holding down the left mouse button you can ‘Drag’ items from one window and ‘Drop’ it in another by releasing the button. This action will be called ‘Drag and Drop’ and it is the key mechanism for creating a procedure.
- Quickly pressing and releasing the mouse button twice is called 'Double-clicking'. A double-click of the left mouse button is used to perform particular actions, and mainly is applied through standard usage in window actions.

The following selection methods are used in NOVA:

- To select any item on screen, click the item.
- To select consecutive items on screen, click the first item, press and hold down **[SHIFT]**, and then click the last item.
- To select nonconsecutive items on screen, press and hold down **[CTRL]**, and then click each item.

2.3 Numbering conventions

All **numeric** values are defined in NOVA according to the local culture defined for the Windows operating system. Depending on these settings, the decimal separator symbol can either be **.** or **,**.

Improper use of the local culture settings defined in Windows may lead to wrong values. For example, typing 0,3 in NOVA on a computer which uses the . as decimal separator will be validated as 3.

NOTE

It is recommended to consult the local culture settings defined in Windows before using the NOVA software.

Scientific (exponential) numbering is done using the **e** or **E** symbol. A value of **1e2** or **1E2** is converted to **100**.

The following prefixes are using in NOVA for engineering notation:

- **T**, for *Tera* (1000000000000).
- **G**, for *Giga* (1000000000).
- **M**, for *Mega* (1000000).
- **k**, for *Kilo* (1000).
- **m**, for *Milli* (0.001).
- **µ**, for *Micro* (0.000001).
- **n**, for *Nano* (0.000000001).
- **p**, for *Pico* (0.000000000001).

2.4 Warning label conventions

The following warning labels are using throughout the documentation provided with all the Metrohm Autolab products *(see Table 3, page 18)*:

Table 3 Warning conventions used in NOVA

	**Warning** This symbol draws attention to a possible life hazard or risk of injury.
	**Warning** This symbol draws attention to a possible hazard due to electrical current.
	**Warning** This symbol draws attention to a possible hazard due to heat or hot instrument parts.
	**Warning** This symbol draws attention to a possible biological hazard.
	**Caution** This symbol draws attention to a possible damage of instruments or instrument parts.
	**Note** This symbol marks additional information and tips.

2.5 NOVA information, warnings and errors

NOVA validates commands and command properties in real-time while the software is used or after each measurement. Depending on the situation, NOVA may provide validation information in the following way:

- **Information:** any item highlighted in blue indicates that information is available for the user to consider for improving the quality of the data. This indication is only provided at the end of a measurement, if applicable *(see Figure 12, page 19)*.

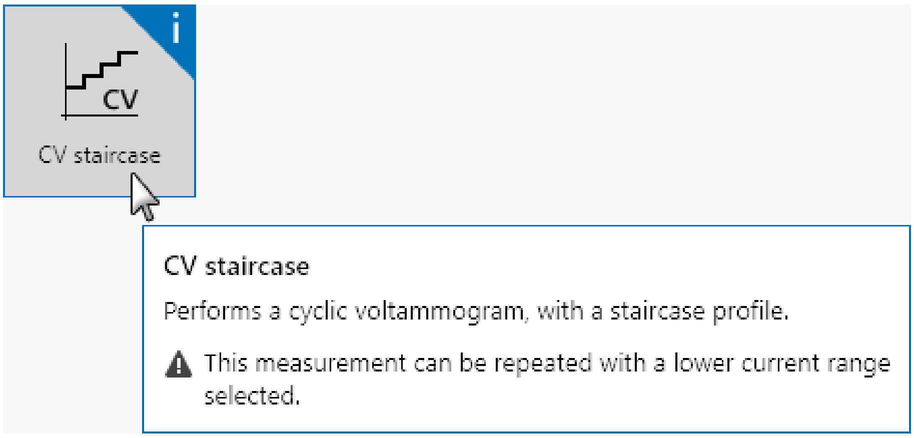


Figure 12 Information is highlighted in blue

- **Warning:** any item highlighted in yellow indicates that an issue has been identified and that user intervention is recommended in order to resolve the issue *(see Figure 13, page 19)*. Whenever possible, the cause and a possible solution will be offered. It is possible to ignore the warning and continue working with the software however this may lead to invalid data.

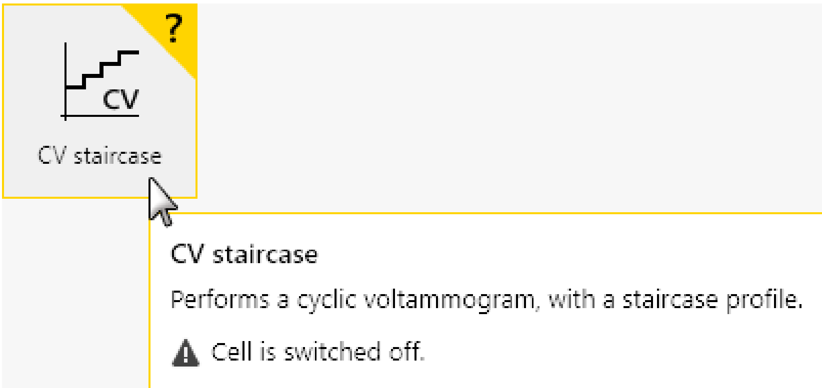


Figure 13 Warnings are highlighted in yellow

- **Error:** any item highlighted in red indicates that a problem has been identified and that user intervention is required in order to resolve the error *(see Figure 14, page 20)*. Whenever possible, the cause and a possible solution will be offered. It is **not** possible to ignore the error. No measurements are possible until the error is resolved.

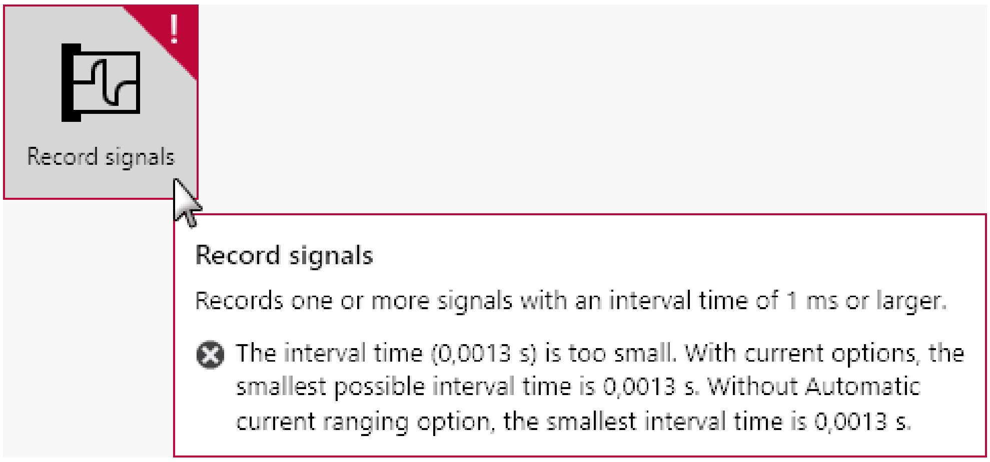


Figure 14 Errors are highlighted in red

2.6 NOVA menus and controls

This chapter describes the software menus and controls provided in NOVA.

File	
Close	Closes the active tab. Keyboard shortcut: **[CTRL]** + **[F4]**
Close All	Closes all open tabs. Keyboard shortcut: **[CTRL]** + **[SHIFT]** + **[F4]**
Save	Saves the content of the active tab as procedure, data or schedule. Keyboard shortcut: **[CTRL]** + **[S]**
Save As...	Saves the content of the active tab as procedure, data or schedule with the specified file and location.
Save All	Saves the content of all open tabs as procedure, data or schedule. Keyboard shortcut: **[CTRL]** + **[SHIFT]** + **[S]**
Exit	Closes the NOVA application. Keyboard shortcut: **[ALT]** + **[F4]**

Edit	
Undo 'Action name'	Undoes the specified action. Keyboard shortcut: **[CTRL]** + **[Z]**

Redo 'Action name'	Redoes the specified action. Keyboard shortcut: **[CTRL]** + **[Y]**
Cut	Cuts the selected item(s) to the clipboard. Keyboard shortcut: **[CTRL]** + **[X]**
Copy	Copies the selected item(s) to the clipboard. Keyboard shortcut: **[CTRL]** + **[C]**
Paste	Pastes the items in the clipboard at the specified location. Keyboard shortcut: **[CTRL]** + **[V]**
Select All	Selects all visible items. Keyboard shortcut: **[CTRL]** + **[A]**
Options	Specifies the default options used in the application.

View	
Zoom in	Zooms in on a plot. Keyboard shortcut: **[CTRL]** + **[=]**
Zoom out	Zooms out on a plot. Keyboard shortcut: **[CTRL]** + **[-]**
Fit all	Adjusts the plot area to the best possible scale. Keyboard shortcut: **[F4]**
Manual control	Displays the Manual control panel for the default instrument. Keyboard shortcut: **[F10]**

Measurement	
Run	Starts the procedure defined in the selected tab on the **default** instrument. Keyboard shortcut: **[F5]**
Run on ▸	Starts the procedure defined in the selected tab on the **specified** instrument.
Instrument #1	
Instrument #2	
...	

Pause	Pauses the running command in the selected tab.
Skip	Skips the running command in the selected tab.
Stop	Stops the measurement running in the selected tab.

Help	
User manual	Displays the NOVA User Manual. Shortcut key: **[F1]**
About	Displays the About dialog.

3 Release notes

This chapter describes the release notes of the current and previous versions of NOVA. The release notes are provided in reverse chronology. The following version have been released:

- **Version 2.1.2:** minor update of NOVA 2.1.
- **Version 2.1.1:** minor update of NOVA 2.1 *(see Chapter 3.2, page 23)*. This version was released on March 24th, 2017.
- **Version 2.1:** the current major release of NOVA *(see Chapter 3.3, page 33)*. This version was released on November 15th, 2016.
- **Version 2.0.2:** minor update of NOVA 2.0 *(see Chapter 3.4, page 47)*. This version was released on July, 6th, 2016.
- **Version 2.0.1:** minor update of NOVA 2.0 *(see Chapter 3.5, page 55)*. This version was released on April, 1st, 2016.
- **Version 2.0:** the original major release of NOVA 2 *(see Chapter 3.6, page 69)*. This version was release on October, 7th, 2015.

3.1 Version 2.1.2 release

Version 2.1.2 several bugs are corrected; no new functionality is added.

3.2 Version 2.1.1 release

Version 2.1.1 adds the following functionality:

1. New signal names and locations when using selected data analysis commands *(see Chapter 3.2.1, page 24)*.
2. Current range logged for all measurement commands *(see Chapter 3.2.2, page 26)*
3. Event logging for all measurement commands *(see Chapter 3.2.3, page 27)*.
4. Export possibility for Spectrophotometer manual control panel *(see Chapter 3.2.4, page 30)*.
5. Step through data option added to the Spectrophotometer manual control panel *(see Chapter 3.2.5, page 31)*.
6. A procedure for spectroelectrochemical measurements has been added to the **Default** procedures *(see Chapter 3.2.6, page 32)*.

3.2.1 Signal names, identity and locations

NOVA 2.1.1 changes the way the following analysis commands work:

- Smooth
- Derivative
- Integrate
- Baseline correction

In previous versions of NOVA, these commands created two result signals regardless of the number of signals in the source data. For example, when using the **Integrate** command in previous versions of NOVA, the calculated signals were called *Integration result X* and *Integration result Y*.

These calculated signals, only available in the analysis command itself, no longer had units or identity.

In NOVA 2.1.1, when either one of these four commands is used on data provided by a parent measurement command, the nature of the signals involved in the analysis command and their units are retained and the resulting signals are duplicated in the original parent measurement command.

For example, applying a **Smooth** command on i vs E data (WE(1).Current vs Potential applied), as shown in *Figure 15*, now produces two new signals called *Smoothed WE(1).Current* and *Potential applied*, as shown in *Figure 16*.

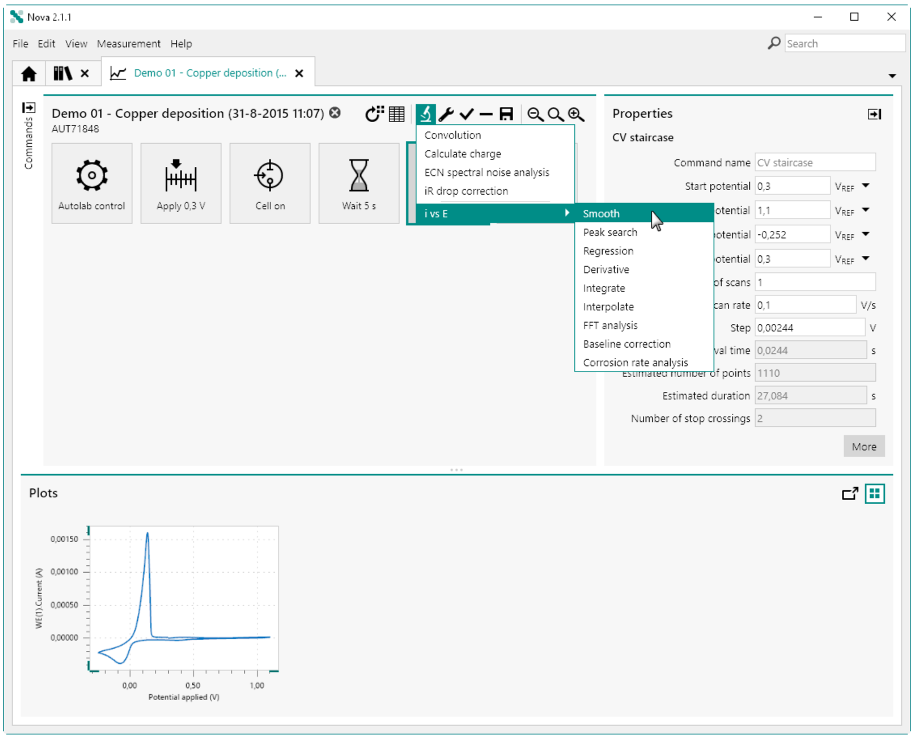


Figure 15 Adding a Smooth command to the i vs E plot

The calculated signals are automatically plotted *(see Figure 16, page 25)*.

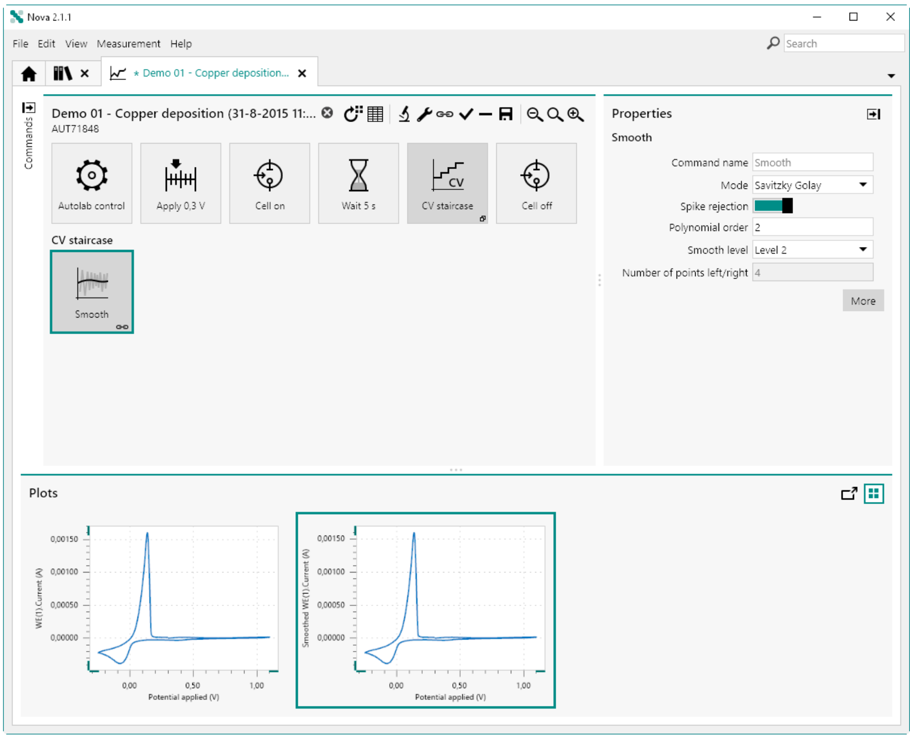


Figure 16 Smoothed WE(1).Current vs Potential applied plot is created

Additionally, the calculated data is copied to the parent measurement command (CV staircase), as shown in *Figure 17*.

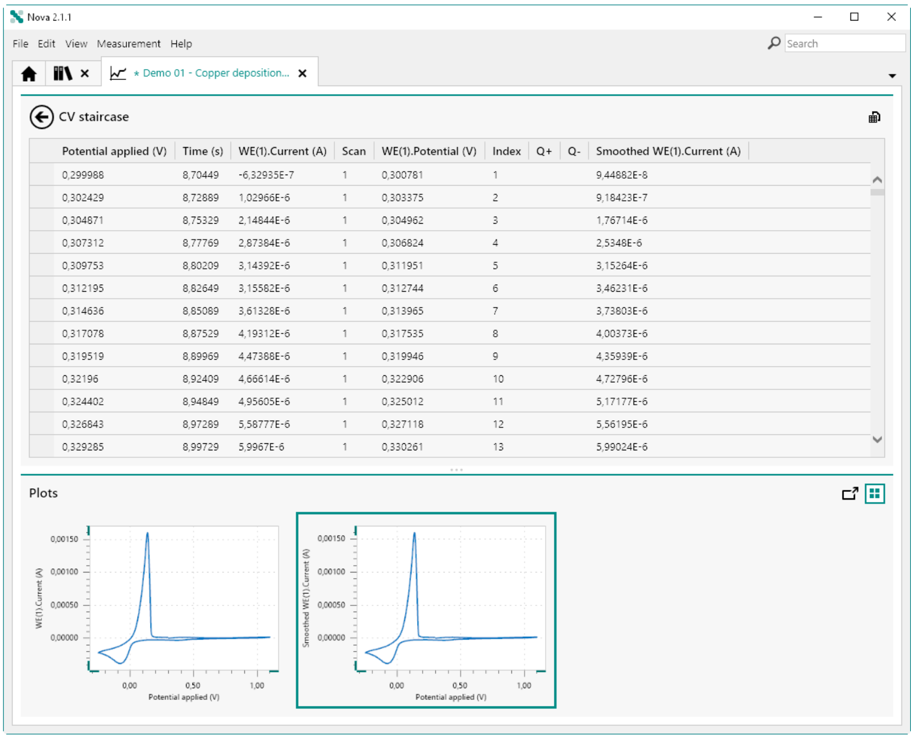


Figure 17 The calculated signals are also available in the parent command

The calculated *Smoothed WE(1).Current* signal is known by the NOVA procedure as a valid current signal, obtained by applying a **Smooth** command on the WE(1).Current signal of the **CV staircase** command.

3.2.2 Current range logging

NOVA 2.1.1 now logs the active current range for each data point recorded in all measurement commands. This information is stored in the data file and is reported in the data grid *(see Figure 18, page 27)*.

LSV staircase

Potential applied (V)	Time (s)	WE(1).Current (A)	WE(1).Potential (V)	Index	Current range
0,288086	10,4533	2,86011E-7	0,287781	118	100 nA
0,290527	10,4777	2,88239E-7	0,290253	119	100 nA
0,292969	10,5021	2,97638E-7	0,292664	120	100 nA
0,29541	10,5265	2,98279E-7	0,295074	121	100 nA
0,297852	10,5509	2,96173E-7	0,297607	122	100 nA
0,300293	10,5753	2,99164E-7	0,299957	123	100 nA
0,302734	10,5997	3,02643E-7	0,302338	124	100 nA
0,305176	10,6241	3,08868E-7	0,305298	125	1 µA
0,307617	10,6485	3,0954E-7	0,30777	126	1 µA
0,310059	10,6729	3,07281E-7	0,310394	127	1 µA
0,3125	10,6973	3,1076E-7	0,312897	128	1 µA
0,314941	10,7217	3,21289E-7	0,315277	129	1 µA
0,317383	10,7461	3,21442E-7	0,317657	130	1 µA
0,319824	10,7705	3,18298E-7	0,32016	131	1 µA
0,322266	10,7949	3,19824E-7	0,322601	132	1 µA

Figure 18 The active current range is now reported in the data grid

NOTE

This new feature only applies to measurements carried out with NOVA 2.1.1 or later. Measurements carried out with earlier versions of NOVA may not display the active current range properly.

NOTE

More information on current range logging is available in *Chapter 11.9.1*.

3.2.3 Event logging

NOVA 2.1.1 now logs events taking place during measurement commands. These events are logged alongside the measured data points and are stored in the data file. Whenever possible, the events are associated to a measured data point. The following events are now logged and stored:

- **Overloads:** these events correspond to situations where a current, voltage or temperature overload was detected during a measurement.
- **Cutoffs:** these events correspond to situations where a cutoff condition is met.
- **Counters:** these events correspond to situations where a counter is activated.

- **User events:** these events correspond to situations where the user changed a measurement property during a measurement or used a flow control option (stop, pause, reverse scan direction) provided by NOVA.

When these events are detected, they are reported in the data grid, as shown in *Figure 19*.

CV staircase

Potential applied (V)	Time (s)	WE(1).Current (A)	WE(1).Potential (V)	Scan	Index	Q+	Q-	Current range	Overload
0,0244141	8,24943	2,68005E-8	0,025705	1	10	1,90342E-6	-1,89706E-6	10 nA	
0,0268555	8,27383	2,84973E-8	0,0281555	1	11	1,90342E-6	-1,89706E-6	10 nA	
0,0292969	8,29823	3,13751E-8	0,0306	1	12	1,90342E-6	-1,89706E-6	10 nA	
0,0317383	8,32263	3,16864E-8	0,0330383	1	13	1,90342E-6	-1,89706E-6	10 nA	
0,0341797	8,34703	3,55682E-8	0,0355286	1	14	1,90342E-6	-1,89706E-6	10 nA	
0,0366211	8,37143	3,83942E-8	0,0378967	1	15	1,90342E-6	-1,89706E-6	10 nA	Current overload
0,0390625	8,39583	4,24744E-8	0,0403625	1	16	1,90342E-6	-1,89706E-6	10 nA	Current overload
0,0415039	8,42023	4,45251E-8	0,0428436	1	17	1,90342E-6	-1,89706E-6	10 nA	Current overload
0,0439453	8,44463	4,38507E-8	0,0452057	1	18	1,90342E-6	-1,89706E-6	10 nA	Current overload
0,0463867	8,46903	4,71863E-8	0,0478027	1	19	1,90342E-6	-1,89706E-6	10 nA	Current overload
0,0488281	8,49343	5,1889E-8	0,0501099	1	20	1,90342E-6	-1,89706E-6	10 nA	Current overload
0,0512695	8,51783	5,44586E-8	0,0526733	1	21	1,90342E-6	-1,89706E-6	10 nA	Current overload
0,0537109	8,54223	5,65277E-8	0,0552063	1	22	1,90342E-6	-1,89706E-6	10 nA	Current overload
0,0561523	8,56663	5,72449E-8	0,0575867	1	23	1,90342E-6	-1,89706E-6	10 nA	Current overload

Figure 19 Events are logged in the data grid

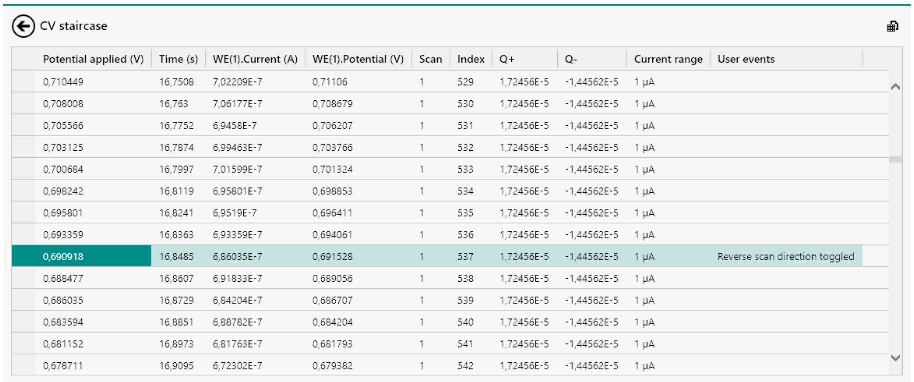
Depending on the type of event, more or less information may be reported in the data grid. In the case of a cutoff condition, information about the signal and value is reported in the data grid *(see Figure 20, page 28)*.

CV staircase

Potential applied (V)	Time (s)	WE(1).Current (A)	WE(1).Potential (V)	Scan	Index	Q+	Q-	Current range	Overload	Cutoffs
0,478516	12,0811	4,81323E-7	0,479523	1	196	1,51173E-6	0	100 nA	Current overload	
0,480957	12,1055	4,83459E-7	0,481964	1	197	1,51173E-6	0	100 nA	Current overload	
0,483398	12,1299	4,85382E-7	0,484436	1	198	1,51173E-6	0	100 nA	Current overload	
0,48584	12,1543	4,89227E-7	0,486816	1	199	1,51173E-6	0	100 nA	Current overload	
0,488281	12,1787	4,91089E-7	0,489258	1	200	1,51173E-6	0	100 nA	Current overload	
0,490723	12,2031	4,92767E-7	0,49176	1	201	1,51173E-6	0	100 nA	Current overload	
0,493164	12,2275	4,93378E-7	0,49408	1	202	1,51173E-6	0	100 nA	Current overload	
0,495605	12,2519	4,9762E-7	0,496552	1	203	1,51173E-6	0	100 nA	Current overload	
0,498047	12,2763	4,99329E-7	0,499115	1	204	1,51173E-6	0	100 nA	Current overload	
0,500488	12,3007	5,03754E-7	0,501495	1	205	1,51173E-6	0	100 nA	Current overload	WE(1).Current > 5E-07 A
0,50293	12,3251	5,05707E-7	0,503937	1	206	1,51173E-6	0	100 nA	Current overload	WE(1).Current > 5E-07 A
0,505371	12,3495	5,05615E-7	0,506256	1	207	1,51173E-6	0	100 nA	Current overload	WE(1).Current > 5E-07 A
0,507813	12,3739	5,10681E-7	0,508759	1	208	1,51173E-6	0	100 nA	Current overload	WE(1).Current > 5E-07 A
0,510254	12,3983	5,12939E-7	0,511047	1	209	1,51173E-6	0	100 nA	Current overload	WE(1).Current > 5E-07 A

Figure 20 The details of the cutoff condition are reported in the data grid

The same applies to user intervention, where the action performed by the user is reported in the grid *(see Figure 21, page 29)*.


CV staircase

Potential applied (V)	Time (s)	WE(1).Current (A)	WE(1).Potential (V)	Scan	Index	Q+	Q-	Current range	User events
0,710449	16,7508	7,02209E-7	0,71106	1	529	1,72456E-5	-1,44562E-5	1 µA	
0,708008	16,763	7,06177E-7	0,708679	1	530	1,72456E-5	-1,44562E-5	1 µA	
0,705566	16,7752	6,9458E-7	0,706207	1	531	1,72456E-5	-1,44562E-5	1 µA	
0,703125	16,7874	6,99463E-7	0,703766	1	532	1,72456E-5	-1,44562E-5	1 µA	
0,700684	16,7997	7,01599E-7	0,701324	1	533	1,72456E-5	-1,44562E-5	1 µA	
0,698242	16,8119	6,95801E-7	0,698853	1	534	1,72456E-5	-1,44562E-5	1 µA	
0,695801	16,8241	6,9519E-7	0,696411	1	535	1,72456E-5	-1,44562E-5	1 µA	
0,693359	16,8363	6,93359E-7	0,694061	1	536	1,72456E-5	-1,44562E-5	1 µA	
0,690918	16,8485	6,86035E-7	0,691528	1	537	1,72456E-5	-1,44562E-5	1 µA	Reverse scan direction toggled
0,688477	16,8607	6,91833E-7	0,689056	1	538	1,72456E-5	-1,44562E-5	1 µA	
0,686035	16,8729	6,84204E-7	0,686707	1	539	1,72456E-5	-1,44562E-5	1 µA	
0,683594	16,8851	6,88782E-7	0,684204	1	540	1,72456E-5	-1,44562E-5	1 µA	
0,681152	16,8973	6,81763E-7	0,681793	1	541	1,72456E-5	-1,44562E-5	1 µA	
0,678711	16,9095	6,72302E-7	0,679382	1	542	1,72456E-5	-1,44562E-5	1 µA	

Figure 21 User events are reported in the data grid

NOTE

This new feature only applies to measurements carried out with NOVA 2.1.1 or later. Measurements carried out with earlier versions of NOVA may not display the recorded events properly.

NOTE

More information on event logging is available in *Chapter 11.9.2*.

Furthermore, NOVA now provides indications whenever the measurement conditions can be improved, by highlighting the affected command in blue in the procedure editor *(see Figure 22, page 29)*.

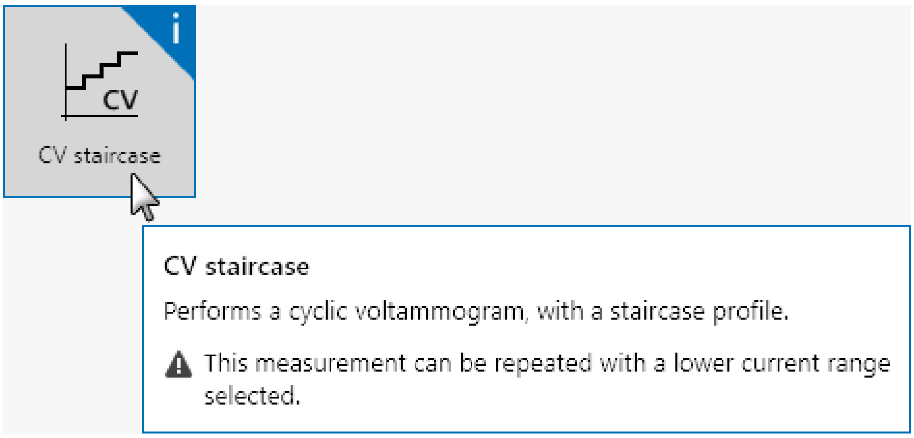


Figure 22 Commands are highlighted in blue when the measurement conditions can be improved

3.2.4 Export options for Spectrophotometer control panel

NOVA 2.1.1 adds the possibility to export data measured in the **Spectrophotometer** control panel. The data can be exported to ASCII or Excel using the button located in the top right corner of the control panel *(see Figure 23, page 30)*.

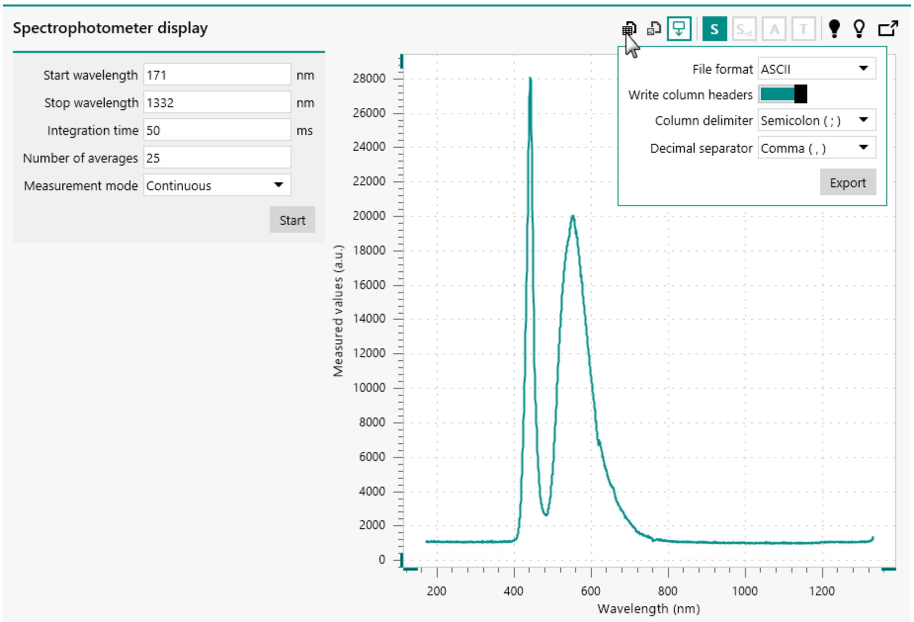


Figure 23 The measured data can be exported to ASCII or Excel

It is also possible to export the chart displayed in the **Spectrophotometer** control panel using the button, as shown in *Figure 24*.

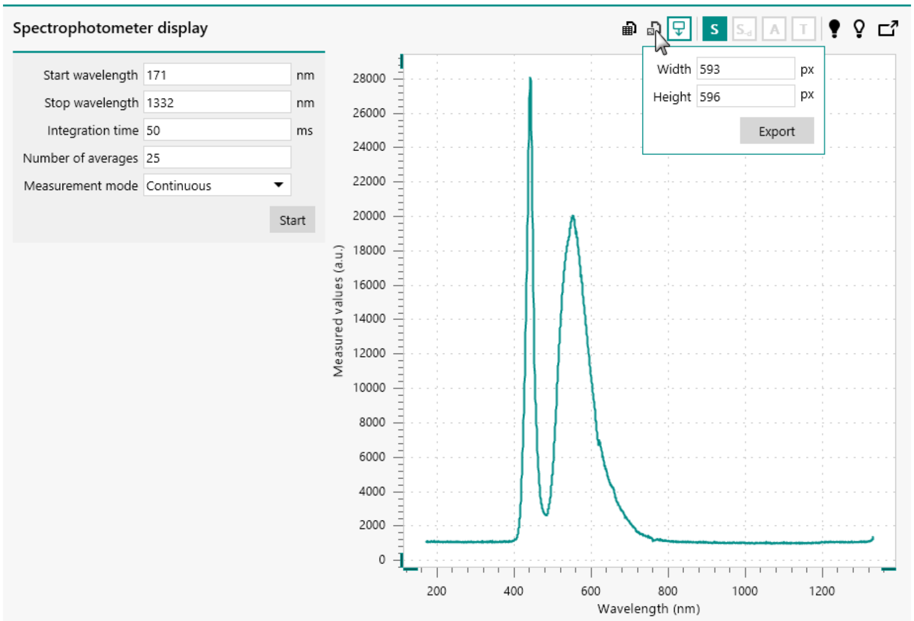


Figure 24 The chart can also be exported

NOTE

More information on the manual control of the Autolab and Avantes spectrophotometers can be found in *Chapter 5.4*.

3.2.5 Export options for Spectrophotometer control panel

NOVA 2.1.1 adds the possibility to toggle the *Step through data* option on or off in the Spectrophotometer control panel using the button in the top right corner of the control panel *(see Figure 25, page 31)*.

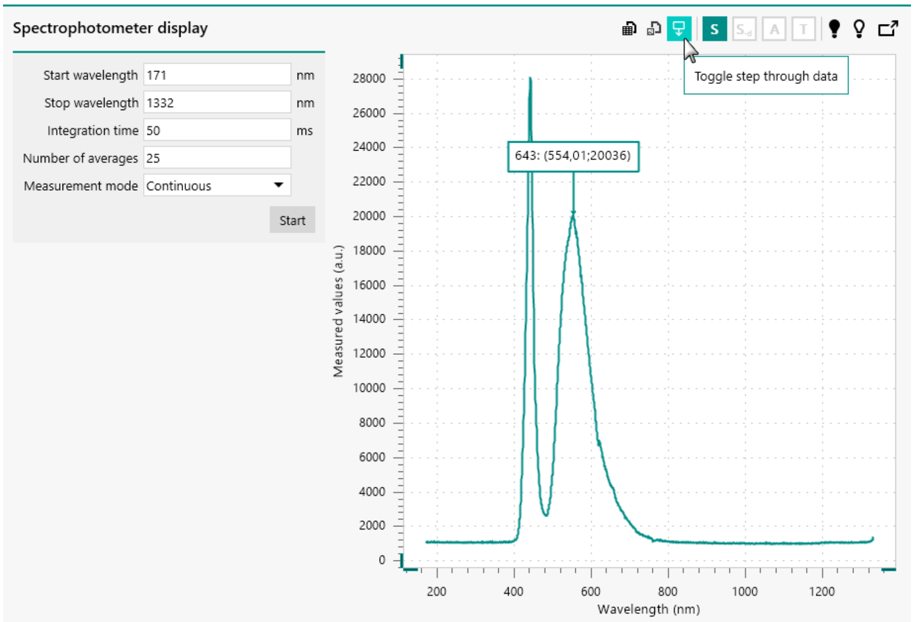


Figure 25 The Step through data option can be used in the Spectrophotometer control panel

When the *Step through data* mode is on, an additional indicator is added to the plot, showing the X and Y coordinates of the point indicated by the arrow. The indicator can be relocated anywhere in the plot area.

NOTE

More information on the manual control of the Autolab and Avantes spectrophotometers can be found in *Chapter 5.4*.

3.2.6 Spectroelectrochemistry procedure

A procedure for spectroelectrochemical measurements in combination with support spectrophotometers has been added to the **Default procedures**. This procedure is based on a synchronized measurement, using a **LSV staircase** command. The procedure is shown in *Figure 26*.

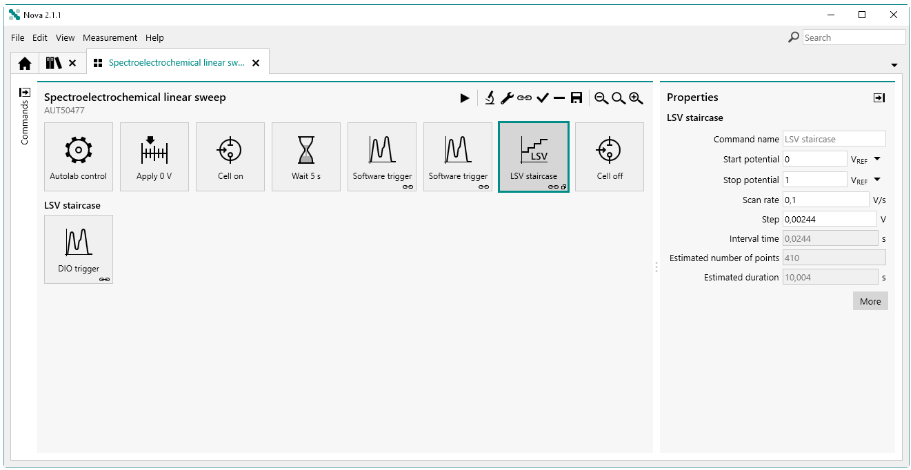


Figure 26 The default procedure for spectroelectrochemical measurements

This procedure includes a counter that is used to trigger a spectroscopy measurement every points. The **Spectroscopy** command stacked on the **LSV staircase** command uses the data from the two preceding **Spectroscopy** commands to calculate the absorbance and transmittance automatically.

CAUTION

This procedure requires an Autolab spectrophotometer or a supported Avantes spectrophotometer.

NOTE

More information on this procedure can be found in *Chapter 8*.

3.3 Version 2.1 release

Version 2.1 adds the following functionality:

1. Application-wide search function for Procedures, Data and Schedules *(see Chapter 3.3.1, page 33)*.
2. Check cell tool *(see Chapter 3.3.2, page 35)*.
3. Cell off after current interrupt *(see Chapter 3.3.3, page 35)*.
4. Manual control for Autolab and Avantes spectrophotometers *(see Chapter 3.3.4, page 36)*.
5. New command and command options for spectroelectrochemical applications *(see Chapter 3.3.5, page 37)*.
6. Repeat number added to **Repeat** command *(see Chapter 3.3.6, page 39)*.
7. Custom names for commands *(see Chapter 3.3.7, page 40)*.
8. Zoom function for procedure editor and schedule editor *(see Chapter 3.3.8, page 41)*.
9. New Electrochemical Frequency Modulation (EFM) measurement command available *(see Chapter 3.3.9, page 42)*.
10. Corrosion rate analysis command expanded with Linear polarization analysis mode *(see Chapter 3.3.10, page 43)*.
11. Improved **Plot** frame controls *(see Chapter 3.3.11, page 44)*.
12. All device drivers are now included in the NOVA installer *(see Chapter 3.3.12, page 46)*

3.3.1 Search function

NOVA now provides the possibility to search for Procedures, Data or Schedules. A dedicated input field is located in the top right corner of the application *(see Figure 27, page 34)*.

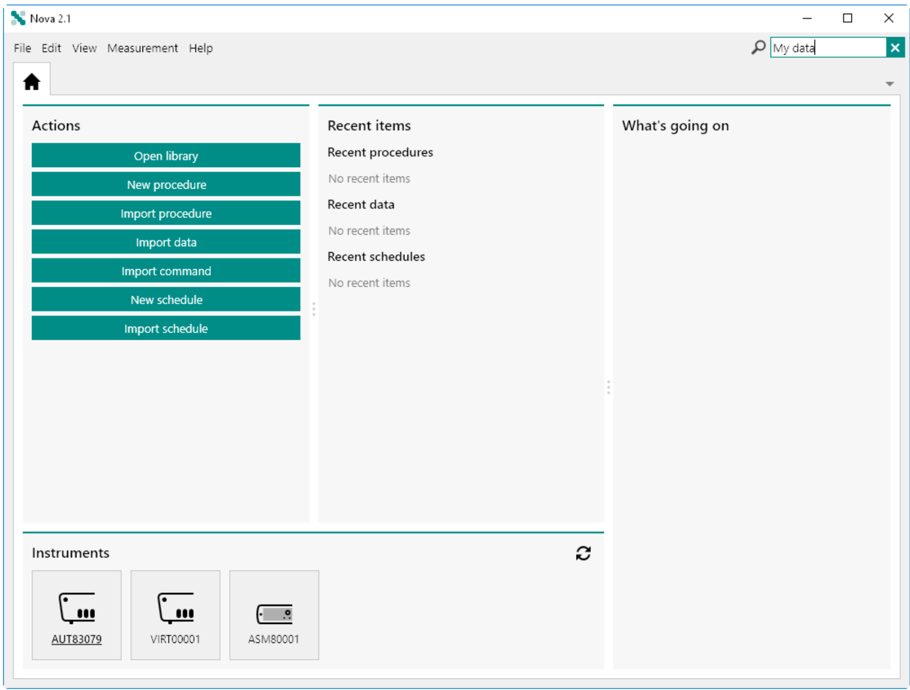


Figure 27 A search box is provided in the top right corner

The search function can be used to specify a string, with or without wild-cards. When triggered, the search function will look for all Procedures, Data and Schedule items in all the *Locations* specified in the **Library**, except the Default procedures.

The results of the search will be reported in a dedicated tab, grouped by item type. The table controls used to display the results are the same as those used by the **Library**. The results can therefore be sorted or filtered as required.

NOTE

The search function will look for all items that match the specified search string in the Name or Remarks.

NOTE

More information on the search function can be found in *Chapter 6.18*.

3.3.2 Check cell

The Check cell tool is now available from the instrument control panel. This tool can be used to check the electrode connections and the noise level by performing five consecutive current or potential measurements and determining the average value and standard deviation of each measurement *(see Figure 28, page 35)*.

Figure 28 The Check cell tool can now be used to check the noise level

The tool can therefore be used to assess the instrument noise pickup and optimize the measurement conditions.

NOTE

More information on the **Check cell** tool can be found in *Chapter 5.2.2.4*.

3.3.3 Current interrupt

The current interrupt tool has been modified and now allows the possibility to switch the cell off at the end of the measurement *(see Figure 29, page 36)*.

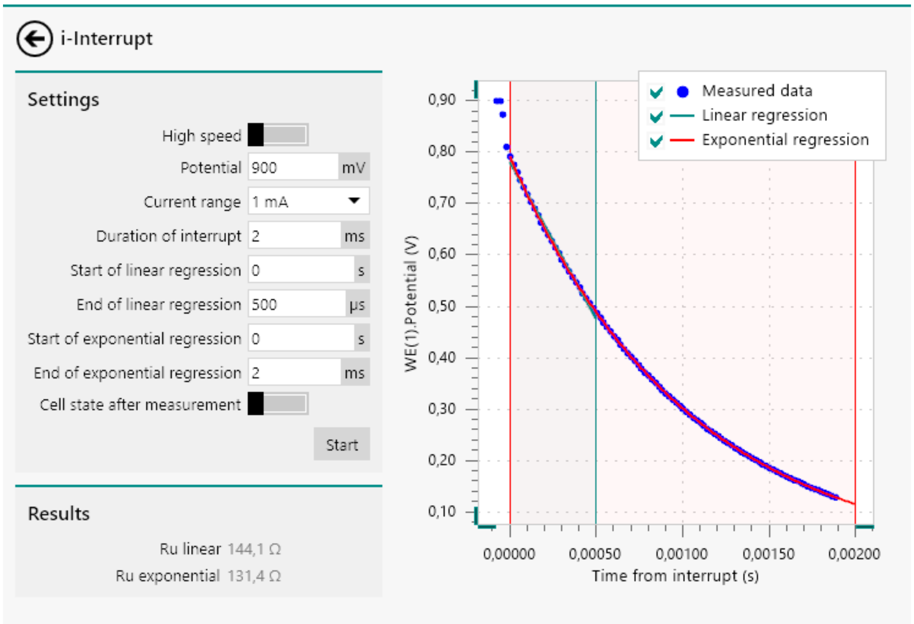


Figure 29 The current interrupt tool now provides the possibility to set the cell end state

The *Cell state after measurement* toggle, located in the **Settings** panel, can be used to specify the state of the cell at the end of the measurement. This toggle in off by default.

NOTE

More information on the current interrupt tool can be found in *Chapter 5.2.2.2*.

3.3.4 Spectrophotometer manual control

NOVA now provides a complete manual control interface for Autolab and Avantes spectrometers. This interface can be used to setup the hardware configuration of the connected spectrophotometer and manually control the spectrophotometer *(see Figure 30, page 37)*.

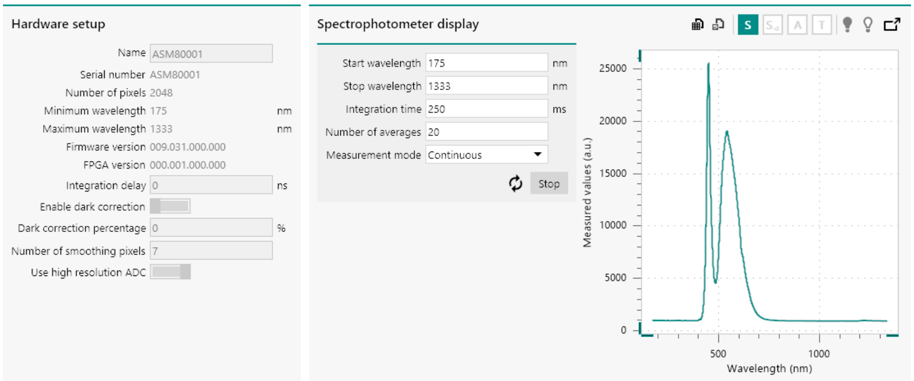


Figure 30 Autolab and Avantes spectrophotometers can be manually controlled

Using this interface it is possible to acquire spectra using the specified properties. It is also possible to save measured spectra as dark and reference (blank) spectra and convert the measured data to absorbance, transmittance or reflectance.

NOTE

More information on the manual control of the Autolab and Avantes spectrophotometers can be found in *Chapter 5.4*.

3.3.5 Spectroelectrochemical measurements

New measurement command and command options have been added to NOVA in order to facilitate spectroelectrochemical measurements. The **Avantes** command is now replaced with the **Spectroscopy** command, which can be used to control Autolab (and Avantes) spectrophotometers.

It is no longer necessary to initialize and close this type of device in a procedure and the new **Spectroscopy** command now supports a software acquisition mode, which can be used at any time without triggers *(see Figure 31, page 38)*.

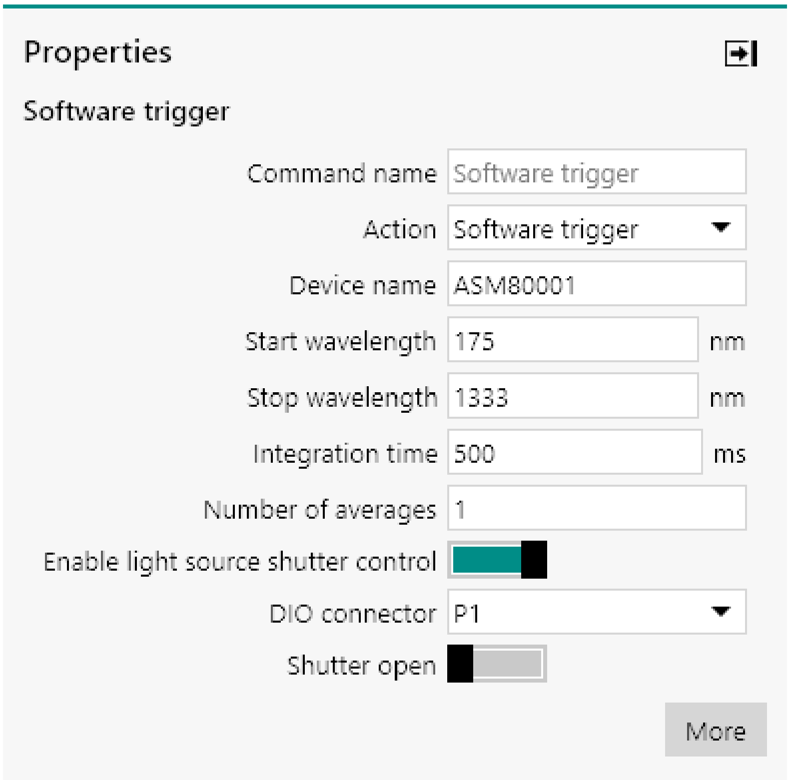


Figure 31 Software and hardware control is now possible to Autolab and Avantes spectrophotometers

New measurement options are available for all measurement commands that support them. These option can be used to control the light source shutter position or to acquire a spectrum on the spectrophotometer connected to the DIO port *(see Figure 32, page 39)*.

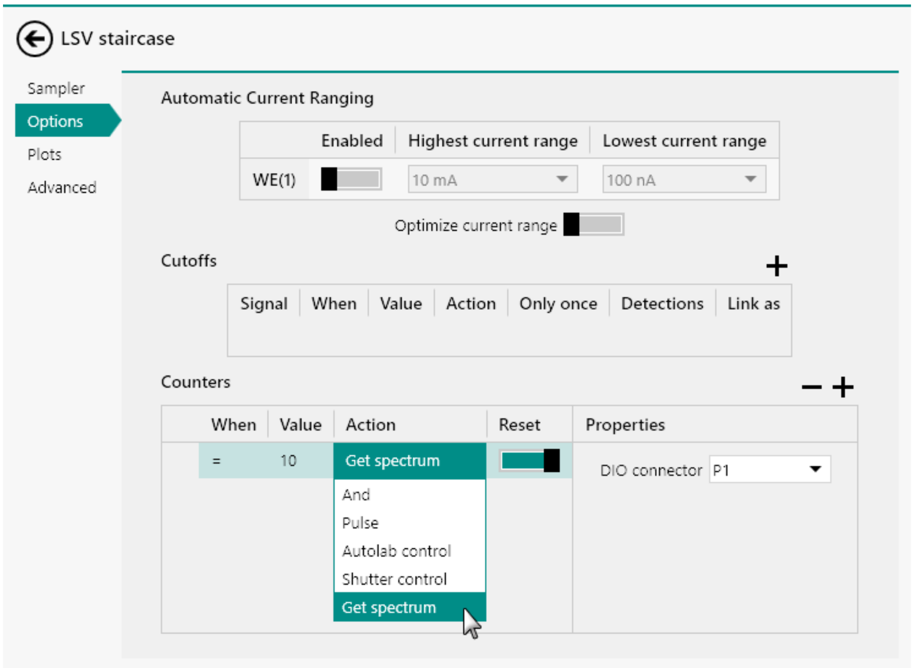


Figure 32 New control options are available for measurement commands

NOTE

More information on the **Spectroscopy** command and the spectrophotometer control options are available in *Chapter 7.11.1* and *Chapter 9*, respectively.

3.3.6 Repeat number in Repeat command

The **Repeat** command now provides a new signal, Repetition number, that can be used in combination with other commands. This new signal is a single value that is incremented at the beginning of each repetition *(see Figure 33, page 40)*.

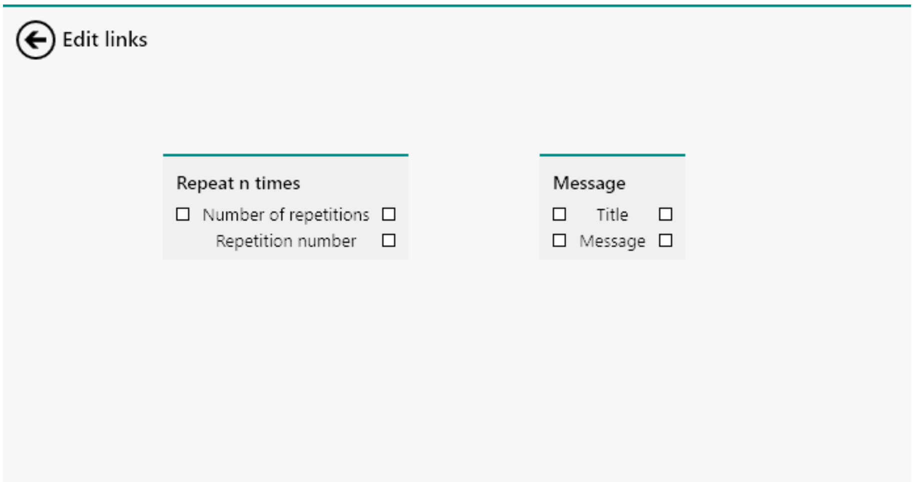


Figure 33 The Repetition number is now available in the Repeat command

NOTE

More information on the Repeat command can be found in *Chapter 7.1.3*.

3.3.7 Custom command name

For improved readability in the procedure editor, it is now possible to specify a name for all commands in a procedure. Providing a custom name will overrule the default name of the command *(see Figure 34, page 41)*.

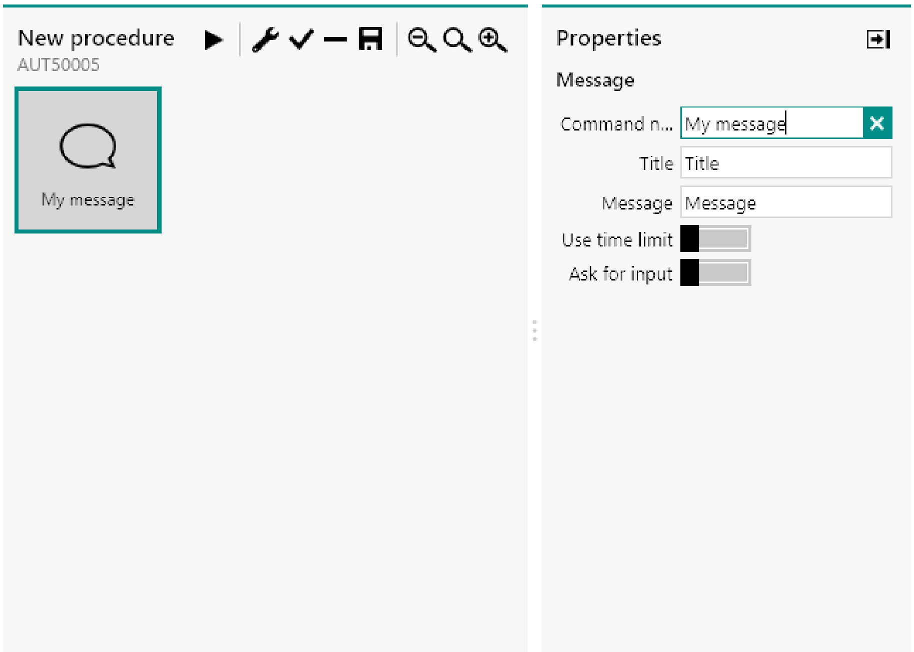


Figure 34 Custom names can now be given to all commands

NOTE

When a custom name is provided for a command, the content of this command is no longer updated during a measurement, if applicable.

3.3.8 Zoom function

The procedure editor and the schedule editor now offer the possibility to zoom in or out at any time to increase or decrease the size of the items shown on screen. The controls for this new zoom function are located in the top right corner of the editor frame *(see Figure 35, page 41)*.

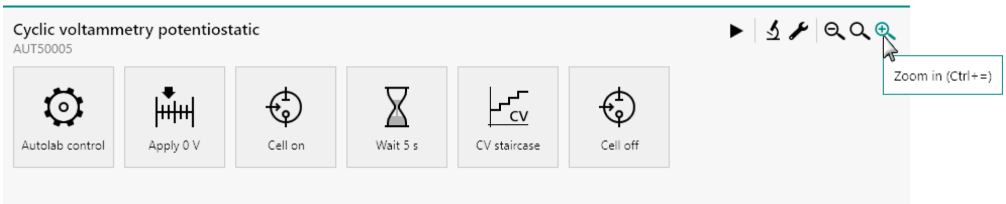


Figure 35 Zoom controls are now available

Using this function will either scale the size of the items and the text up or down (between 200 % and 50 % of the original size), as shown in *Figure 36*.

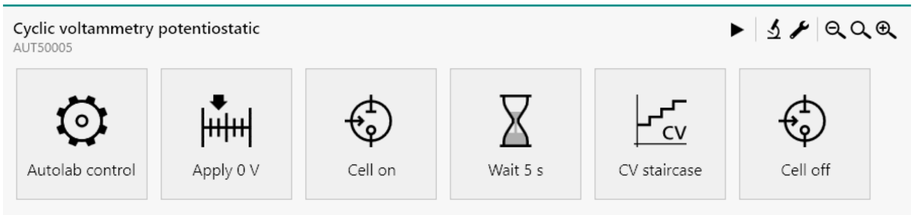


Figure 36 Zooming in on the procedure editor

The following zooming controls are available:

- **Zoom out:** decreases the scaling of the items and text shown on screen. The button or **[CTRL]** + **[-]** keyboard shortcut can be used to do this.
- **Zoom to 100%:** resets the scaling of the items and text shown on screen to the default size. The button or **[F4]** keyboard shortcut can be used to do this.
- **Zoom in:** increases the scaling of the items and text shown on screen. The button or **[CTRL]** + **[=]** keyboard shortcut can be used to do this.

NOTE

More information on the zoom controls of the procedure editor and the schedule editor can be found in *Chapter 10.6* and *Chapter 15.7*, respectively.

3.3.9 Electrochemical Frequency Modulation

This version of NOVA provides support for Electrochemical Frequency Modulation (EFM) measurements. These measurements are based on the application of a small amplitude voltage perturbation and recording of the electrochemical response of the cell. Using the measured data, corrosion rate information can be determined.

EFM measurements use a special two component sinewave modulation. During this type of measurements, the response from the cell at the applied frequency, higher harmonics of these frequencies and intermodulated frequencies are recorded. *Figure 37* shows a typical measurement.

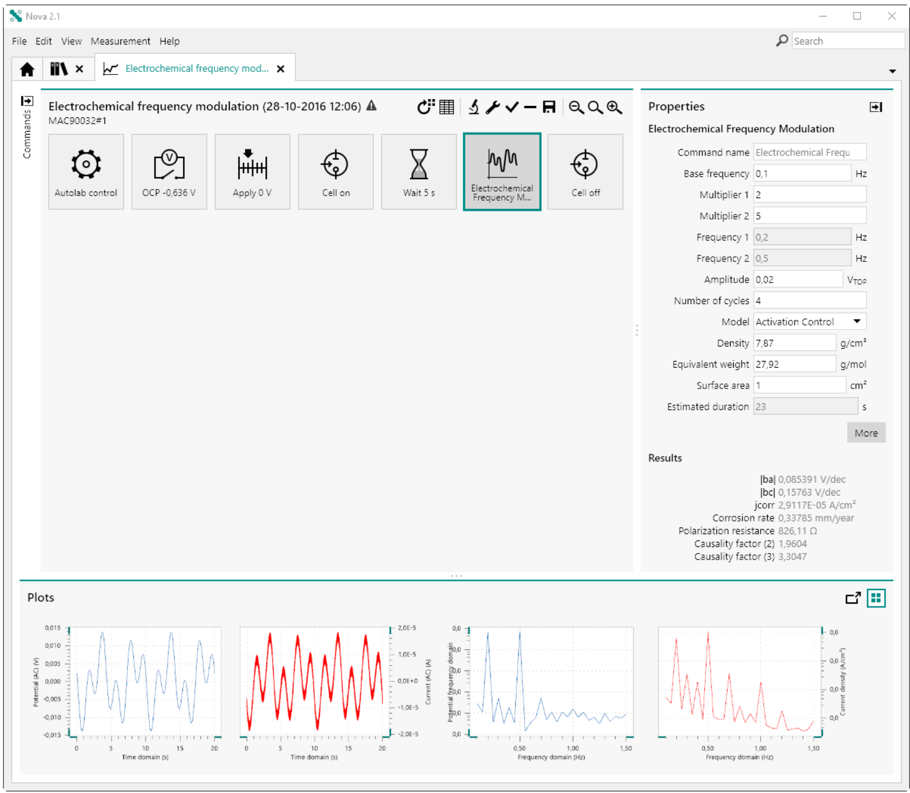


Figure 37 Example of an EFM measurement

CAUTION

Electrochemical Frequency Modulation measurements require a **FRA32M** module.

NOTE

More information on Electrochemical Frequency Modulation command can be found in *Chapter 7.6.4*.

3.3.10 Corrosion rate analysis

The **Corrosion rate analysis** command has been complemented with a new mode: **Polarization Resistance**. This analysis method is based on the **ASTM G59** standard and it uses the *Stern-Geary* equation to determine the corrosion current and the corrosion rate *(see Figure 38, page 44)*.

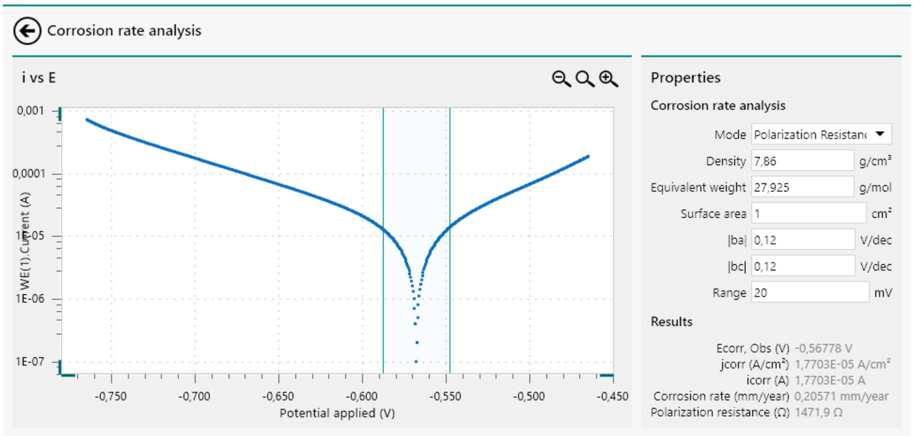


Figure 38 The Linear polarization method has been added to the Corrosion rate analysis command

Provided that the analysis is carried out in a low overpotential range with respect to the corrosion potential, the Linear polarization analysis method can provide a direct estimation of the corrosion current and corrosion rate, using user-defined Tafel slopes.

NOTE

More information on the **Corrosion rate analysis** command can be found in *Chapter 7.8.14*.

3.3.11 New Plots frame controls

The **Plots** frame now provides new controls that can be used to disable plots *(see Figure 39, page 45)*.

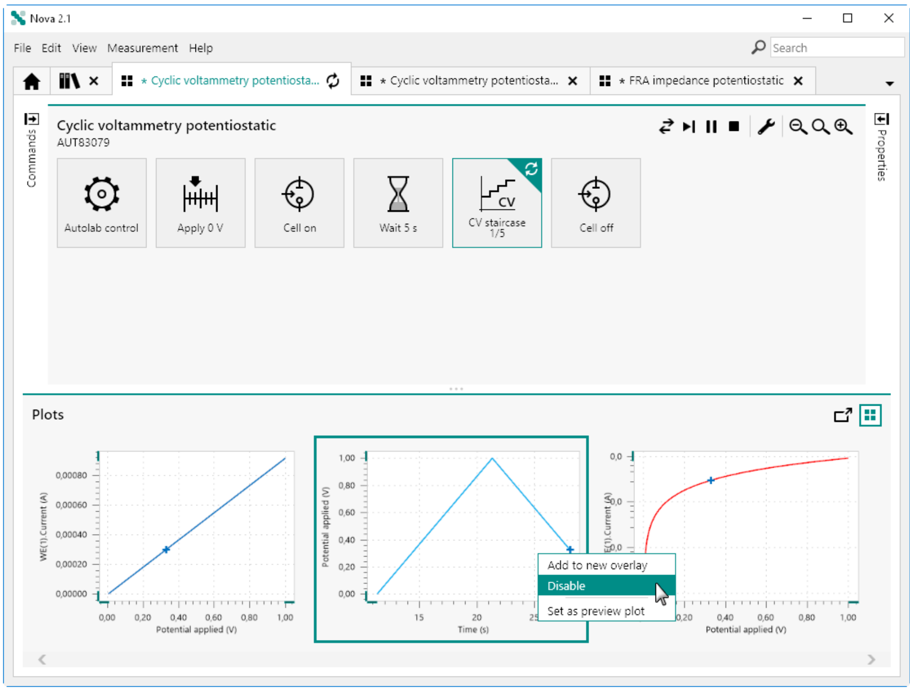


Figure 39 Disabling plots in the Plots frame

NOTE

Disabling plots can be done at any time.

It is also now possible to relocate the plot order or overlay plots by dragging the plots in the frame *(see Figure 40, page 46)*.

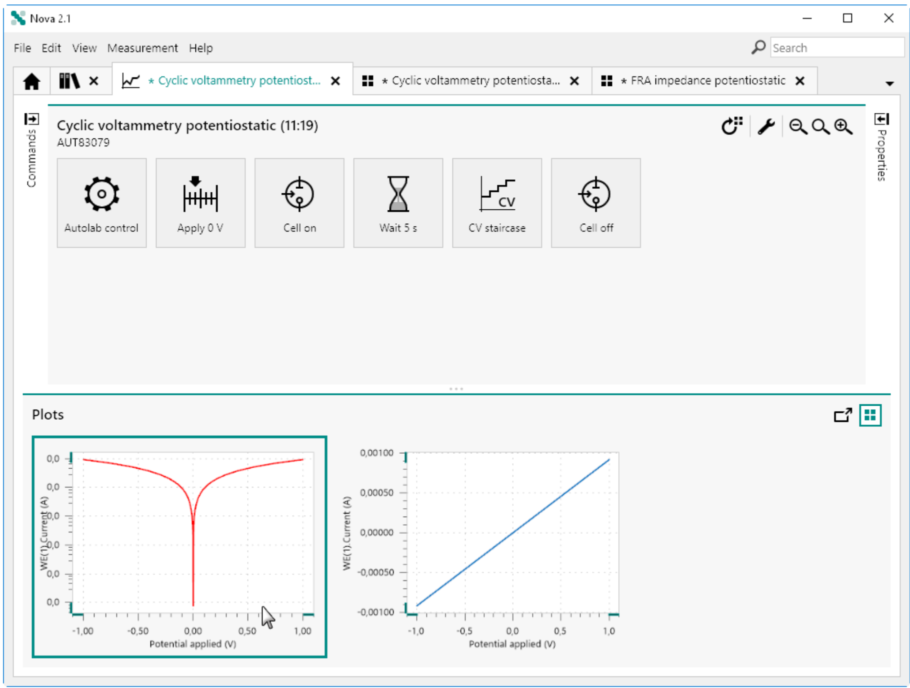


Figure 40 Rearranging the plot order

NOTE

It is not possible to relocate plot during a measurement.

NOTE

More information on the disabling of plots and the relocation of plots can be found in *Chapter 11.5.5* and *Chapter 11.8.8*, respectively.

3.3.12 Device drivers installation

The installation package of NOVA 2.1 now installs all required device drivers during the installation process, as described in *Chapter 1.3*.

The following drivers are installed:

- **Autolab device drivers:** required for using the Autolab potentiostat/ galvanostat.
- **Metrohm device driver:** required for using any supported Metrohm liquid handling instrument.
- **Spectrophotometer device driver:** required for using any supported Autolab (or Avantes) spectrophotometer.

NOTE

If needed, the **Driver manager** application can be used to change the driver used to control the Autolab potentiostat/galvanostat as described in *Chapter 1.6*.

3.4 Version 2.0.2 release

Version 2.0.2 adds the following functionality:

1. Managed schedules *(see Chapter 3.4.1, page 47)*.
2. Plot color picker *(see Chapter 3.4.2, page 48)*.
3. Data handling tools shortcut button *(see Chapter 3.4.3, page 50)*.
4. Filters in **Library** *(see Chapter 3.4.4, page 51)*.
5. Detailed **Sampler** information *(see Chapter 3.4.5, page 52)*.
6. Zoom function for data analysis commands *(see Chapter 3.4.6, page 53)*.
7. Extension to the **Build signal** command *(see Chapter 3.4.7, page 54)*.

3.4.1 Managed schedules

Schedules can now be managed through the **Library** in the same way as procedures and data. The **Library** now provides a default location for **Schedules**, as shown in *Figure 41*, and additional **Locations** can be added if needed.

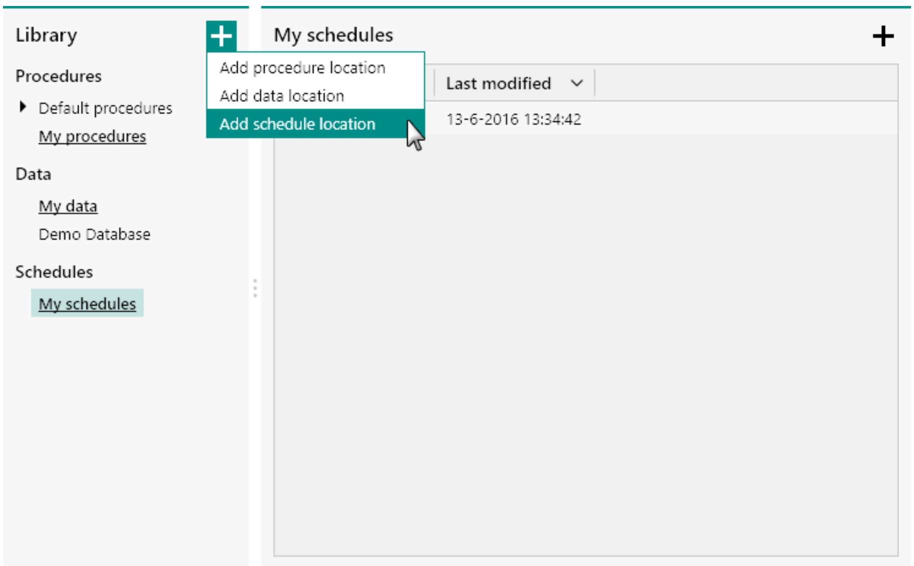


Figure 41 Schedules are now managed through the Library

The most recent **Schedules** are now also listed in the **Recent items** panel on the **Dashboard**, as shown in *Figure 42*.

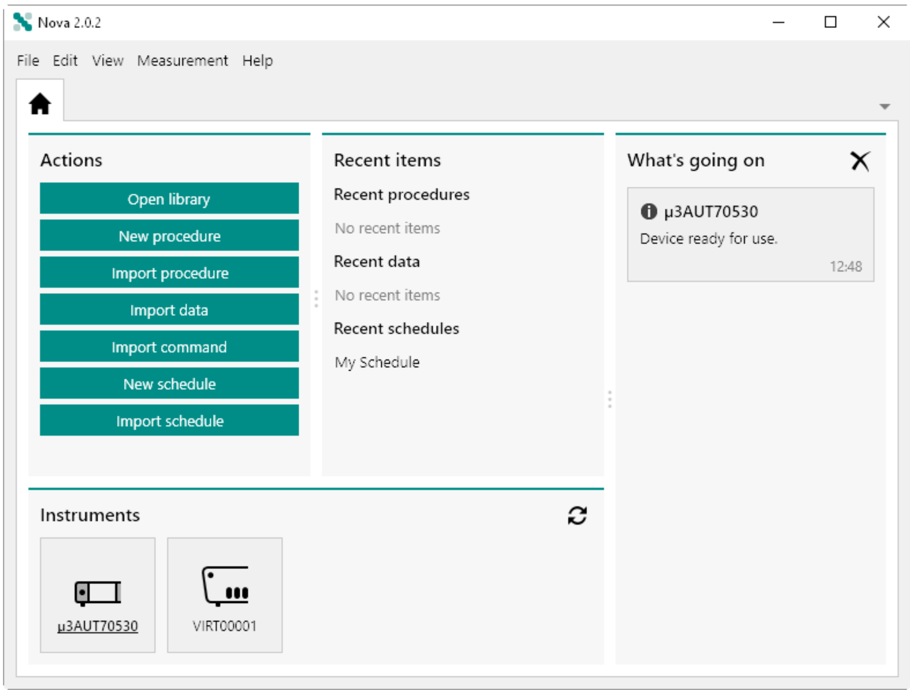


Figure 42 The most recent Schedules are now listed in the Recent items panel

NOTE

More information on the **Library** can be found in *Chapter 6*.

3.4.2 New color picker

To simplify the editing of plots, the existing color picker of NOVA 2.0 has been replaced with a new version shown in *Figure 43*.

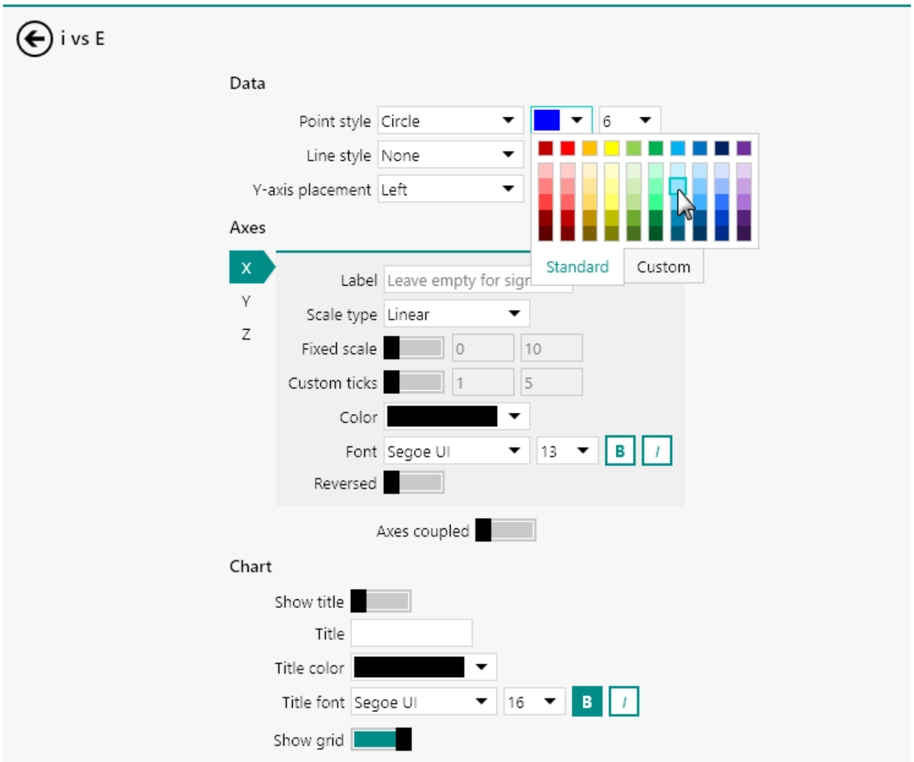


Figure 43   A new color picker is now available

The new color picker provides a list of default colors to choose from. Alternatively, it is possible to define a custom color using the controls provided in the Custom tab *(see Figure 44, page 50)*.

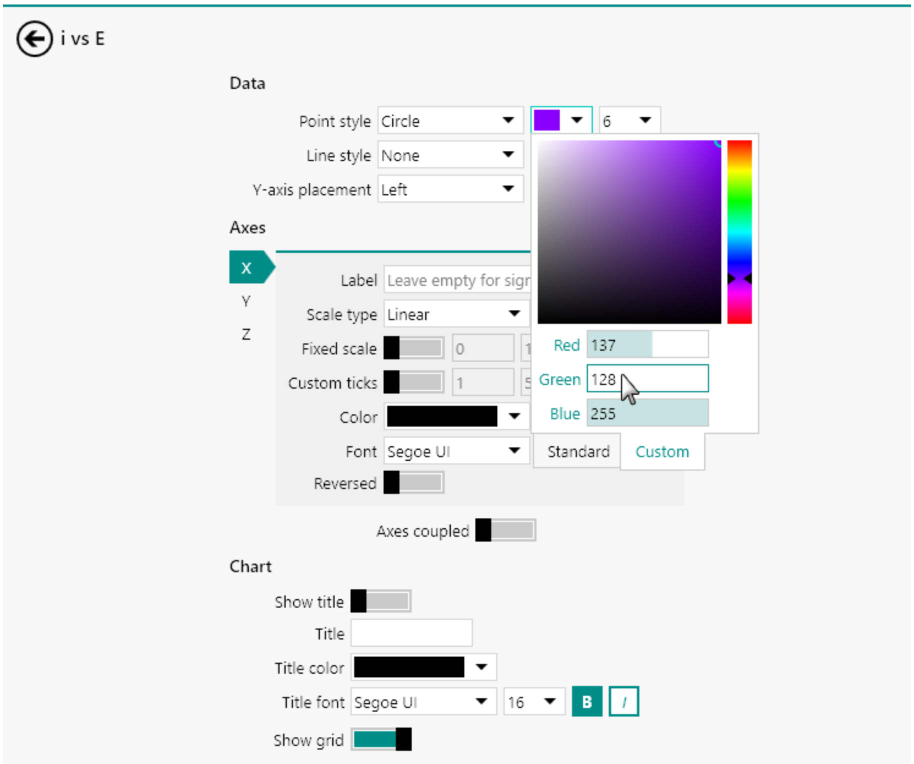


Figure 44 The Custom tab provides additional controls for specifying the color

On the Custom tab, colors can be specified using RGB values or by changing the hue of the selected color or by selecting any available color in the provide RGB color matrix.

3.4.3 Data handling command shortcut button

A shortcut button has been added to this version of NOVA allowing data handling commands to be added to a procedure or data. The 🔧 shortcut button, located in the top right corner of the procedure editor, works in the same way as the data analysis shortcut button already available in the NOVA *(see Figure 45, page 51)*.

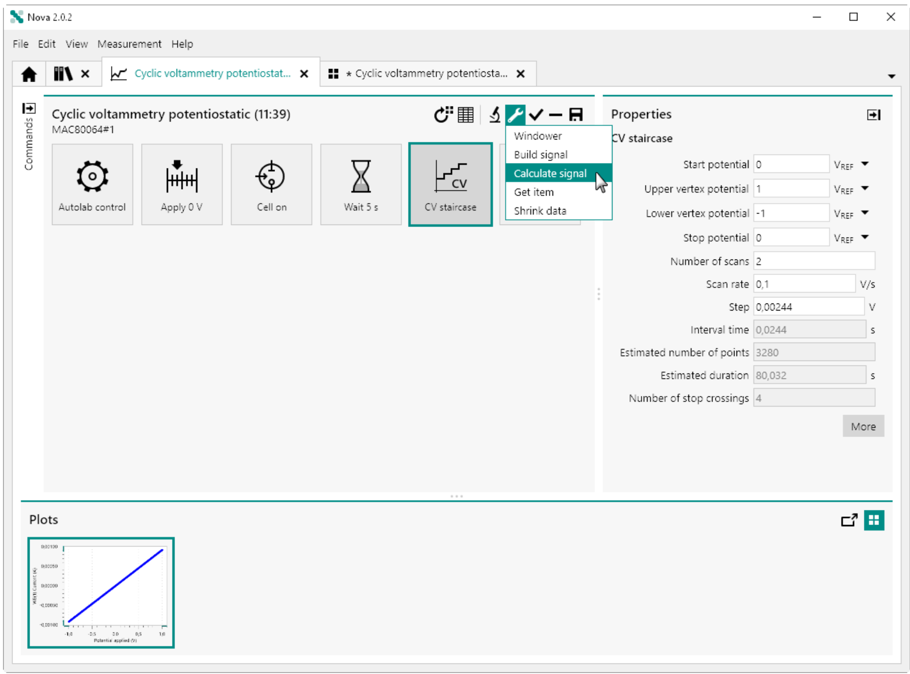


Figure 45 The data handling shortcut button can be used to add data handling commands easily

NOTE

The data handling commands shown in the popout menu depend on the selected command.

NOTE

More information on the use of the data handling shortcut button can be found in *Chapter 13*.

3.4.4 Library filters

To facility data management, the **Library** now provides filtering options that can be used to force the **Library** to display items that fit within the specified filter conditions. *Figure 46* shows an example using two filter conditions, one on the instrument serial number and one on the rating.

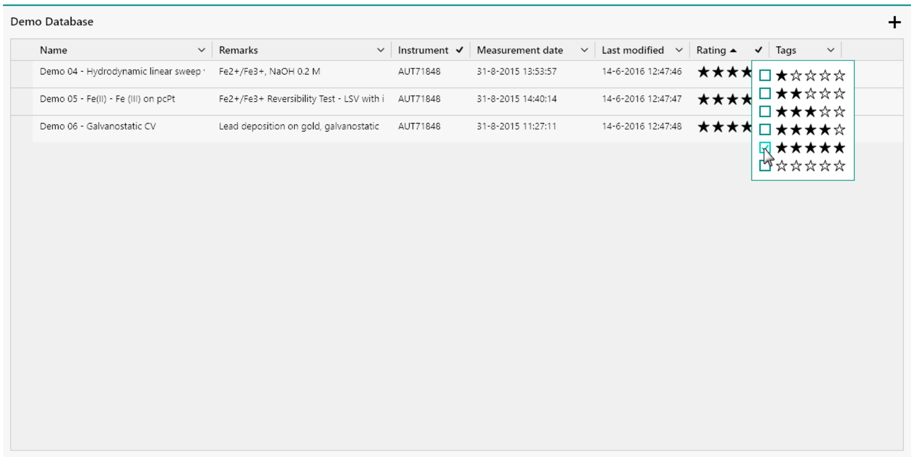


Figure 46 The Library now provides filtering options for better data handling

NOTE

More information on the **Library** filters can be found in *Chapter 6.11*.

3.4.5 Extended Sampler information

To provide more information on how and when the **Sampler** records the signals during any electrochemical measurement, the **Sampler** editor has been extended with a table that provides more details on the sampling conditions. In the **Sampler** editor, shown in *Figure 47*, an additional More button is now available.

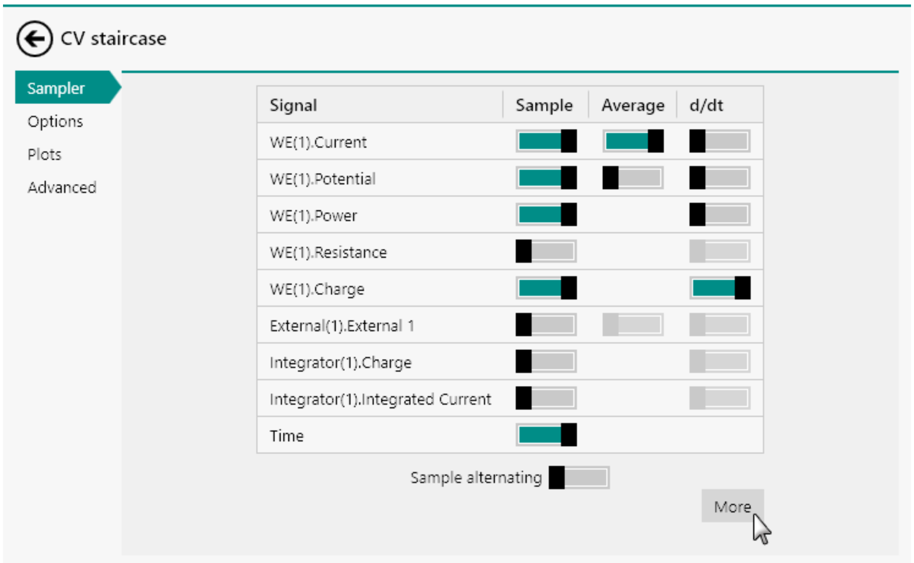
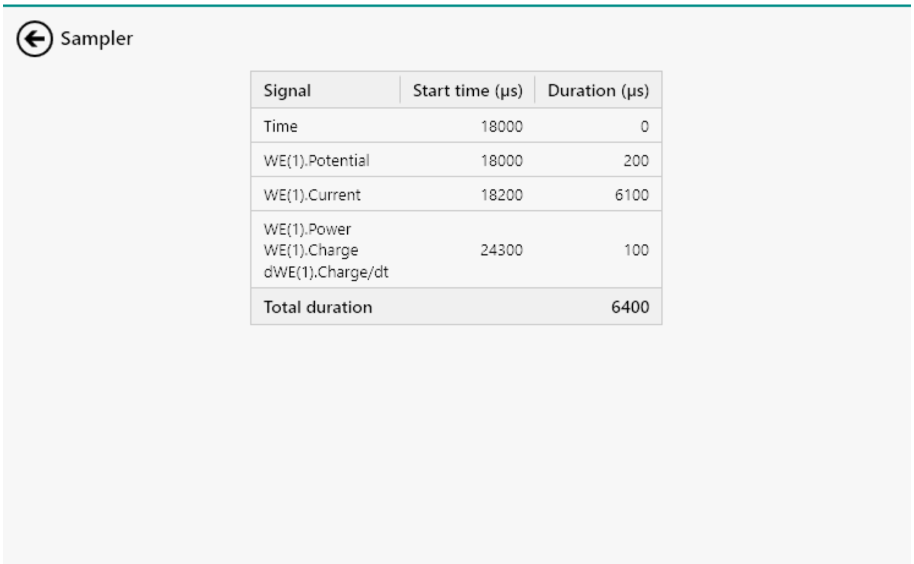


Figure 47 The Sampler editor now provides more information

Clicking the More button brings up a detailed table provide an overview of the sampling timing and duration *(see Figure 48, page 53)*.



Signal	Start time (µs)	Duration (µs)
Time	18000	0
WE(1).Potential	18000	200
WE(1).Current	18200	6100
WE(1).Power WE(1).Charge dWE(1).Charge/dt	24300	100
Total duration		**6400**

Figure 48 The sampling timing and duration is specified in the table

NOTE

More information on the **Sampler** can be found in *(see Chapter 9.1, page 595)*.

3.4.6 Zoom function for data analysis commands

The **Baseline correction** and **Electrochemical circle fit** analysis commands have been complemented with a zoom function that can be used to fine tune the location of the markers used by these commands *(see Figure 49, page 54)*.

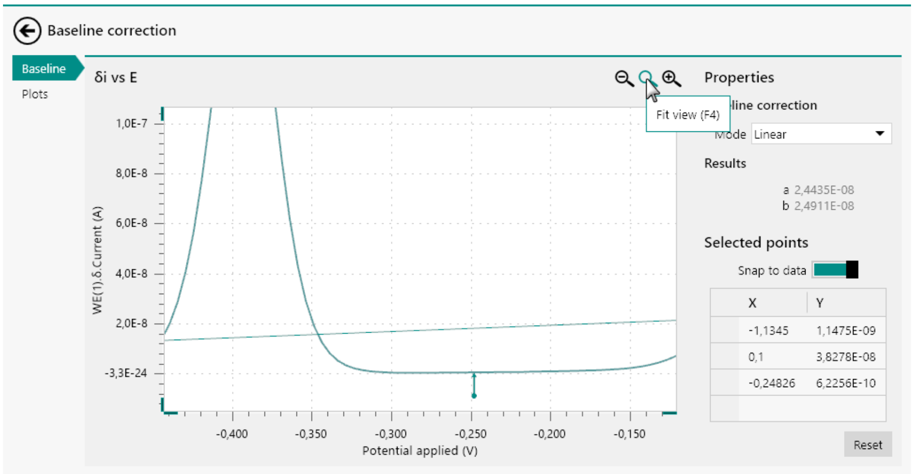


Figure 49 Zooming in and out is now possible for relevant analysis commands

NOTE

More information on this new option can be found in *Chapter 12.7.1* and in *Chapter 12.9.1* for the **Baseline correction** and **Electrochemical circle fit** commands, respectively.

3.4.7 Custom name for Build signal command

It is now possible to define a custom name to the signals generated by the **Build signal** command, as shown in *Figure 50*.

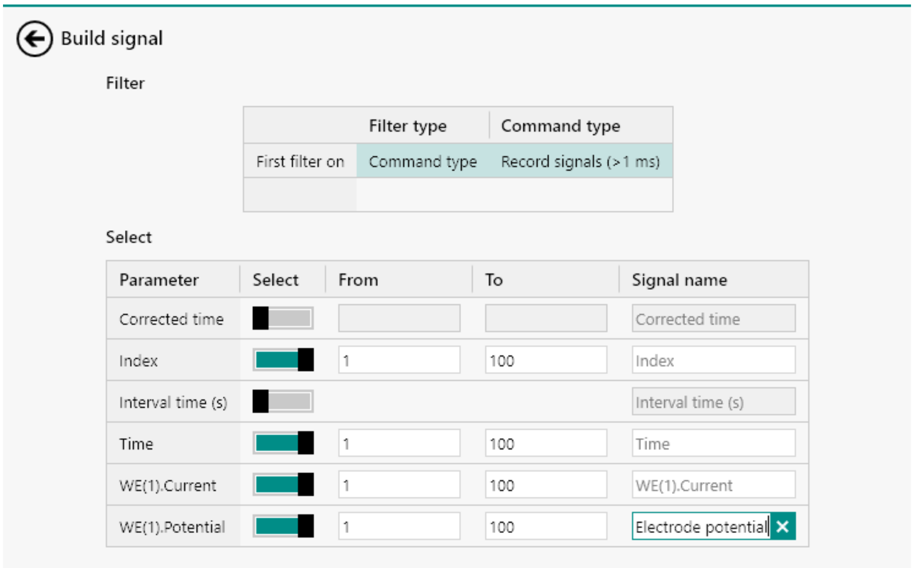


Figure 50 The Build signal command now offers the possibility to provide custom names

By default, the signal name will be the same of the signal selected by the **Build signal** command. This can be overruled by specifying a custom name.

NOTE

More information on the **Build signal** command can be found in *Chapter 7.7.2*.

3.5 Version 2.0.1 release

Version 2.0.1 adds the following functionality:

1. Tagging of procedure and data files *(see Chapter 3.5.1, page 56)*.
2. New plot options *(see Chapter 3.5.2, page 57)*.
3. A new mechanism for importing data from GPES and FRA into NOVA *(see Chapter 3.5.3, page 59)*.
4. The number of recent items shown in the **Recent items** panel can now be edited *(see Chapter 3.5.4, page 59)*.
5. Data files in the **Library** can now be expanded with a preview plot *(see Chapter 3.5.5, page 60)*.
6. Print functionality for plots *(see Chapter 3.5.6, page 61)*.
7. NOVA is now completely region insensitive *(see Chapter 3.5.7, page 63)*.
8. A new mechanism for quickly switching the active electrochemical interface between the PGSTAT and the ECI10M *(see Chapter 3.5.8, page 63)*.
9. A dedicated indicator is now used for measurements using the ECI10M module *(see Chapter 3.5.9, page 64)*.
10. The display settings used in the Library are now non-volatile *(see Chapter 3.5.10, page 66)*.
11. The display settings used in data grids are now non-volatile *(see Chapter 3.5.11, page 66)*.
12. The Estimated duration value is now shown in the **Properties** panel *(see Chapter 3.5.12, page 67)*.
13. An **Interpolate** command is now available in the Analysis - general group of commands *(see Chapter 3.5.13, page 68)*.
14. The **Hydrodynamic analysis** command has been extended with the Koutecký-Levich analysis technique *(see Chapter 3.5.14, page 68)*.

3.5.1 Procedure and data tags

It is now possible to assign tags to procedures and data files. This is a convenient tool for bookkeeping purposes. The controls for tagging data or procedures are provided in **Tags** sub-panel of the **Properties** panel of the procedure editor *(see Figure 51, page 56)*.

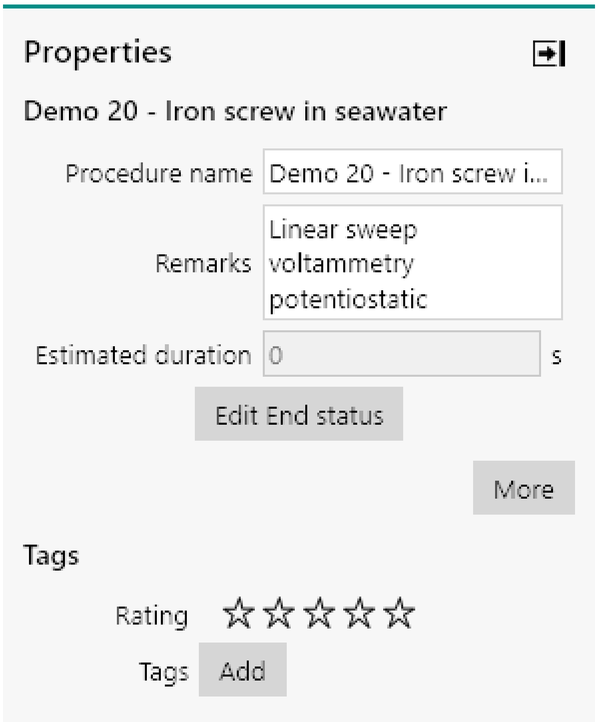


Figure 51 The Tags sub-panel provides controls for tagging data and procedure files

Two types of tags can be assigned to data or procedures:

- **Rating:** a rating based on a stars system can be assigned to each data or procedure file. By default, no stars are assigned to a file, but it is possible to change this at any time.
- **Tags:** text tags can be added to each data or procedure file. By default, no tags are assigned to a file, but it is possible to change this at any time.

NOTE

The rating and tags are updated when the file is saved.

It is also possible to provide a rating and tags directly from the Library *(see Figure 52, page 57)*.

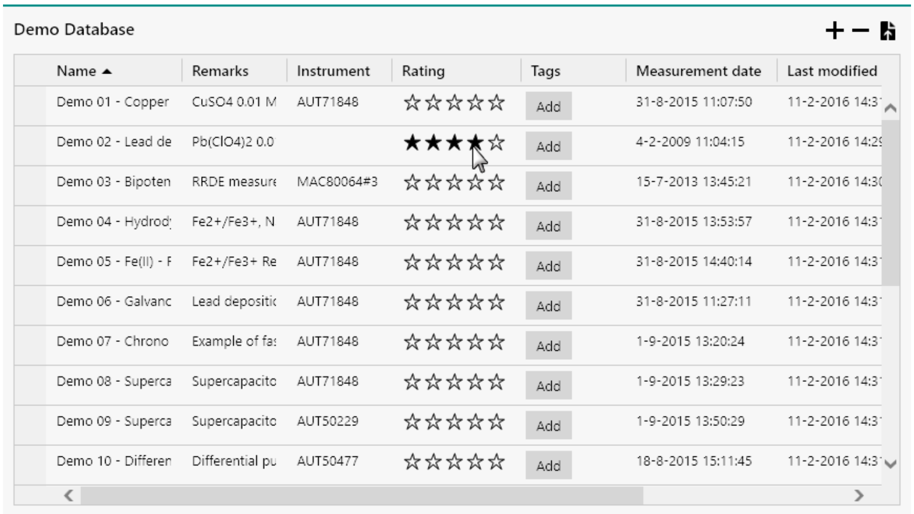


Figure 52 Tags can also be defined in the Library

NOTE

More information on the rating and tagging of procedures and data can be found in *Chapter 6.8*.

3.5.2 New plot options

The plot options have been expanded in order to allow for additional control of the plotting of data *(see Figure 53, page 58)*.

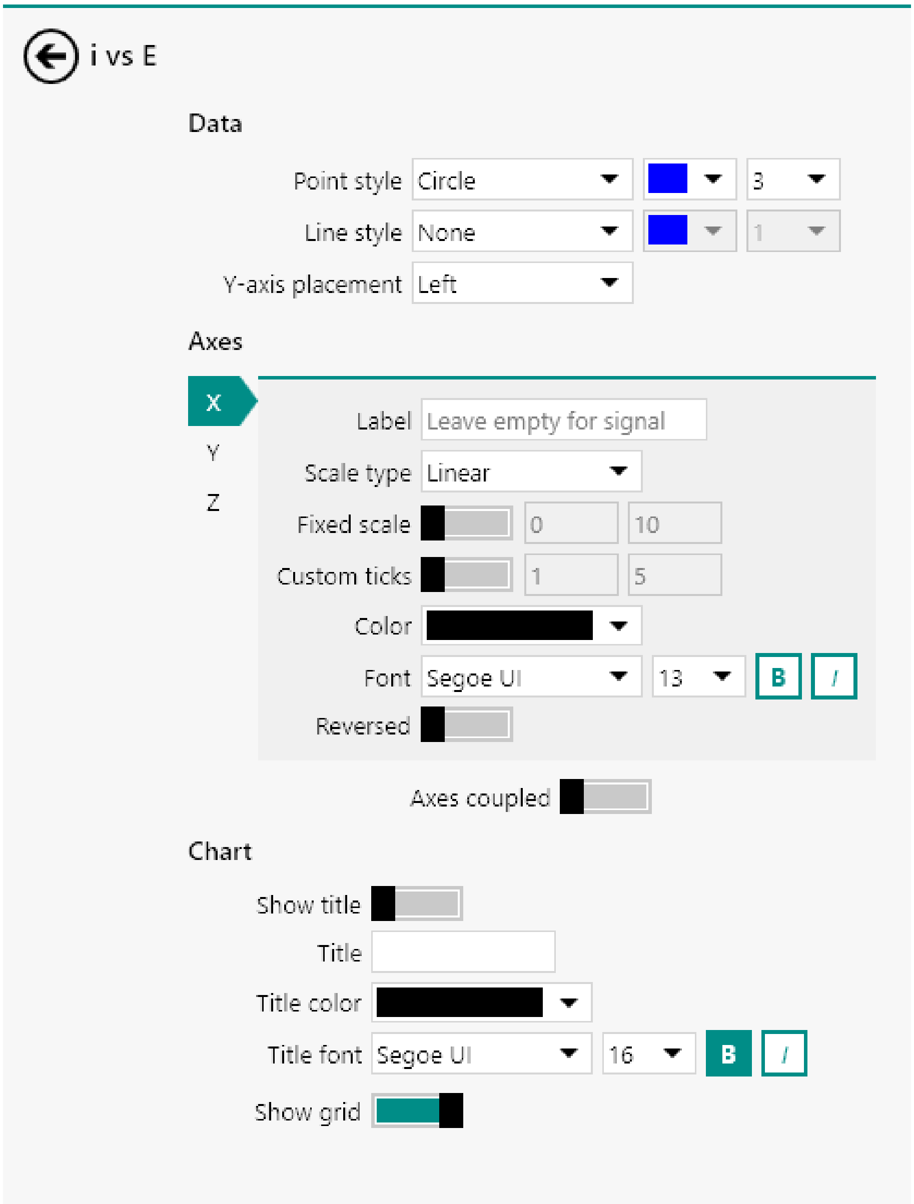


Figure 53 New plot options have been added to the Axes sub-panel

The following options have been added to the **Axes** sub-panel:

- **Fixed scale (on/off):** defines if the axis should be automatically scaled or if a fixed scale should be used. When this property is switched on, it is possible to define a minimum and maximum value for the axis.
- **Custom ticks (on/off):** defines if custom major and minor ticks should be used for the axis. When this property is switched on, it is possible to define the distribution of major and minor ticks.

3.5.3 Import data

It is now possible to import data from the **GPES** and **FRA** software into NOVA using the Import data button in the **Actions** panel. When this button is clicked, the file type can be selected in the Open file dialog window *(see Figure 54, page 59)*.

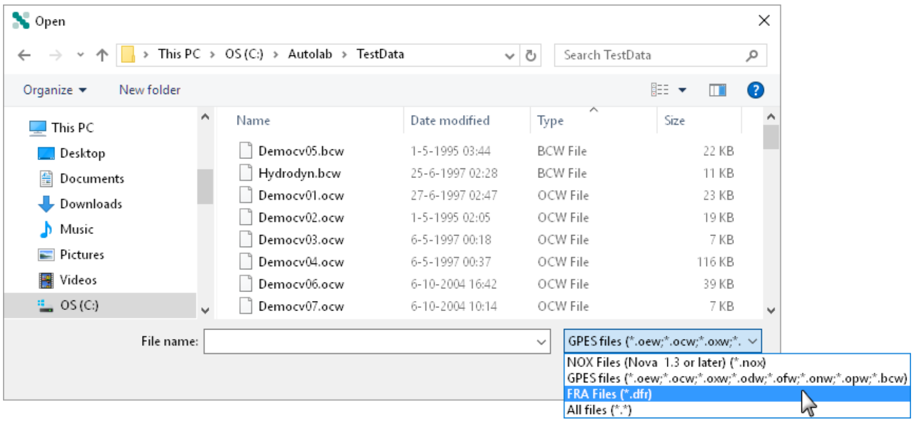


Figure 54 The file type can be specified in the Open file dialog window

The specified GPES or FRA data file will be imported directly into the Library.

This new functionality carries out the following steps after the file is selected by the user:

1. A new procedure is created.
2. The name of the new procedure is changed to the name of the file specified by the user.
3. An **Import data** command is added to the new procedure.
4. The specified file and path are used for the **Import data** command.
5. The procedure is automatically executed and the data is saved.

NOTE

The new Import data functionality works in the same way as the existing functionality provided by the **Import data** command *(see Chapter 7.7.5, page 343)*.

3.5.4 Number of recent items

The number of recent items can now be edited in the NOVA options *(see Figure 55, page 60)*.

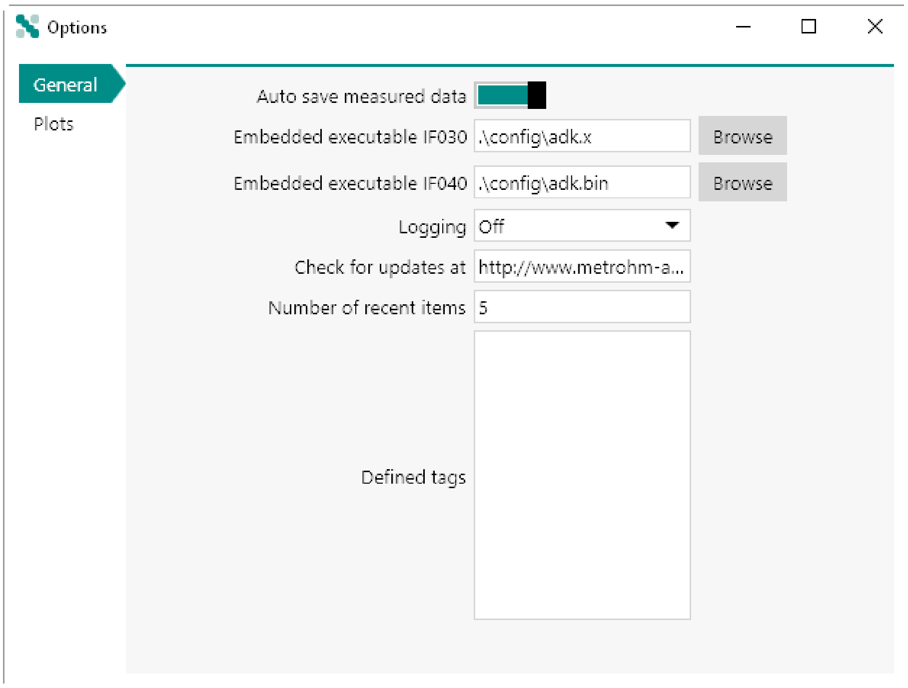


Figure 55 The number of recent items can be edited in the NOVA options

The default number of items is 5 and can be edited at any time.

3.5.5 Plot preview

NOVA now offers the possibility to assign one of the plots of a data file as a preview plot to display in the Library. This provides a quick preview of the data contained in each data file. The preview plot is shown in a tooltip *(see Figure 56, page 61)*.

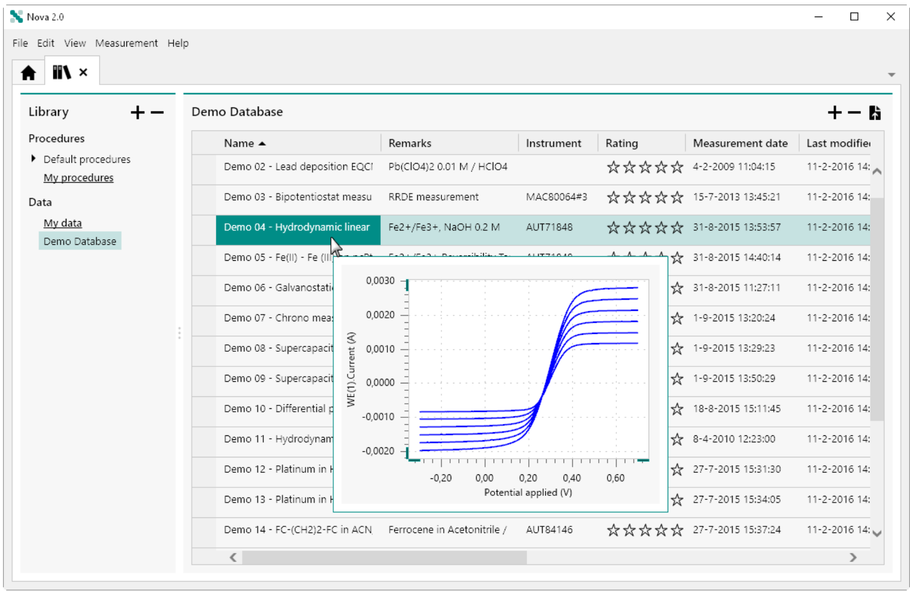


Figure 56 A plot preview is now added to each data item in the Library

NOTE

Data measured with previous versions of NOVA will create a preview plot when changes to the file are saved in the current version.

NOTE

More information on the plot previews can be found in *Chapter 6.9*.

3.5.6 Print plot

It is now possible to print plots using the provided button *(see Figure 57, page 62)*.

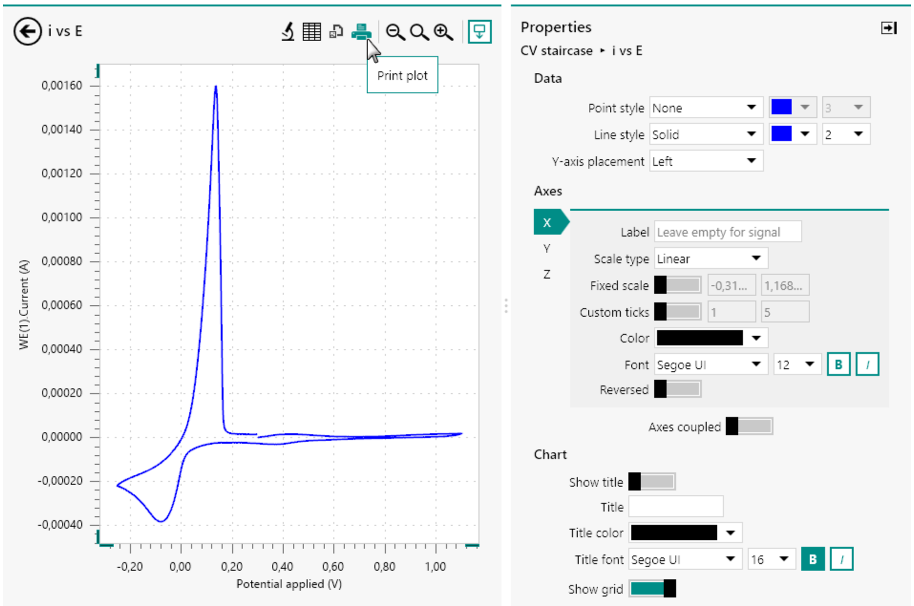


Figure 57 Plots can now be printed

A print preview dialog will be displayed, allowing finetuning of the print output *(see Figure 58, page 62)*.

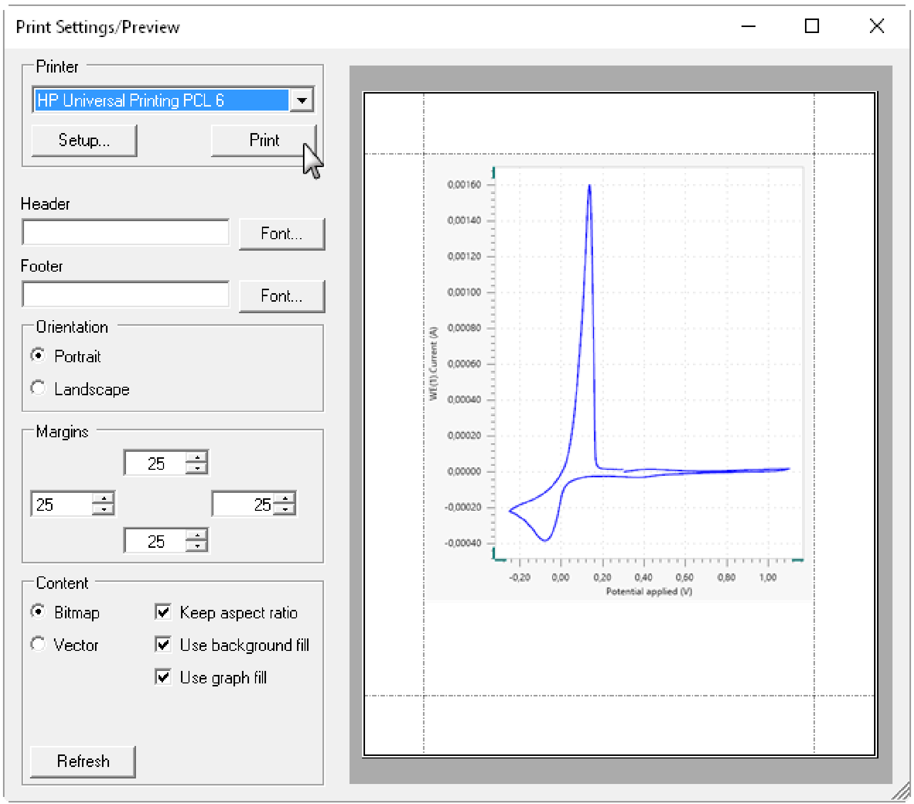


Figure 58 A print preview dialog is shown

NOTE

More information on the printing of plots can be found in *Chapter 11.8.6*.

3.5.7 Region insensitivity

This version of NOVA is completely region independent. The application uses the regional settings defined on the computer. The Calculate signal command has been modified for this purpose. Mathematical operators that use more than one argument now use the semi-colon (**;**) to separate the arguments in the mathematical expression.

3.5.8 Electrochemical interface toggle

For instruments fitted with the optional **ECI10M** module it is now possible to set the active electrochemical interface directly from **Instruments** panel in the **Dashboard**, using the right-click menu *(see Figure 59, page 63)*.

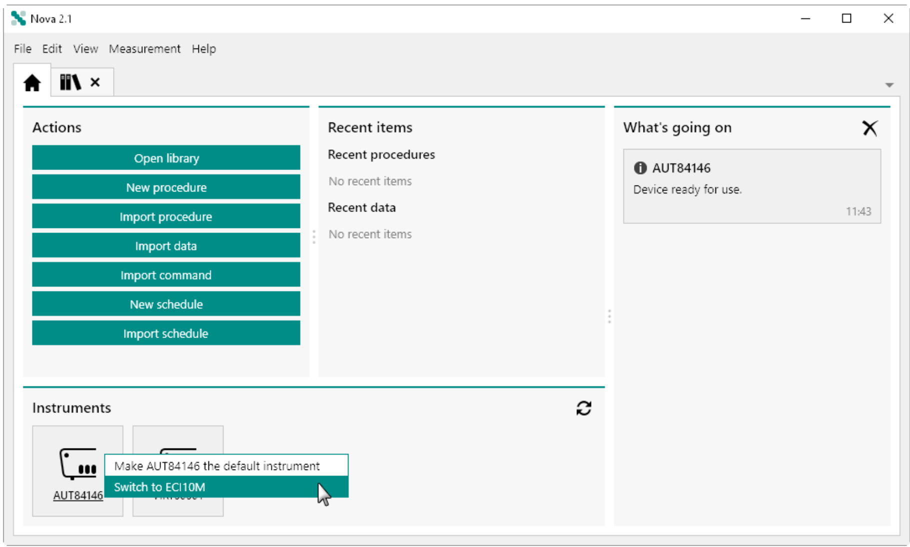


Figure 59 The electrochemical interface can be directly selected through the Dashboard

At any time, a tooltip shows the active electrochemical interface, in bold *(see Figure 60, page 64)*.

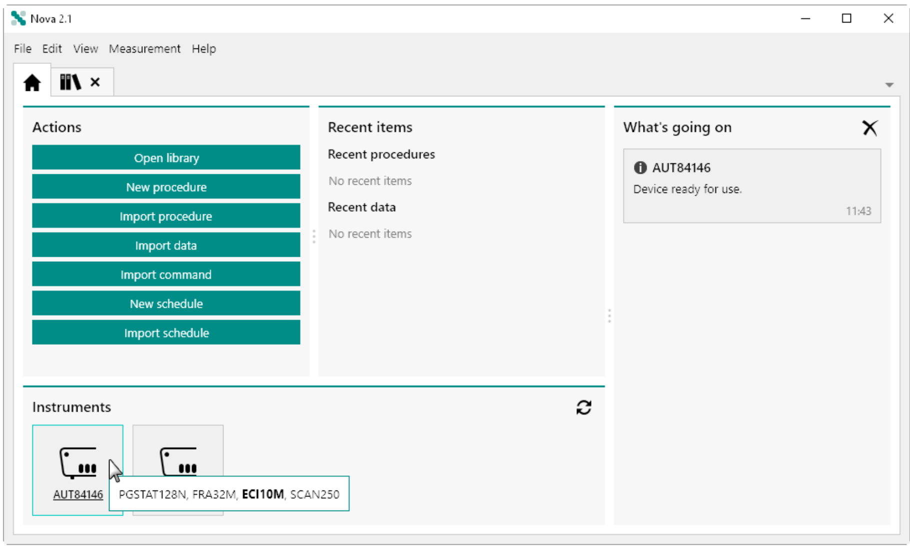


Figure 60 A tooltip shows the active electrochemical interface in bold

NOTE

More information on the optional ECI10M module can be found in *Chapter 16.3.2.8*.

3.5.9 ECI10M measurements

In order to more easily identify measurements carried out with the **ECI10M** as the active electrochemical interface, the **(ECI10M)** suffix will be shown in the procedure editor, next to the serial number of the active instrument, below the procedure title *(see Figure 61, page 65)*.

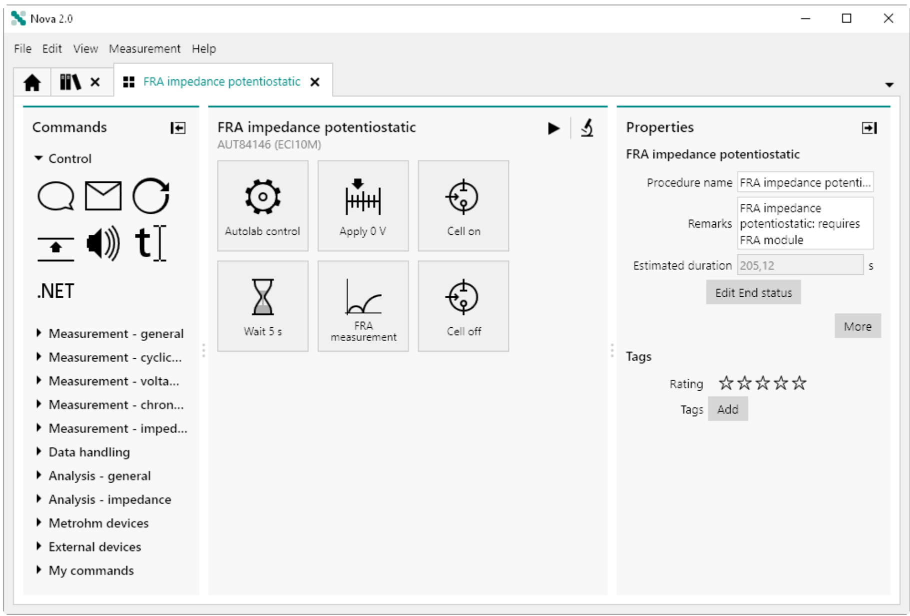


Figure 61 The ECI10M suffix is shown in the procedure editor

For measurements that have been carried out with the **ECI10M**, the same suffix will be added to the instrument serial number in the **Library***(see Figure 62, page 65)*.

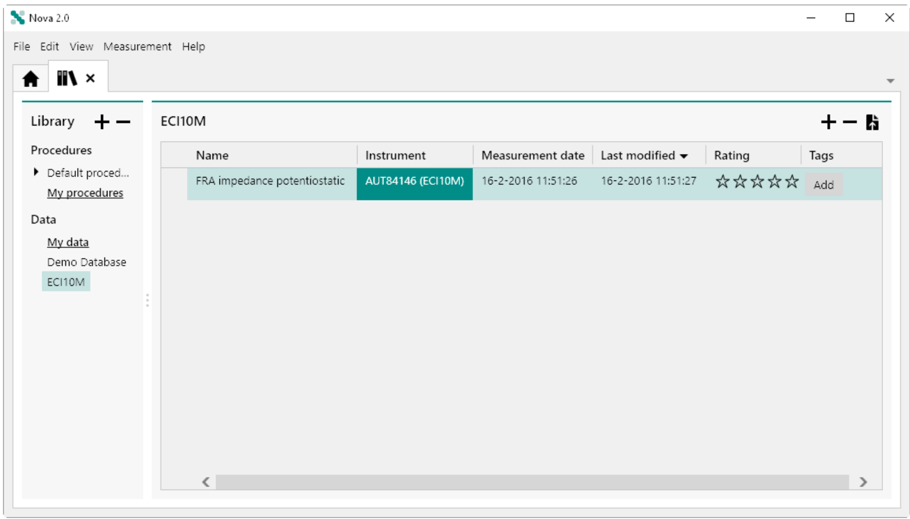


Figure 62 The ECI10M suffix is shown in the Library

NOTE

This suffix is not shown for measurements carried out with previous version of NOVA.

3.5.10 Library column display

The display settings of the grids used in the **Library** are now non-volatile. This change affects the following settings of the Library:

- **Column order:** the order in which the columns appear.
- **Column visibility:** the visibility of the available columns.
- **Sorting options:** the sorting options used in the Library.

NOTE

The display settings used in the Library are stored on the local computer and can be defined for each type of Library location.

NOTE

More information on the display settings used in the Library can be found in *Chapter 6.10* and *Chapter 6.13*.

3.5.11 Data grid column display

The display settings of the grids used in the **Data grid** are now non-volatile. This change affects the following settings of the data grid:

- **Column order:** the order in which the columns appear.
- **Column formatting:** the data formatting used in each column.
- **Sorting options:** the sorting options used in the data grid.

NOTE

These settings are stored for each command in the data file.

NOTE

More information on the display settings used by the data grid can be found in *Chapter 11.9*.

3.5.12 Estimated duration

The **Estimated duration** is now shown as read only field for all commands in the **Properties** panel *(see Figure 63, page 67)*.

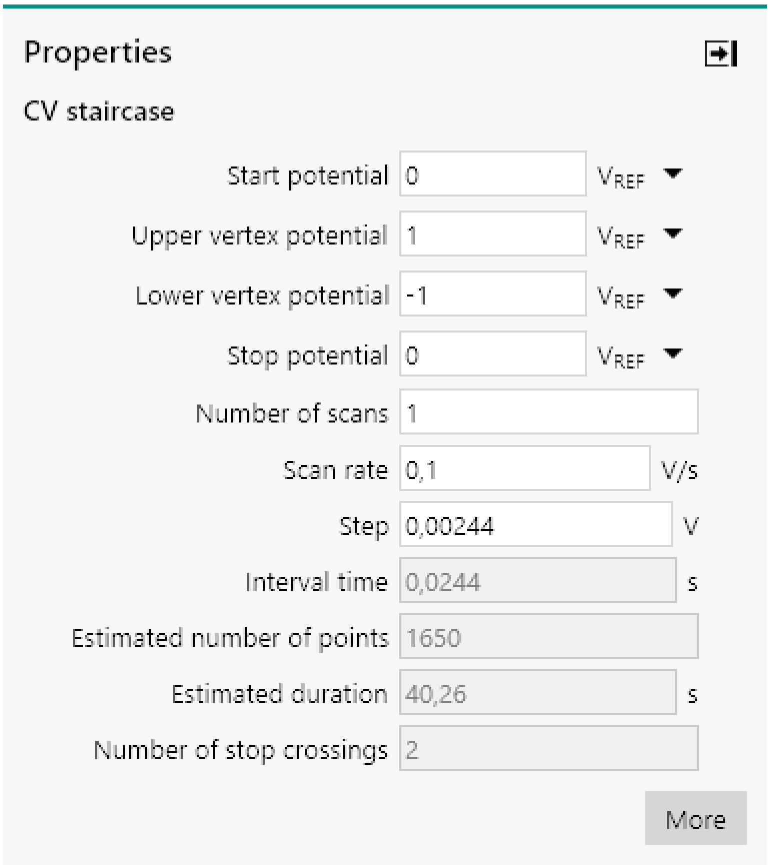


Figure 63 The Estimated duration is now specified in the Properties panel

If the command is part of a command stack, the Estimated duration value will take into account the duration of the all the underlying commands in the stack, as shown in *Figure 64*.

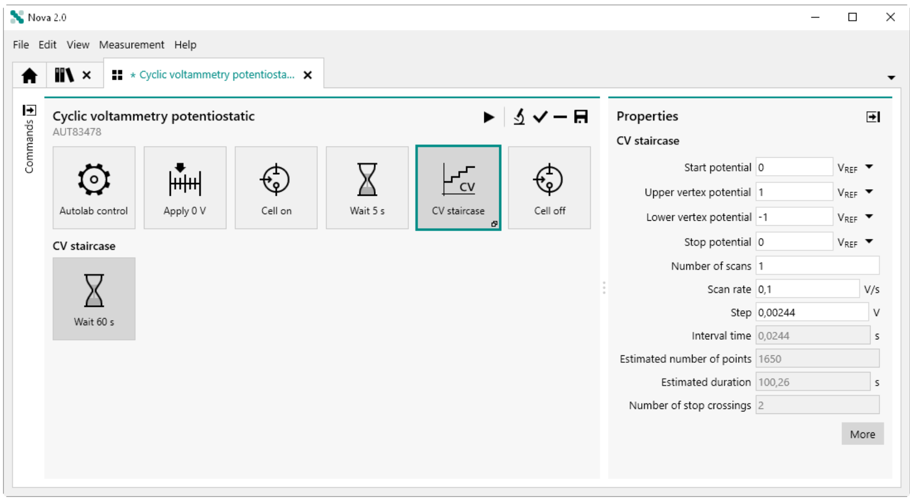


Figure 64 When commands are stacked, the Estimated duration takes underlying commands into account

NOTE

The Estimated duration is determined based on the interval time and the estimated number of points as well as the duration of underlying commands, if applicable.

3.5.13 Interpolate command

The new Interpolate command, available in the Analysis - general group of commands, is now available. This command can be used to determine Y or X value based on a user-defined X or Y value, by linear interpolation.

NOTE

More information on the Interpolate command can be found in *Chapter 7.8.6*.

3.5.14 Hydrodynamic analysis

The Hydrodynamic i vs $\sqrt{\omega}$ command has been renamed to **Hydrodynamic analysis** and it has been extended with the Koutecký-Levich analysis method .

> **NOTE**
>
> More information on the **Hydrodynamic analysis** command can be found *(see Chapter 7.8.10, page 371)*.

3.6 Version 2.0 release

NOVA 2.0 is completely redesigned to improve the user experience. The tree view from the previous generation of NOVA (NOVA 1.2 to NOVA 1.11) has been replaced by a graphical interface with a clear presentation. This manual explains all the controls of NOVA 2.0.

The following new functionality has been added to the software with respect to the previous versions of NOVA:

1. Dynamic data buffers *(see Chapter 3.6.1, page 69)*.
2. Value of alpha for staircase cyclic voltammetry and staircase linear sweep voltammetry *(see Chapter 3.6.2, page 70)*.
3. Support for the Autolab RHD Microcell HC system *(see Chapter 3.6.3, page 70)*.
4. PGSTAT204 and M204 compatibility with the Booster10A *(see Chapter 3.6.4, page 71)*.
5. Support for the ECI10M module *(see Chapter 3.6.5, page 71)*.
6. AC voltammetry procedure and command *(see Chapter 3.6.6, page 72)*.

3.6.1 Dynamic data buffers

NOVA 2.0 introduces **dynamic data buffers**, a data storage mechanism that allows data points to be stored during a measurement. This means that the measurement commands are no longer affected by static buffers and that they can record as many data points as needed. The actual limit of to the number of data points is therefore only limited to the storage space available on the computer.

> **NOTE**
>
> Dynamic buffers are not available for measurements carried out with the **Chrono methods** command and for all measurements carried out with the **ADC10M** module or the **ADC750** module.

3.6.2 Value of Alpha

For the **CV staircase** command and the **LSV staircase** command, the *Alpha value* advanced property is now available *(see Figure 65, page 70)*.

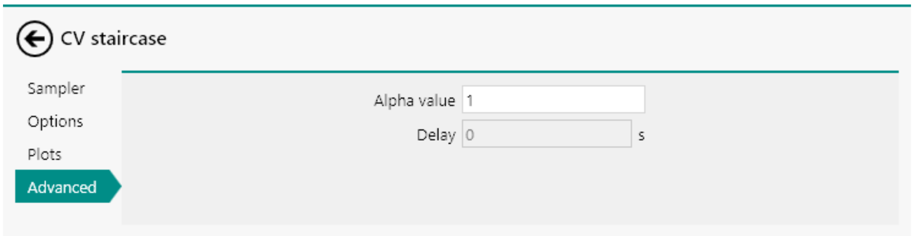


Figure 65 The Alpha value property is available for the CV staircase and LSV staircase command

NOTE

More information on the use of the value of Alpha can be found in *Chapter 9.7*.

3.6.3 Autolab RHD Microcell HC support

NOVA 2.0 introduces support for the Autolab RHD Microcell HC support *(see Figure 66, page 70)*.

Figure 66 The Autolab RHD Microcell system

NOTE

More information on the Autolab RHD Microcell HC is provided in *Chapter 5.3* and *Chapter 7.11.3*.

3.6.4 PGSTAT204 and M204 combination with Booster10A support

NOVA 2.0 introduces support for the combination of the Autolab PGSTAT204 potentiostat/galvanostat and the M204 potentiostat/galvanostat module with the Booster10A *(see Figure 67, page 71)*.

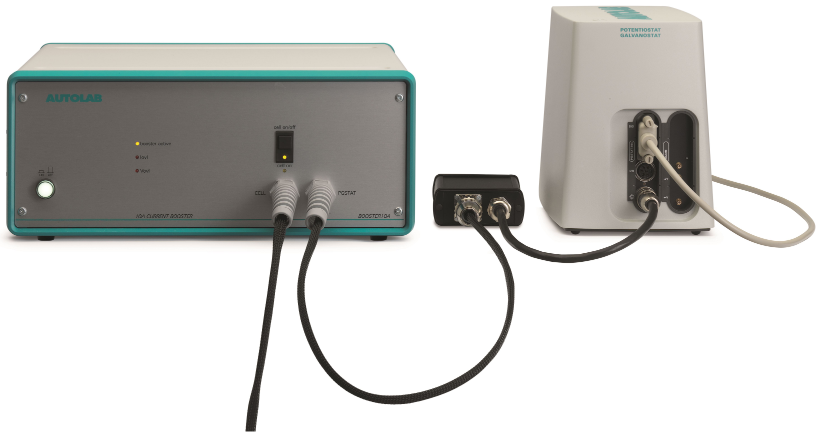


Figure 67 The combination of the Booster10A with the PGSTAT204/M204 system is now supported

NOTE

More information on the combination of the Booster10A with the PGSTAT204 and the M204 can be found in *Chapter 16.3.2.5*.

3.6.5 ECI10M module support

NOVA 2.0 introduces support for the ECI10M module *(see Figure 68, page 72)*.

Figure 68 NOVA introduces the support of the ECI10M module

NOTE

More information on the ECI10M module can be found in *Chapter 16.3.2.8*.

3.6.6 AC voltammetry

This new version of NOVA provides a new command for AC voltammetry measurements *(see Chapter 7.4.7, page 269)*. A default procedure for this electrochemical method is also available *(see Chapter 8.3.6, page 535)*.

4 Dashboard

The **Dashboard** is the home screen of NOVA. Whenever NOVA starts, the **Dashboard** is always shown to the user *(see Figure 69, page 73)*.

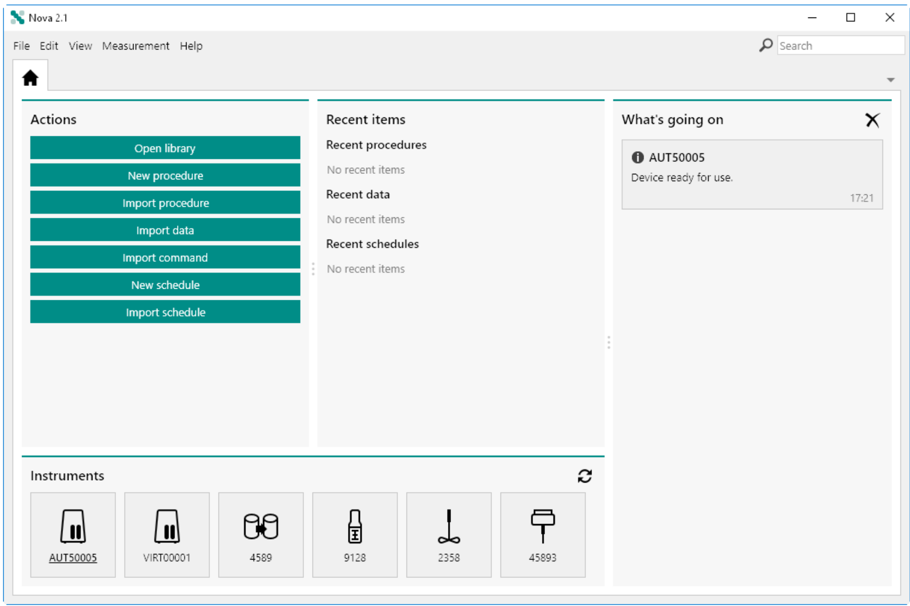


Figure 69 The Dashboard

At any time, when NOVA is used, it is possible to show the display by clicking the home tab ( ). The Dashboard provides four different panels:

- **Actions:** this panel provides a list a shortcut buttons to trigger a common task in NOVA *(see Chapter 4.1, page 74)*.
- **Recent items:** this panel provides a list of the last procedures, data and schedule items *(see Chapter 4.2, page 75)*.
- **What's going on:** this panel provides messages to the user about ongoing or finished events in NOVA *(see Chapter 4.3, page 77)*.
- **Instruments:** this panel provides a list of connected instruments *(see Chapter 4.4, page 79)*.

4.1 Actions

The **Actions** panel provides a series of buttons that can be used to quickly trigger a common action or control of the NOVA software *(see Figure 70, page 74)*.

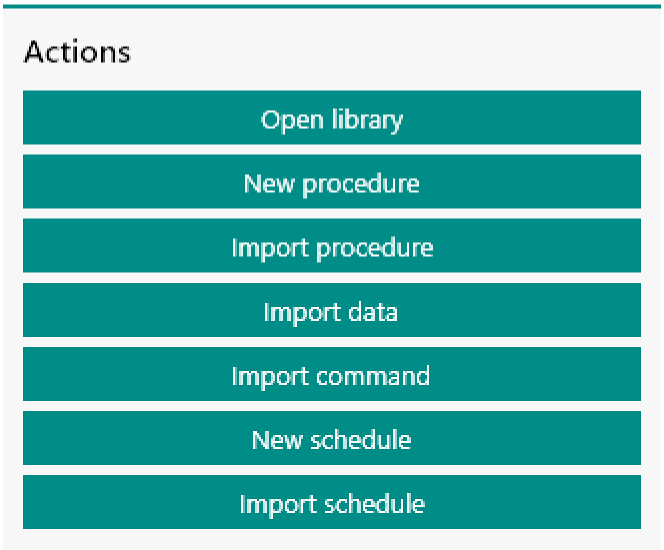


Figure 70 The Actions panel provides shortcut to the most common actions in NOVA

The following shortcut buttons are provided:

- **Open library:** this button open the **Library**. More information on the **Library** can be found in *Chapter 6*.
- **New procedure:** creates a new blank procedure. More information on the procedure editor can be found in *Chapter 10.1*.
- **Import procedure:** imports a procedure from a *.nox* file in the **Library**. More information on the **Library** can be found in *Chapter 6*.
- **Import data:** imports NOVA data from a *.nox* file in the **Library**. More information on the **Library** can be found in *Chapter 6*.
- **Import command:** imports a command from a *.noi* file in the **My Commands** group of command. More information on the **My Commands** can be found in *Chapter 10.14*.
- **New schedule:** creates a new procedure schedule. More information on the **Procedure scheduler** can be found in *Chapter 15*.
- **Import schedule:** imports a procedure schedule from a *.nos* file in the **Library**. More information on the **Procedure scheduler** can be found in *Chapter 15*.

NOTE

The **Import data** action can also be used to directly import data from the **GPES** and **FRA** software into NOVA.

4.2 Recent items

The **Recent items** panel lists the most recent procedures, data and schedules items *(see Figure 71, page 75)*.

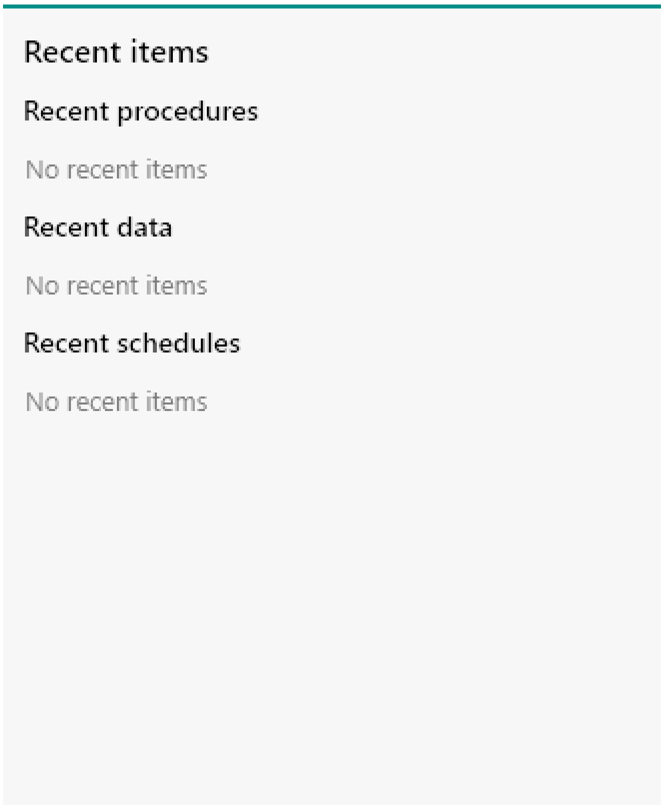


Figure 71 The Recent items panel shows the last procedures, data and schedules items

NOTE

By default, the **five** most recent items are shown in the **Recent items** panel. This number can be adjusted in the NOVA Options *(see Chapter 1.9, page 13)*.

The last items that are saved are automatically updated in the **Recent items** panel each time an item is saved *(see Figure 72, page 76)*.

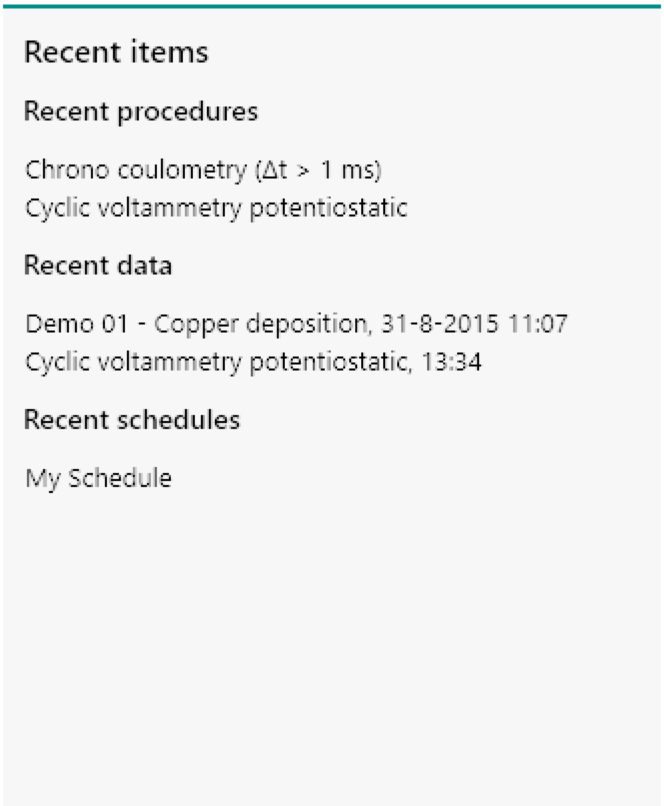


Figure 72 The Recent items panel is automatically updated when data or procedures are saved

It is possible to remove items from the **Recent items** panel by right-clicking an item and selecting the *Remove from recent items* option from the context menu *(see Figure 73, page 77)*.

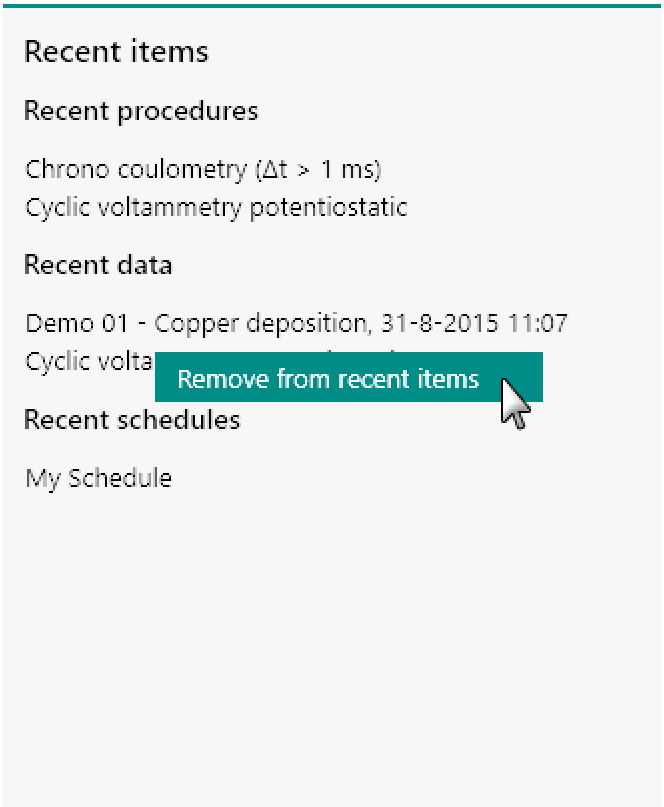


Figure 73 Removing items from the Recent items panel

4.3 What's going on

The **What's going on** panel is used to report information to the user. On startup, this panel will be populated with messages indicating that the connected Autolab instruments are ready for use *(see Figure 74, page 78)*.

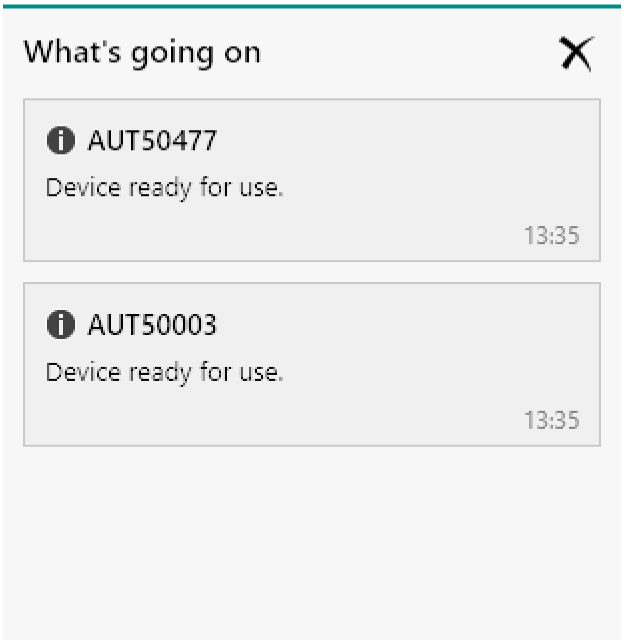


Figure 74 The What's going on panel is used to provide messages to the user

At any time, it is possible to clear the **What's going on** panel using the ✕ button *(see Figure 75, page 78)*.

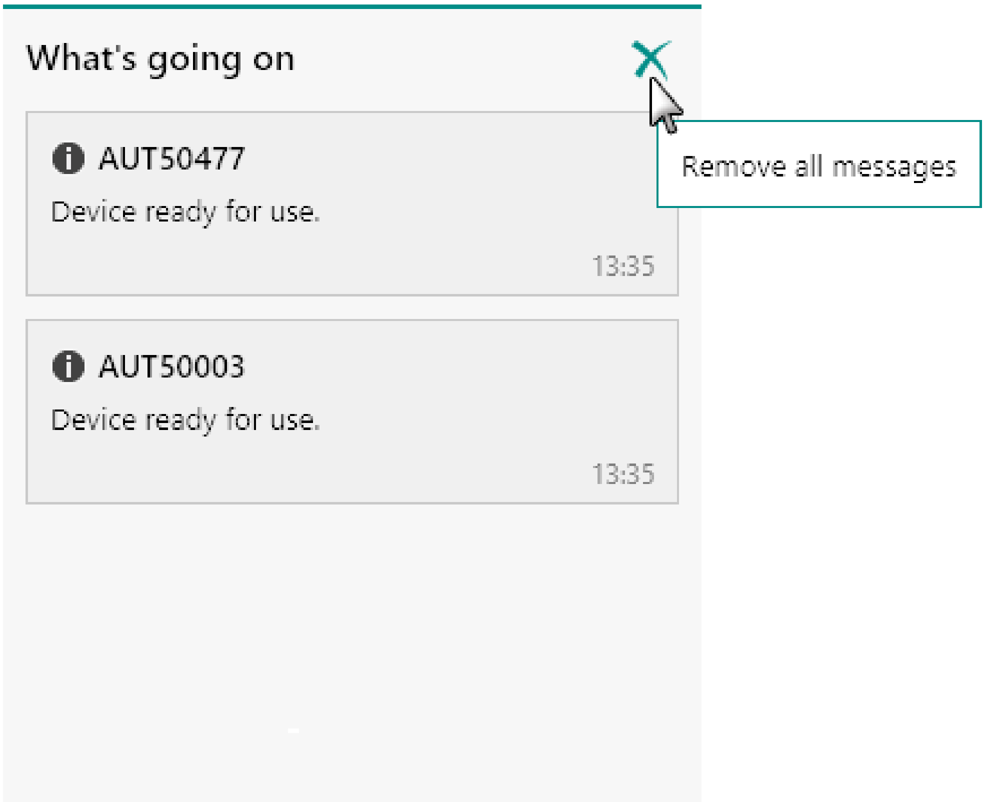


Figure 75 Clearing the panel content

The panel will be cleared of the currently displayed message.

Any time a measurement is started or a measurement stops, the **What's going on** panel will be updated *(see Figure 76, page 79)*.

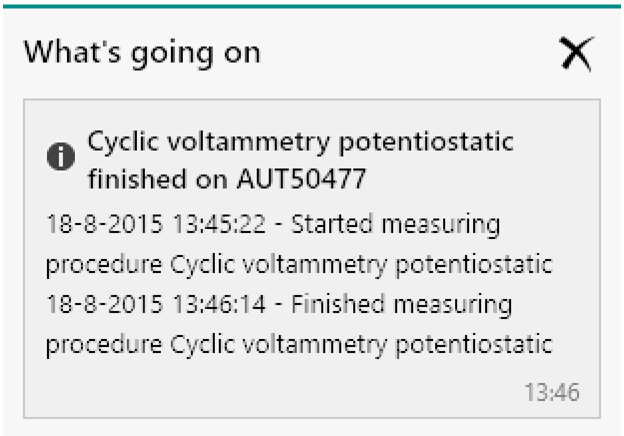


Figure 76 The What's going on panel is updated each time a measurement is started or finishes

4.4 Instruments panel

The **Instruments** panel shows all the connected instruments identified by NOVA *(see Figure 77, page 79)*.

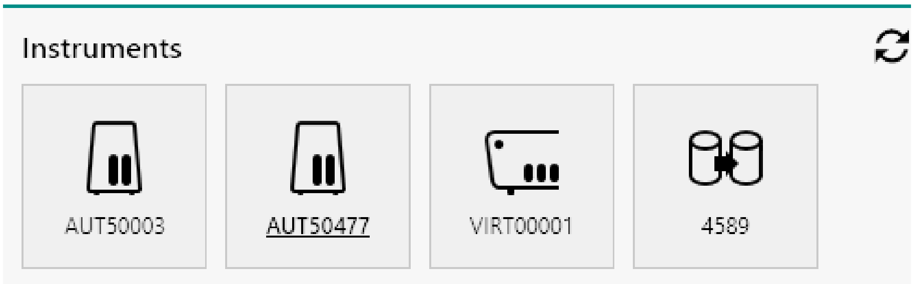


Figure 77 The Instruments panel lists all connected instruments

On startup, NOVA will look for all supported devices and will list these in **Instruments** panel. The content of the panel is automatically refreshed whenever an Autolab potentiostat/galvanostat, Autolab RHD Microcell HC controller of Autolab Spectrophotometer is connected or disconnected from the computer. The panel content is **not** refreshed automatically when a Metrohm liquid handling device is connected or disconnected. To refresh the content of the **Instruments** panel and update the list of available instruments, it is possible to click the button *(see Figure 78, page 80)*.

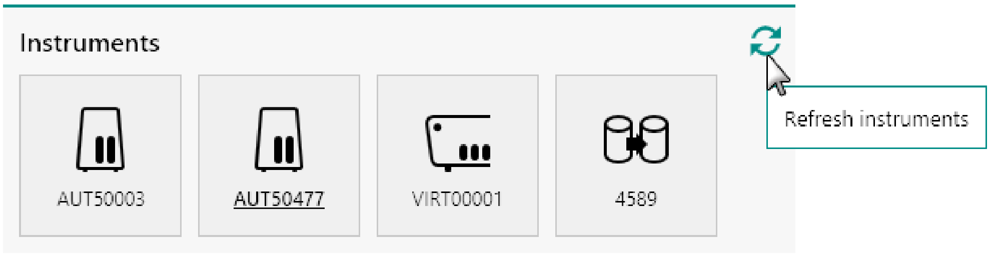


Figure 78 Clicking the refresh button will update the content of the Instruments panel

The content of the Instrument panel is updated when the button is clicked *(see Figure 79, page 80)*.

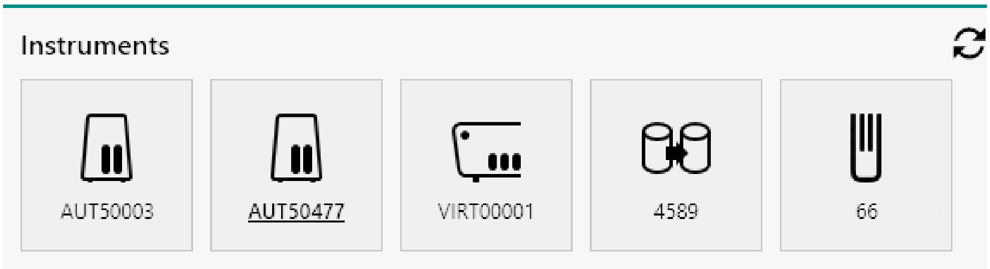


Figure 79 The Instruments panel is refreshed

5 Instruments panel

All connected instruments are listed in the **Instruments** panel of the dashboard *(see Figure 80, page 81)*.

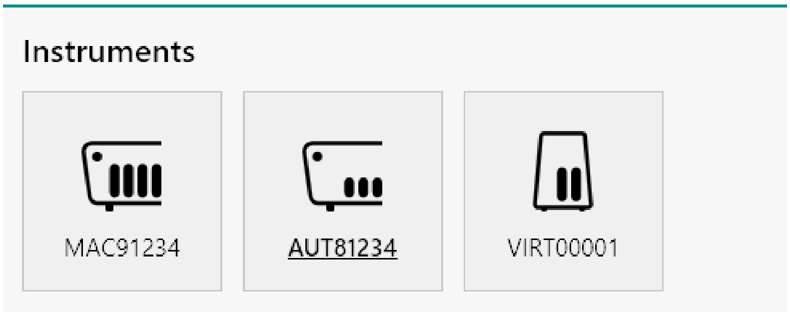


Figure 80 Connected instruments are listed in the Instruments panel

The following instruments can be identified by NOVA:

- Autolab potentiostat/galvanostat instruments with USB interface
- Autolab RHD Microcell HC controllers
- Autolab or Avantes spectrophotometers
- Supported Metrohm liquid handling devices

The instruments are identified by type and by serial number, and are represented by a device tile with a drawing of the instrument. Alongside the connected instrument, a **virtual** instrument is also shown.

NOTE

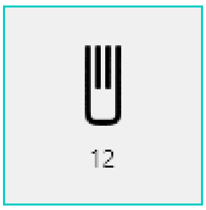
The virtual instrument is provided for procedure validation purposes.

The following device tiles are used to identify the connected **Autolab** instruments:

	This symbol is used to identify all **Autolab N Series** instruments (PGSTAT302N, PGSTAT128N, PGSTAT100N) or **Autolab F Series** instruments (PGSTAT302F). These instruments have a serial number starting with **AUT8**.

	This symbol is used to identify all **Autolab PGSTAT204** instruments. These instruments have a serial number starting with **AUT5**.
	This symbol is used to identify all **Autolab PGSTAT101** instruments. These instruments have a serial number starting with **AUT4**.
	This symbol is used to identify all **Multi Autolab Series** instruments (M101 and M204). These instruments have a serial number starting with **MAC8** (for the M101 Multi Autolab systems) and **MAC9** (for the M204 Multi Autolab systems).
	This symbol is used to identify all **µAutolab type II** and **µAutolab type III** instruments. The µAutolab type II instruments are identified by a serial number starting with **µ2AUT7** and the µAutolab type III instruments are identified by a serial number starting with **µ3AUT7**.
	This symbol is used to identify all **Autolab 7 Series** instruments (PGSTAT302, PGSTAT30, PGSTAT12, PGSTAT100) as well as the older **Autolab 9 Series** instruments (PGSTAT30, PGSTAT20, PGSTAT10 and PGSTAT100). These instruments are identified by a serial number starting with **AUT7** or **USB7**.

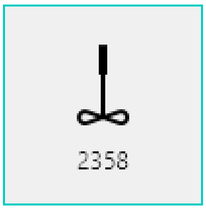
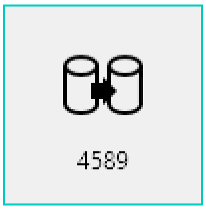
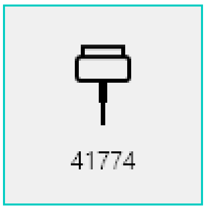
The following tiles are used to identify the connected **Autolab RHD Microcell HC** controllers:

	This symbol is used to identify all **Autolab RHD Microcell HC controllers** connected to the computer through a RS232 connection. These instruments are identified by their serial number (or device name).

The following tiles are used to identify the connected **Autolab or Avantes** spectrophotometers:

	This symbol is used to identify all **Autolab or Avantes spectrophotometers** connected to the computer through a USB connection. These instruments are identified by their serial number (or device name).

The following tiles are used to identify the connected **Metrohm** devices:

	This symbol is used to identify all **Metrohm 800 Dosino** devices connected to a USB controlled Metrohm device. These instruments are identified by their serial number (or device name).
	This symbol is used to identify all **Metrohm 801 Magnetic Stirrers** or **804 Titration Stands** with a stirrer connected to it (either **Metrohm 802 Rod Stirrer** or **Metrohm 741 Magnetic Stirrer**). These instruments are identified by their serial number (or device name).
	This symbol is used to identify all **Metrohm 814**, **815** or **858 Sample Processor** devices connected by USB to the host computer. These instruments are identified by their serial number (or device name).
	This symbol is used to identify all **Metrohm 6.2148.010 Remote Box** devices connected to a USB controlled Metrohm Device. These instruments are identified by their serial number (or device name).

In *Figure 80*, two instruments are connected (a Multi Autolab system with Serial Number MAC91234 and an Autolab N Series instrument with serial number AUT81234). A virtual instrument, with serial number VIRT00001 is also connected. This instrument is identified as a PGSTAT204.

The serial number of the N series instrument is shown in bold underlined font (**AUT81234**) indicating that this is the *default* instrument.

NOTE

When a measurement is started, it will always be executed on the *default* instrument, unless otherwise specified.

The following actions can be performed in the **Instruments** panel:

- Change the *default* **Autolab** instrument.
- Open the instrument control panel.

5.1 Change the default instrument

The default instrument, displayed in the bold underline in the **Instruments** panel, is the instrument used in any measurement, unless otherwise specified. This is also the instrument used for procedure validation purposes.

To change the default instrument, right-click any instrument tile in the **Instruments** panel and select the *Make* **[Instrument serial number]** *the default instrument* from the context menu *(see Figure 81, page 84)*.

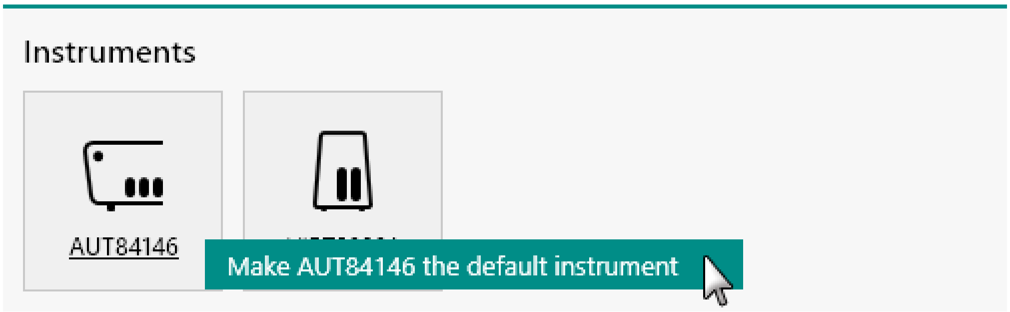


Figure 81 Defining the default instrument

NOTE

Only one instrument can be set as default instrument.

5.2 Autolab control panel

Double clicking an Autolab tile in the **Instruments** panel opens the **Autolab control** panel in a new **tab**, as shown in *Figure 82* and *Figure 83*.

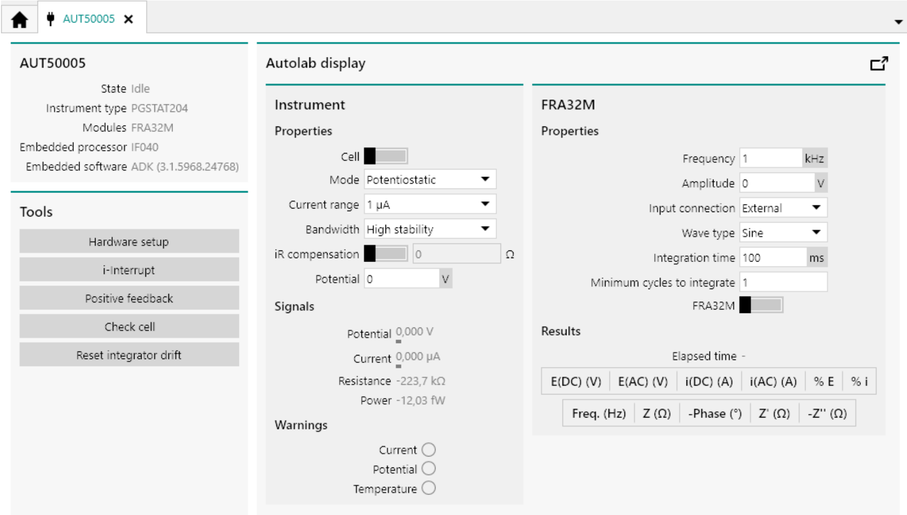


Figure 82 The single channel Autolab control panel

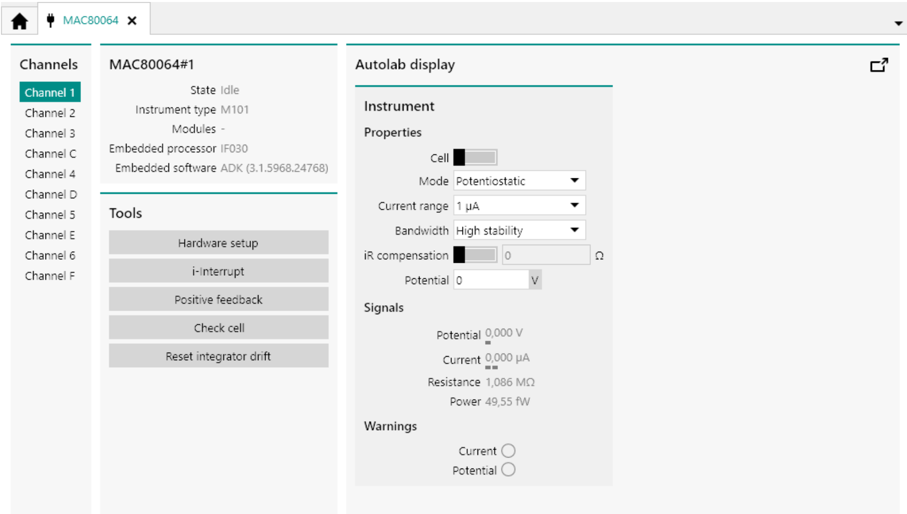


Figure 83 The multi channel Autolab control panel

Depending on the type of instrument, the Autolab control panel shows either three or four sub-panels:

- **Channels:** this panel displays the available channels located in the Multi Autolab instrument. The information of the highlighted channel is shown in the rest of the screen. This panel is only visible for Multi Autolab instruments.

- **Instrument information panel:** this panel displays information about the instrument.
- **Tools panel:** this panel provides quick access to the hardware setup and a number of direct measurement tools like current interrupt and positive feedback.
- **Autolab display panel:** this panel provides a number of manual controls of the instrument.

NOTE

The available channels in a multi channel Autolab are listed in the **Channels** sub-panel. Each channel is identified by a letter or a number. More information is provided in *Chapter 16.2.5*.

5.2.1 Instrument information panel

The **Instrument information** panel shown in the instrument control panel provides information on the selected instrument *(see Figure 84, page 86)*.

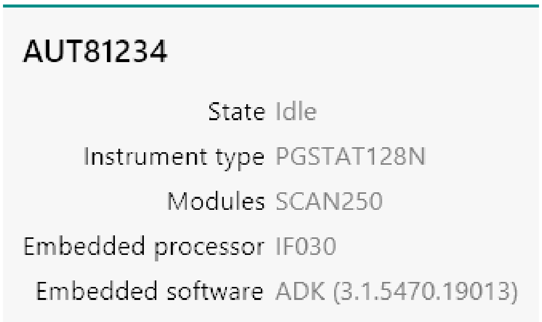


Figure 84 The Instrument information panel

This information is updated in real time and is provided for information only. The following items are listed:

- **State:** indicates the state of the instrument (idle or measuring).
- **Instrument type:** indicates the type of instrument.
- **Modules:** shows the extension modules of the instrument.
- **Embedded processor:** shows the type of embedded processor installed in the instrument (IF030 or IF040).
- **Embedded software:** shows the embedded application name and version number.

NOTE

The embedded processor and embedded software reported in the instrument information panel are provided for information purposes only.

5.2.2 Tools panel

The **Tools** panel, shown in *Figure 85*, provides access to the following controls:

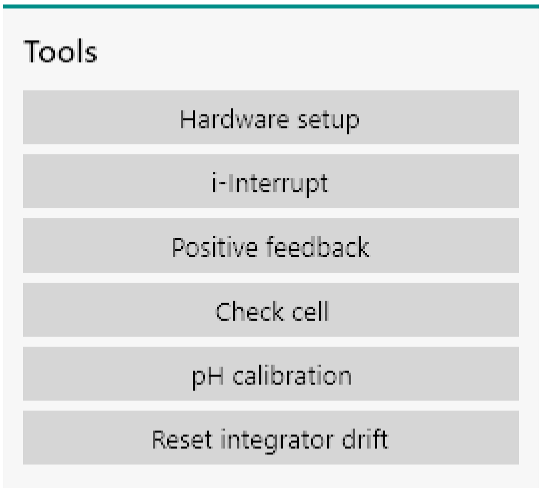


Figure 85 The Tools panel

- **Hardware setup:** used to adjust the hardware configuration of the instrument *(see Chapter 5.2.2.1, page 88)*.
- **i-Interrupt:** performs a current interrupt measurement (i-Interrupt) on the connected electrochemical cell in order to determine the uncompensated resistance value, R_u, by regression *(see Chapter 5.2.2.2, page 90)*.
- **Positive feedback:** performs a positive feedback measurement in order to determine the uncompensated resistance value, R_u, by inspecting the control loop stability on the connected electrochemical cell *(see Chapter 5.2.2.3, page 96)*.
- **Check cell:** performs a noise test to evaluate the electrode connections *(see Chapter 5.2.2.4, page 99)*.
- **pH calibration:** calibrates the pH sensor connected to the instrument *(see Chapter 5.2.2.5, page 103)*.
- **Reset integrator drift:** performs a determination of the drift, in C/s, of the analog integrator and resets the compensation of the drift to the appropriate value *(see Chapter 5.2.2.6, page 115)*.

NOTE

The i-Interrupt, Positive feedback, pH calibration and Reset integrator drift tools are only shown on if the instrument provides the functionality used by these tools.

5.2.2.1 Hardware setup

The Hardware setup button can be used to edit the **Hardware setup**. The hardware setup screen shows three panels *(see Figure 86, page 88)*.

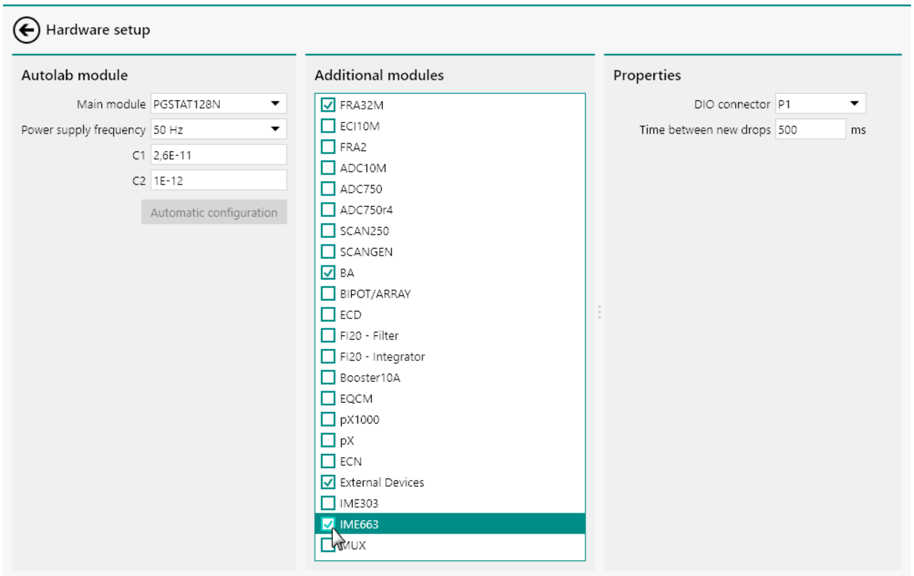


Figure 86 The hardware setup

The following panels are provided:

- **Autolab module panel:** used to specify the type of Autolab and additional properties of this instrument.
- **Additional modules panel:** provides a list of compatible extension modules which can be installed in the instrument or connected to the instrument.
- **Properties panel:** provides additional parameters for the extension modules.

5.2.2.1.1 Autolab module panel

The **Autolab module** panel can be used to specify the following properties *(see Figure 87, page 89)*:

- **Main module:** specifies the type of Autolab using the provided drop-down list.
- **Power supply frequency:** specifies if the mains frequency is 50 or 60 Hz.

- **C1 and C2:** specifies the C1 and C2 correction parameters for electrochemical impedance measurements.

NOTE

The values of **C1** and **C2** are preconfigured when NOVA is installed from the CD-ROM or USB support provided with the instrument. If these values are not configured properly (0 by default), the values can be determined experimentally. Please refer to *Chapter 16.3.2.13.3* and *Chapter 16.3.2.12.4* for more information.

Furthermore, an additional Automatic configuration button is provided. Clicking this button automatically adjusts the hardware setup of the instrument. The main module and the optional modules are determined based on the information stored on the instrument *(see Figure 87, page 89)*.

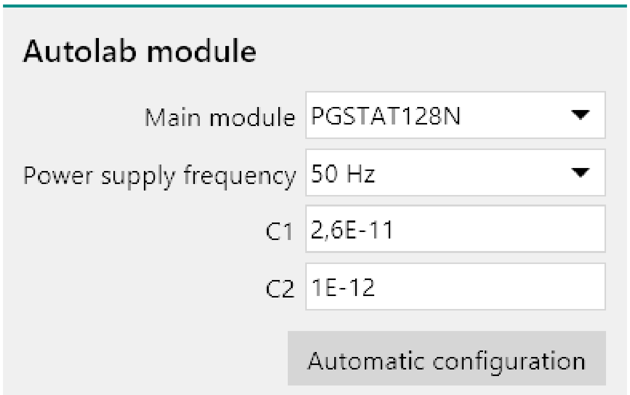


Figure 87 The Autolab module panel

5.2.2.1.2 Additional modules and properties panel

The **Additional modules** panel can be used to specify the optional modules installed in the instrument or connected to the instrument. The list of available modules depends on the main module specified in the Autolab module panel. The **Properties panel** on the right-hand side of the **Additional modules** panel can be used to specify additional properties of modules installed in the instrument or connected to the instrument *(see Figure 88, page 90)*.

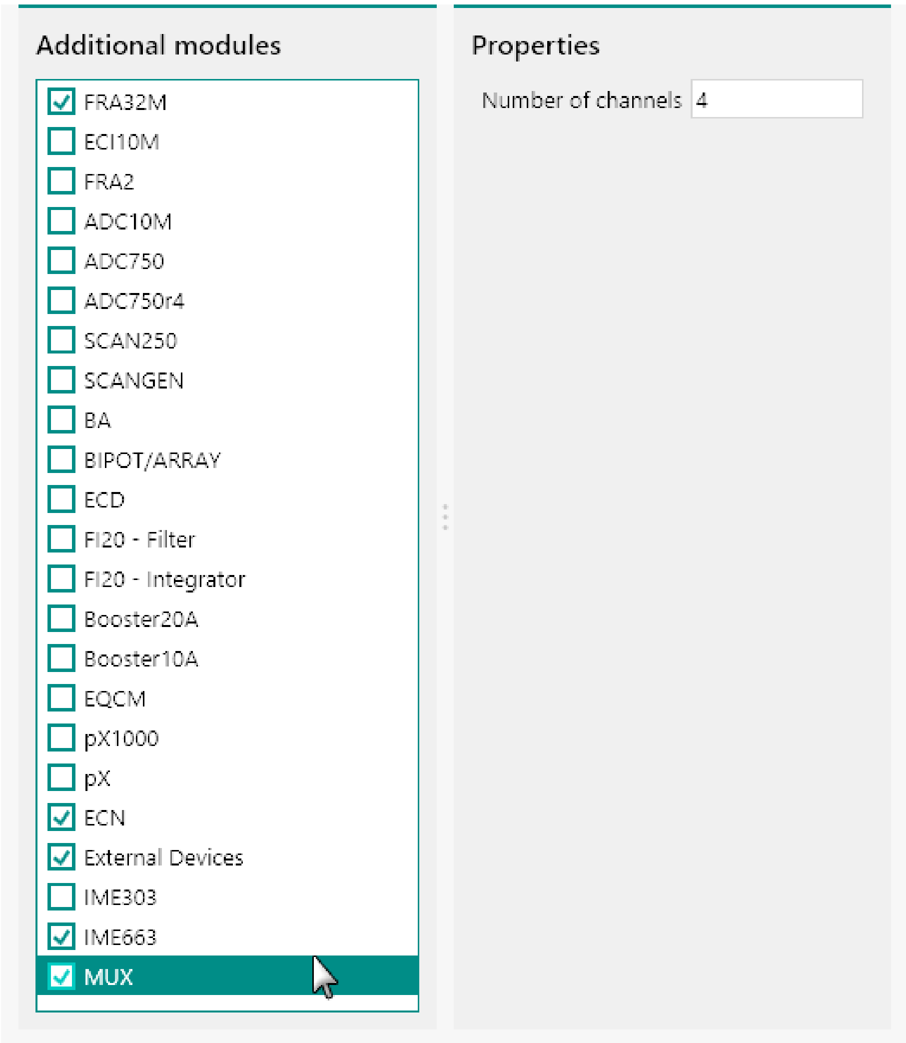
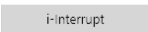


Figure 88 Optional module can be selected in the optional modules panel

NOTE

Only the FRA2, BA, FI20 - Integrator, Booster10A, Booster20A, External Devices, IME303, IME663 and MUX module have additional properties to display in the **Properties** panel.

5.2.2.2 Current interrupt

The 

button can be used to perform a current interrupt (i-Interrupt) measurement. This tool can be used to determine the uncompensated resistance, R_u.

NOTE

This tool is not available for µAutolab type II and type III instrument as well as the Autolab PGSTAT10.

During a current interrupt measurement, a constant potential is applied on the cell before the current interrupt circuit is triggered. This circuit interrupts the current flow in the cell and measures the potential decay. From the measured potential decay, the uncompensated resistance (R_u) value is determined, using a linear and an exponential regression.

Two values of the uncompensated resistance, R_u, are determined automatically at the end of the measurement:

- **Ru linear:** this value is obtained from a linear regression performed on the initial segment of the voltage decay.
- **Ru exponential:** this value is obtained from an exponential regression performed on the initial segment of the voltage decay.

Proper determination of this value requires an accurate measurement of the current. The measurements must therefore be carried out at a potential value where the current is high enough to be measured properly and the current range must be adjusted in accordance.

NOTE

For accurate measurements, the current should be at least in the order of 1 mA.

When the i-Interrupt tool is used, the control screen for this tool will be displayed *(see Figure 89, page 92)*. The control screen provides two panels and one plot area.

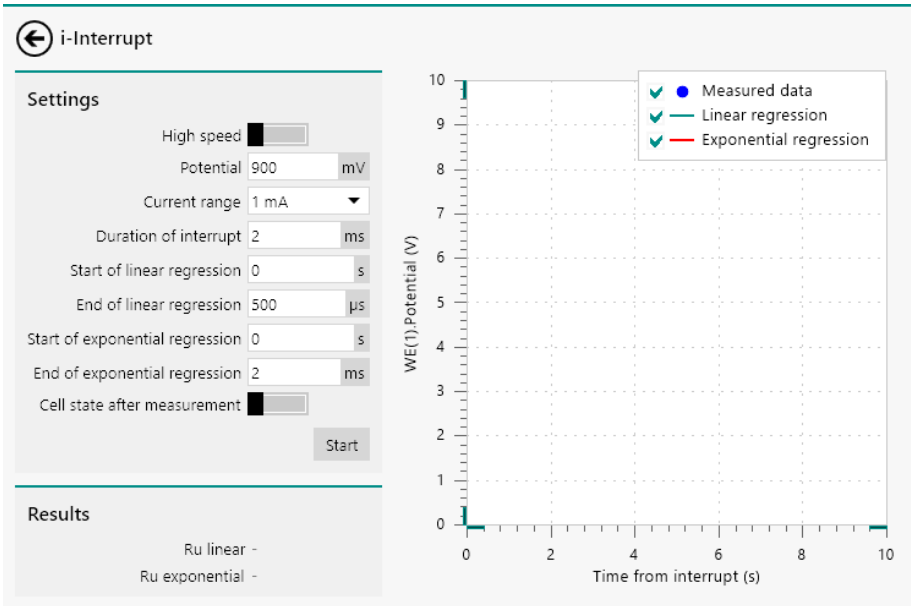


Figure 89 The i-Interrupt tool

The **Settings** panel shows the properties used in the current interrupt measurement *(see Figure 90, page 92)*.

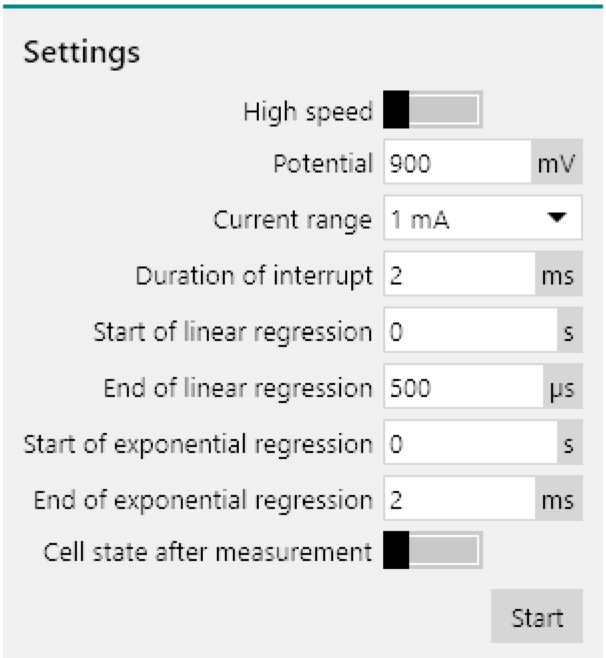


Figure 90 The i-Interrupt Settings panel

The following properties and controls are available:

- **High speed:** a  toggle that can be used to switch the high speed ADC module (ADC10M or ADC750) off or on (default off).

NOTE

This **High speed** property is only shown when the instrument is fitted with the optional **ADC10M** or **ADC750** module *(see Chapter 16.3.2.1, page 977)*. If the instrument is fitted with this optional module it is highly recommended to use the high speed mode since it will decrease the interval time from 100 µs to 4 µs, resulting in a more reliable regression.

- **Potential:** the potential applied on the cell before the current interrupt circuit is triggered, in V.
- **Current range:** the current range in which the current interrupt measurement is performed.
- **Duration of the interrupt:** the duration of the current interrupt measurement, in s.
- **Start of linear regression:** the abscissa of the first point on the time axis relative to the start of the interrupt used for *linear* regression, in s.
- **End of linear regression:** the abscissa of the last point on the time axis relative to the start of the interrupt used for *linear* regression, in s.
- **Start of exponential regression:** the abscissa of the first point on the time axis relative to the start of the interrupt used for *exponential* regression, in s.
- **End of exponential regression:** the abscissa of the last point on the time axis relative to the start of the interrupt used for *exponential* regression, in s.
- **Cell state after measurement:** a toggle that can be used to define the state of the cell switch at the end of the measurement (default off).

Clicking the Start button initiates the current interrupt measurement, using the specified properties.

CAUTION

The current interrupt tool switches the cell on and applies a constant potential before triggering the current interrupt circuit. It is highly recommended to specify the measurement properties carefully before starting the measurement.

While the current interrupt measurement is running, the spinning symbol is shown. The instrument cannot be used until the measurement is finished. When the measurement is finished, the measured data is displayed next to the **Measurement** panel *(see Figure 91, page 94)*.

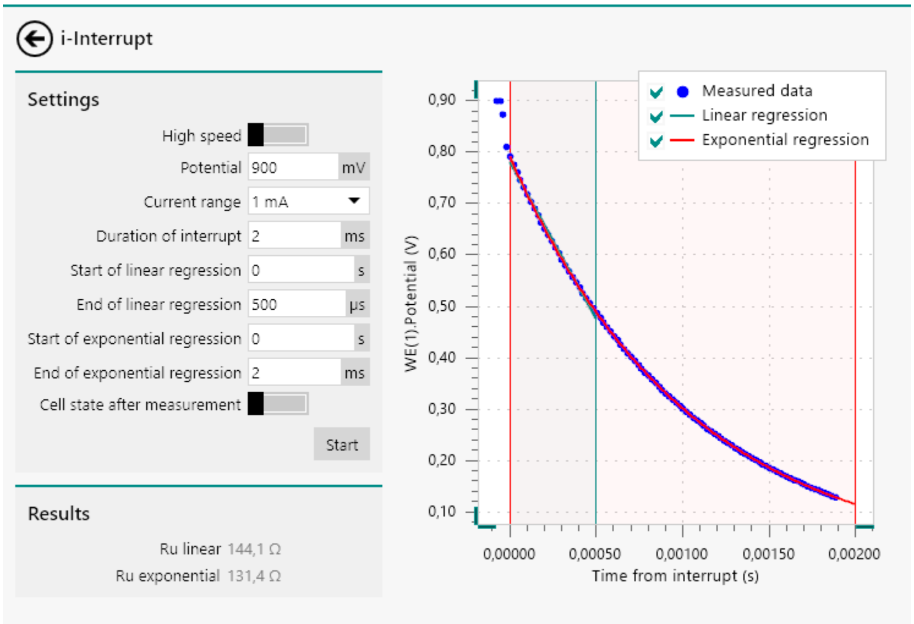


Figure 91 The measured and fitted data

The measured data points are shown as a point plot. The linear regression is shown using a green line and the exponential regression is shown as a red line. The start and end value of the two regression methods are shown using vertical lines with matching colors.

It is possible to hide or show the measured data or the regression data by checking or unchecking the check boxes shown in the legend *(see Figure 92, page 94)*.

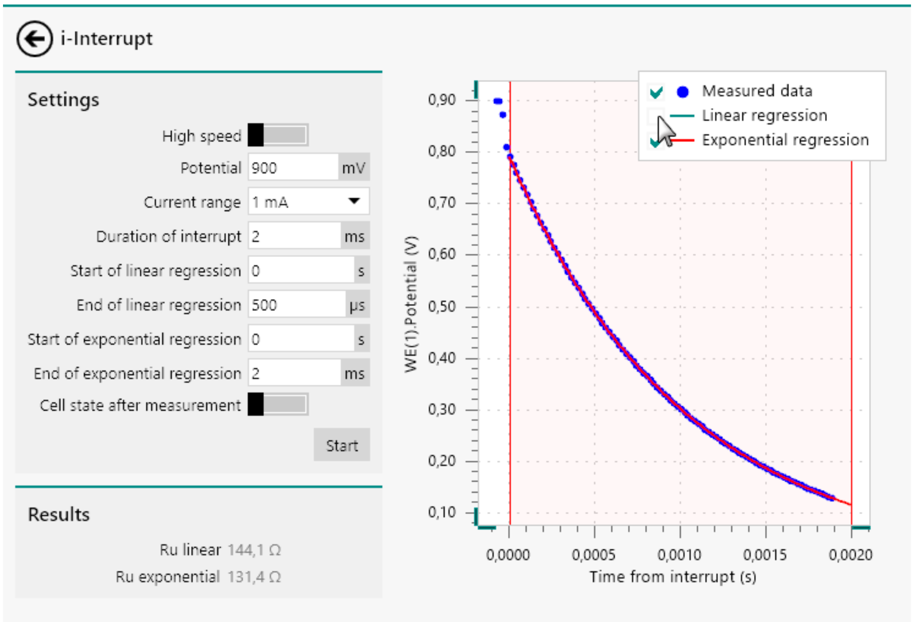


Figure 92 Using the check boxes to show or hide the data measured during the current interrupt

It is possible to fine-tune the start and stop values of both regressions by adjusting the properties in the **Settings** panel or by clicking and dragging the vertical lines *(see Figure 93, page 95)*.

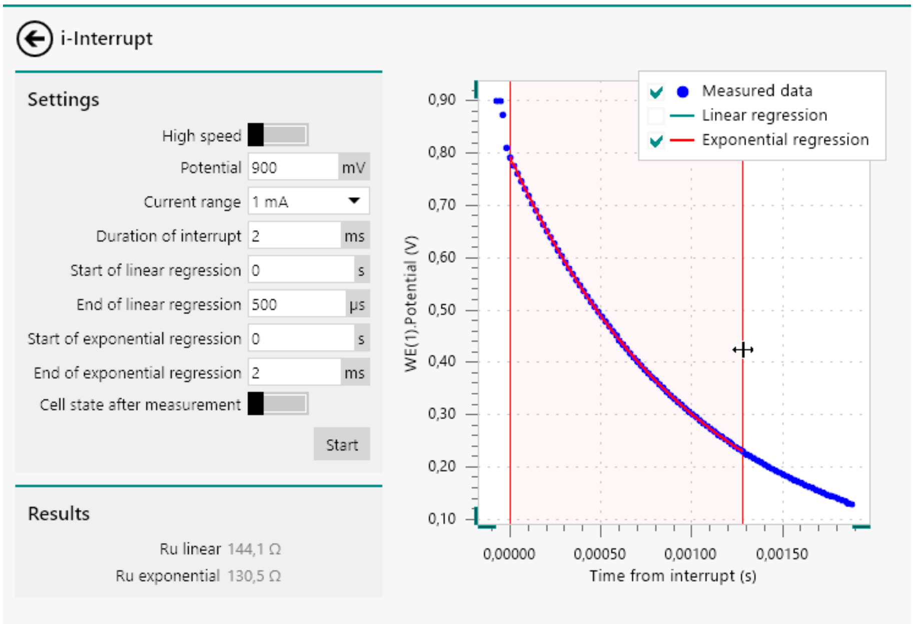


Figure 93 Fine-tuning the regression parameters

If the regression properties are adjusted after the measurement, the R_u is automatically recalculated.

The **Results** panel shows the calculated values of R_u, determined from the experimental data by linear and exponential regression *(see Figure 94, page 95)*.

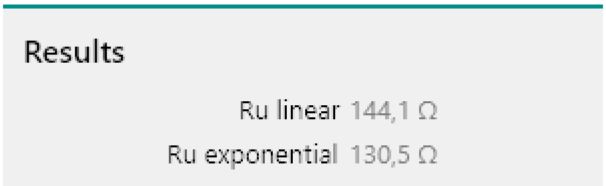


Figure 94 The fitted R_u values

It is possible to copy either one of the two values by right-clicking one of R_u values shown in the **Results** panel and using the *Copy value* option shown in the context menu *(see Figure 95, page 95)*.

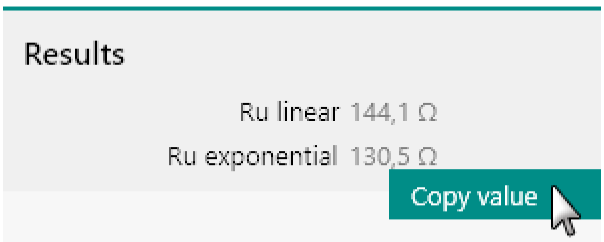


Figure 95 Copying the fitted value of R_u

NOTE

The copied value can be pasted in a suitable property field in NOVA.

5.2.2.3 Positive feedback

The Positive feedback button can be used to perform a positive feedback measurement. This tool can be used to determine the uncompensated resistance, R_u.

NOTE

This tool is not available for μAutolab type II and type III instrument as well as the Autolab PGSTAT10.

During a positive feedback measurement, a potential pulse is applied on the cell and the potential is recorded. The iR compensation value can be adjusted upwards manually until its value is close to the actual value of the uncompensated resistance, R_u. When the compensated resistance reaches a value close to the actual value of R_u, potentiostatic loop will start to ring. When the compensated resistance exceeds the R_u value, the potentiostatic loop is no longer stable and the instrument will oscillate.

When the positive feedback tool is used, the control screen for this tool will be displayed *(see Figure 96, page 96)*. The control screen provides two panels and one plot area.

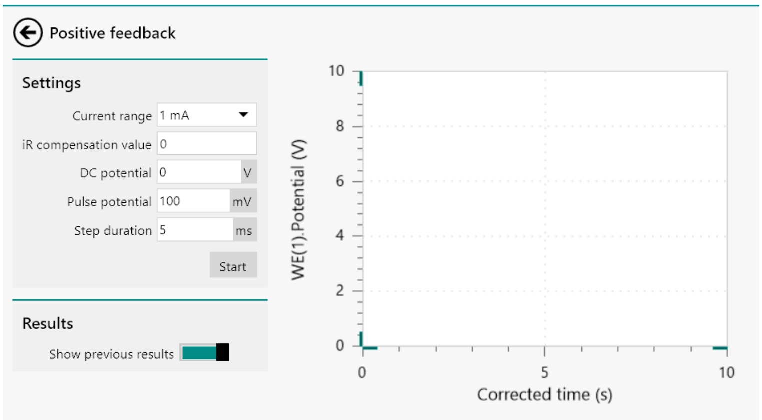


Figure 96 The positive feedback tool

The **Settings** panel shows the properties used in the current interrupt measurement *(see Figure 97, page 97)*.

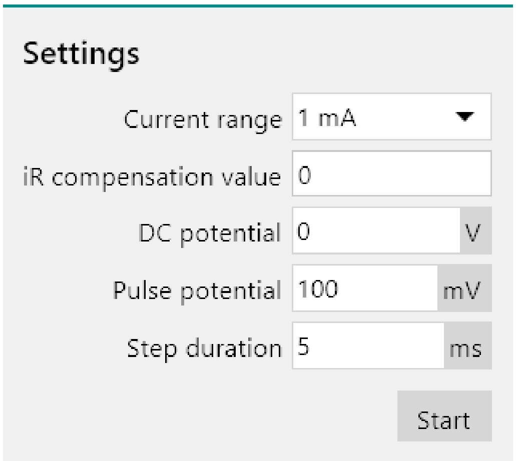


Figure 97 The positive feedback Settings panel

The following properties and controls are available:

- **Current range:** the current range in which the positive feedback measurement is performed.
- **iR compensation value:** the value of the compensated resistance, Ω.
- **DC potential:** the start and stop potential applied during the positive feedback measurement, in V.
- **Pulse potential:** the potential value applied in the pulse during the positive feedback measurement, in V.
- **Step duration:** the duration of the pulse applied during the positive feedback measurement, in s.

Clicking the Start button initiates the positive feedback measurement, using the specified properties.

CAUTION

The positive feedback tool switches the cell on and applies a potential pulse. It is highly recommended to specify the measurement properties carefully before starting the measurement.

While the positive feedback measurement is running, the spinning symbol is shown. The instrument cannot be used until the measurement is finished. When the measurement is finished, the measured data is displayed next to the **Settings** panel *(see Figure 98, page 98)*.

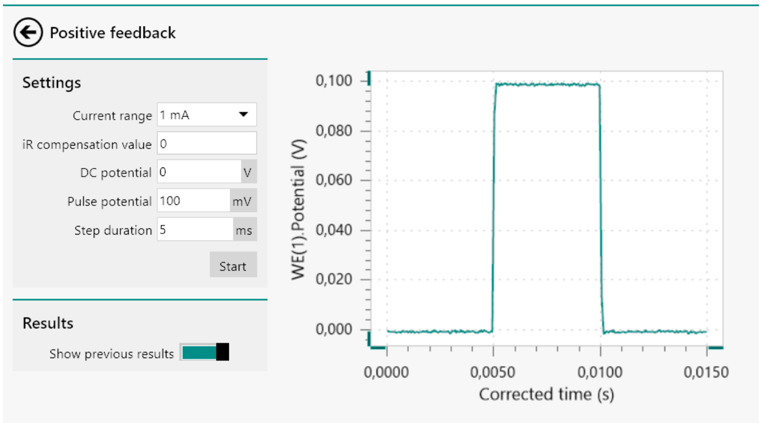


Figure 98 The measured data

The measured data shows the potential profile applied on the cell. Since the positive feedback tool uses an iterative approach, it is possible to adjust the value of the iR compensation value property and repeat the measurement *(see Figure 99, page 98)*.

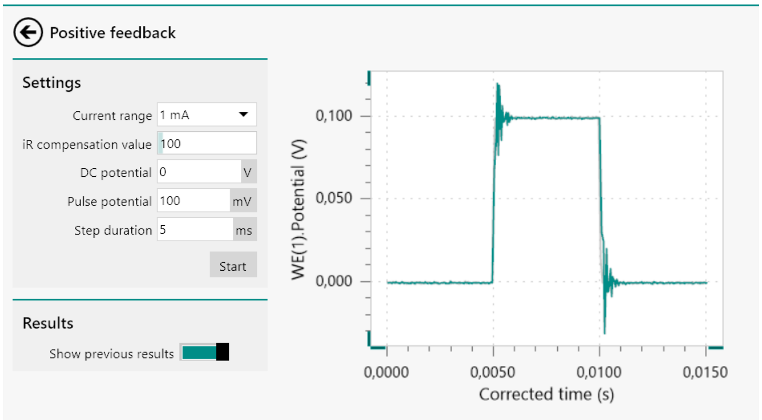


Figure 99 The measured data and previous results shown in overlay

The Show previous results toggle provided in the **Results** panel can be used to show or hide the data from the previous measurement *(see Figure 100, page 98)*.

NOTE

The tool only stores the current measured data and the data from the previous measurement.

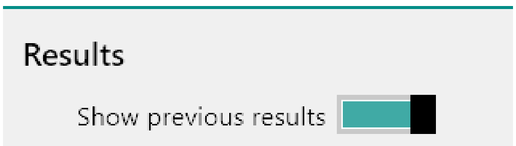


Figure 100 The previous data can be enabled and disabled

The tool can be used to test different iR compensation values. When the actual uncompensated resistance value is exceeded, the measured potential profile will become unstable and will start to oscillate *(see Figure 101, page 99)*.

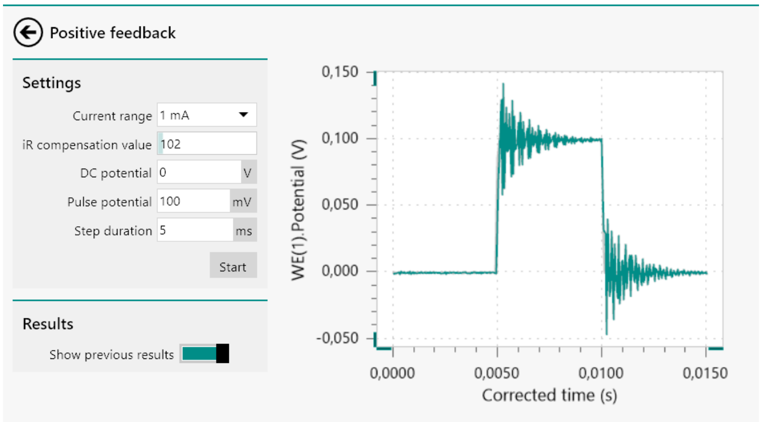


Figure 101 Oscillation is detected when the R_u value is overcompensated

NOTE

Whenever using the iR drop compensation is used in any electrochemical measurement, it is recommended to set the compensated resistance to about 80-90 % of the estimated R_u value.

5.2.2.4 Check cell

The Check cell button can be used to perform a cell check. This tool can be used to evaluate the cell connections, test the stability of the feedback loop and evaluate the noise levels.

During a cell check, five consecutive, high-speed current measurements are carried out at the specified potential or current. The software will evaluate the stability of the feedback loop of the instrument and determine the average value and standard deviation of the measured current or potential. The presented results can be used to assess the quality of the cell connections and the noise levels.

NOTE

The duration of each measurement is determined by the **Power supply frequency** property. With 50 Hz, each measurement will take 20 ms. With 60 Hz, each measurement will take 16.66 ms.

When the check cell tool is used, the control screen for this tool will be displayed *Figure 102*. The control screen provides two panels and one plot area.

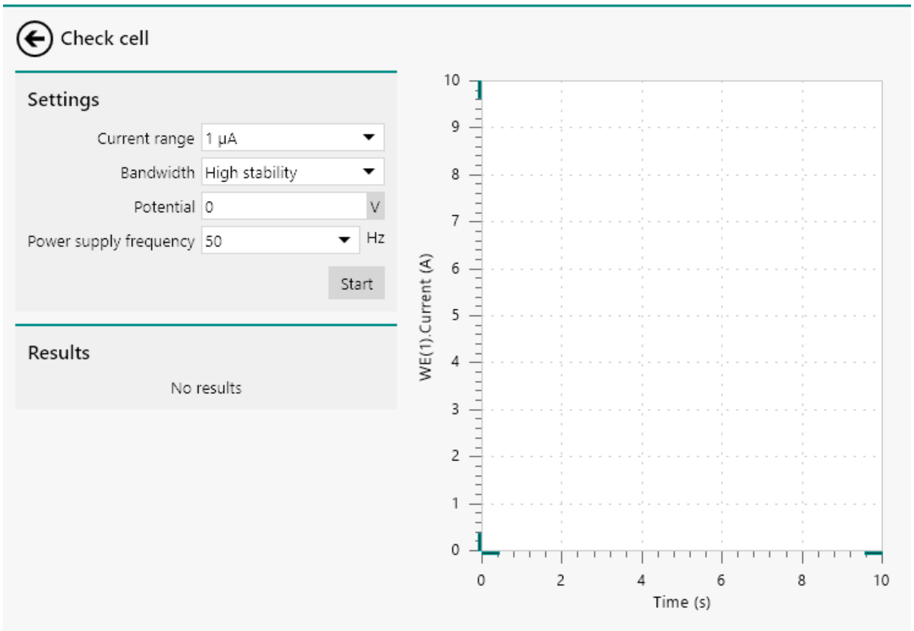


Figure 102 The check cell tool

The **Settings** panel shows the properties using the cell check measurement *(see Figure 103, page 100)*.

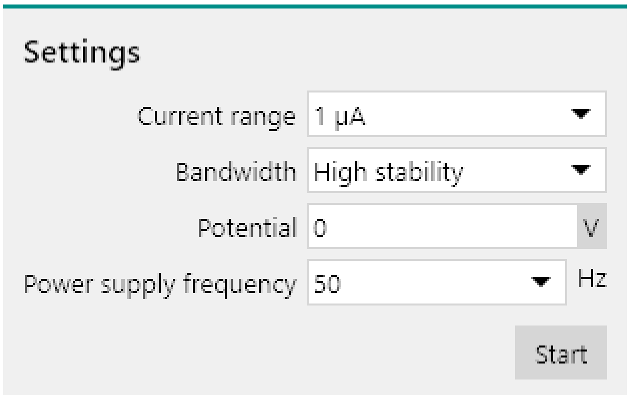


Figure 103 The check cell Settings panel

The following properties and controls are available:

- **Current range:** the current range in which the cell check is performed.
- **Bandwidth:** a drop-down control that can be used to specify the bandwidth of the instrument (high stability, high speed or ultra-high speed).
- **Potential/Current:** a numeric field that can be used to specify the applied potential (in potentiostatic mode) or the applied current (in galvanostatic mode).

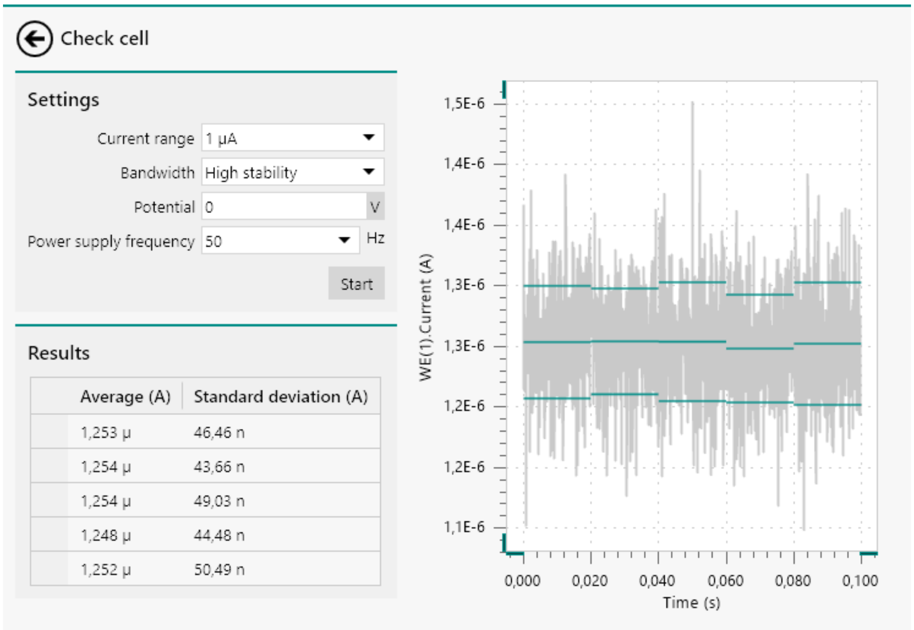
- **Power supply frequency:** specifies if the mains frequency is 50 or 60 Hz.

NOTE

More information on the instrument bandwidth settings can be found in *Chapter 16.1.2.3*.

Clicking the Start button initiates the cell check measurement, using the specified properties.

While the cell check measurement is running, the spinning symbol is shown. The instrument cannot be used until the measurement is finished. When the measurement is finished, the measured data is displayed next to the **Settings** panel and the average and standard deviation of the data is reported in the **Results** panel *(see Figure 104, page 101)*.



Average (A)	Standard deviation (A)
1,253 µ	46,46 n
1,254 µ	43,66 n
1,254 µ	49,03 n
1,248 µ	44,48 n
1,252 µ	50,49 n

Figure 104 The measured data and the results are shown after the measurement

The data plotted in light grey corresponds to the raw current data measured during the cell check. For each of the five consecutive measurements, the average value and the standard deviation is graphically reported in the plot, in dark green.

The check cell tool calculates the average value of the measured standard deviations, $\bar{\sigma}$, and uses this value to evaluate if the measured current noise is within acceptable limits for the used current range, [CR], according to:

$$\frac{\overline{\sigma}}{[CR]}$$

If the ratio exceeds a value of 0.25, a message is shown, indicating that the measured noise is too high *(see Figure 105, page 102)*.

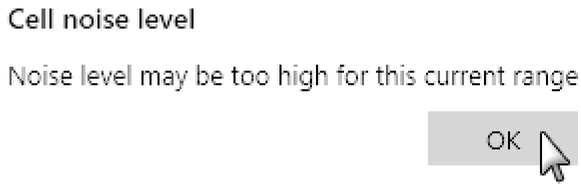


Figure 105 A message is shown if the noise levels are too high

The check cell tool also evaluates the stability of the feedback loop by testing if the measured potential, $E_{measured}$, is within the acceptable limits of the applied potential, $E_{applied}$, according to:

$$\left|E_{measured} - E_{applied}\right| \geq 2\frac{20}{2^{16}} + 0.005$$

If this inequality is true, then the measured potential does not correspond to the applied potential and a message is displayed *(see Figure 106, page 102)*.

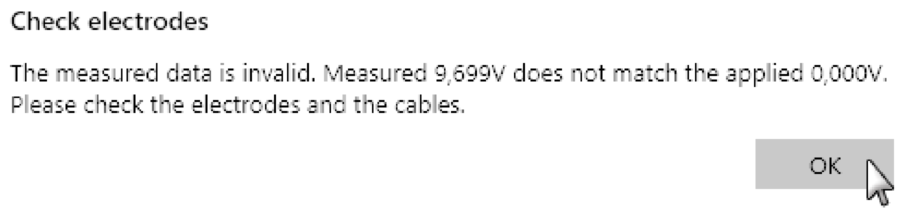


Figure 106 A message is shown if the data is invalid

In this case, the measurement is invalid and no data will be displayed. This problem usually occurs when the connections to the cell are not correct or when the reference electrode is not functional.

NOTE

This test is only carried out when the check cell tool is used in **Potentiostatic** mode.

If the current range can (in case of a current underload) or must be (in case of an overload) optimized, a message will also be displayed at the end of the measurement *(see Figure 107, page 103)*.

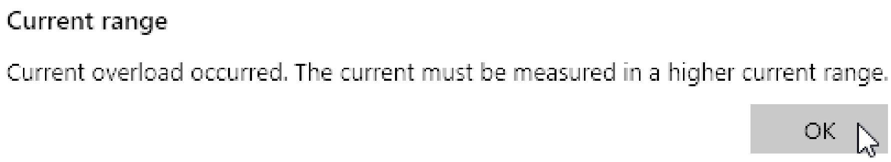


Figure 107 A message is shown if the current range can or must be changed

5.2.2.5 pH calibration

The pH calibration button can be used to calibrate a pH electrode connected to the **pX1000** or **pX** module.

NOTE

This tool is only available for instruments fitted with the optional **pX1000** module or **pX** module *(see Chapter 16.3.2.18, page 1141)*.

CAUTION

The pH sensor **must** be calibrated in a separate cell. Make sure that the working electrode of the Autolab PGSTAT is **not** located in the vessel used for the pH sensor calibration. With the **pX1000** module, grounding of the sensor is performed automatically by the software. For the **pX** module, grounding must be done manually, using the provided 50 Ω resistor BNC shunt connected to the ⊙ G BNC input on the front panel of the module.

When the pH calibration tool is used, the control screen for this tool will be displayed *(see Figure 108, page 104)*. The control screen provides three panels and one plot area.

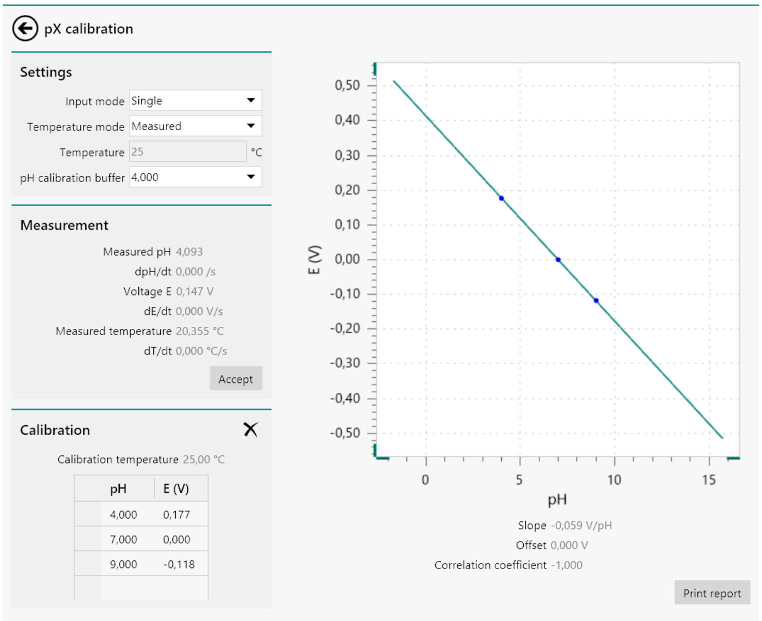


Figure 108 The pH calibration tool

The **Settings** panel shows the properties used for the pH measurement *(see Figure 109, page 104)*.

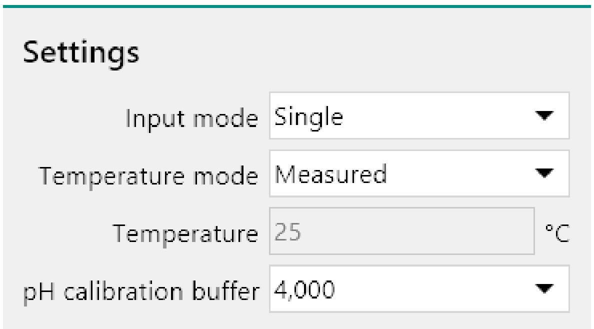


Figure 109 The pH calibration Settings panel

The following properties can be specified using the drop-down lists:

- **Input mode:** defines how the pH electrode is connected (single, differential). This setting depends on the specifications of the pH sensor. For pH sensors fitted with an internal reference electrode, the single input mode is used. For pH sensors using an external reference electrode, the differential mode is used. For all supported Metrohm sensors, the input mode is single.
- **Temperature mode:** defines if the temperature is measured through the pH sensor, if possible, or if the temperature is specified manually. It is only possible to measure the temperature if the pH sensor is fitted with an internal temperature sensor.
- **Temperature:** defines the temperature at which the pH sensor is calibrated, in °C. This value can only be specified if the **Temperature mode** property is set to manual control.

- **pH calibration buffer:** sets the pH value of the calibration buffer. Three predefined pH buffer values are available (4, 7 and 9) but it is possible to specify any buffer value manually.

NOTE

The measurement of the temperature is only available with the **pX1000** module.

The **Measurement** panel shows the real time data measured by the pH sensor *(see Figure 110, page 105)*.

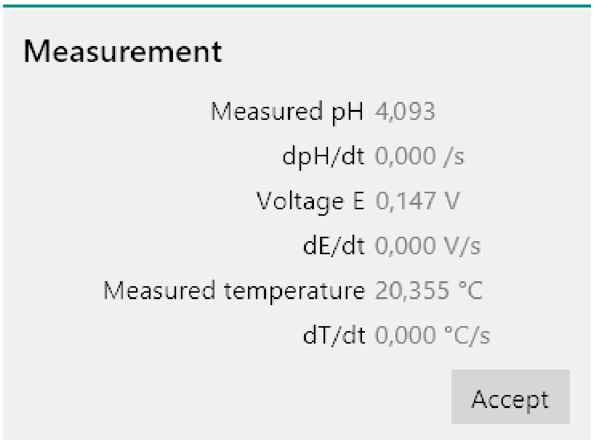


Figure 110 The Measurement panel

The following information is updated in real time:

- **Measured pH:** the current calculated pH value.
- **dpH/dt:** the time derivative value of the measured pH, in (/s).
- **Voltage E:** the voltage measured by the module, in V.
- **dE/dt:** the time derivative value of the measured voltage, in V/s.
- **Measured temperature:** the temperature measured by the module, in °C.
- **dT/dt:** the time derivative value of the measured temperature, in °C/s.

Furthermore, the Accept button is provided to validate a measured value during the calibration process.

The **Calibration** panel shows the calibration data currently used and stored *(see Figure 111, page 106)*.

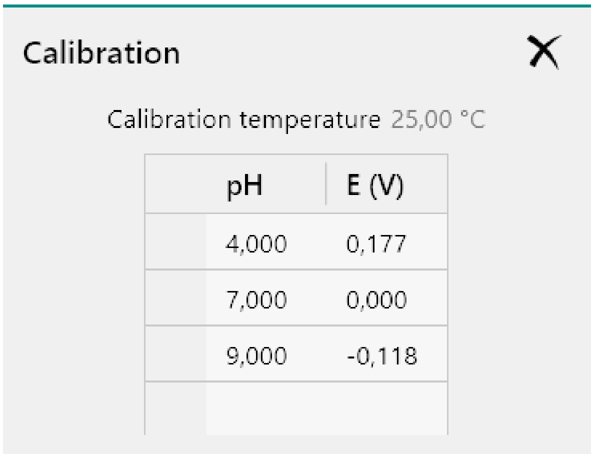


Figure 111 The Calibration panel

Predefined calibration data points are available. These data points are stored on the module (for the pX1000 module) or in a file locally stored on the computer (for the pX module).

Finally, a plot is shown on the right-hand side of the control panels. This plot shows the three stored calibration points and the regression line. Below the plot, the equation of the regression line and the slope is displayed, in V/pH units, as well as the offset and the correlation coefficient.

The Print report button located below the plot. This button can be used to create a printable calibration report *(see Chapter 5.2.2.5.5, page 112)*.

The pH calibration tool can be used to perform the following tasks:

- **Clear all calibration points** *(see Chapter 5.2.2.5.1, page 106)*
- **Add a new calibration point** *(see Chapter 5.2.2.5.2, page 107)*
- **Edit a calibration point** *(see Chapter 5.2.2.5.3, page 111)*
- **Remove a calibration point** *(see Chapter 5.2.2.5.4, page 111)*
- **Save the calibration data** *(see Chapter 5.2.2.5.6, page 114)*

5.2.2.5.1 Remove all previous calibration points

Clicking the X button in the **Calibration** panel removes all calibration points from the table *(see Figure 112, page 106)*.

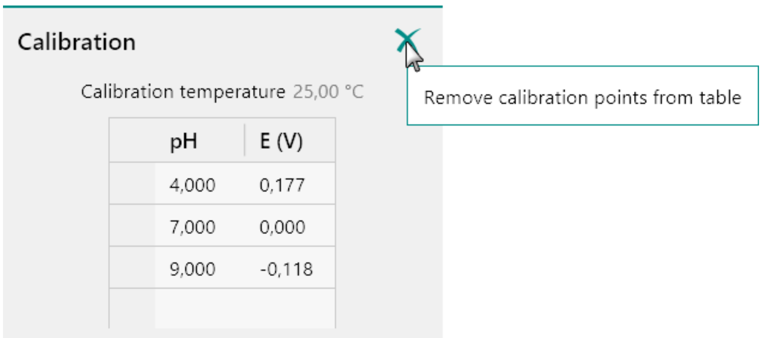


Figure 112 Removing all the calibration data

NOTE

The calibration points are still stored in the on-board memory of the pX1000 module or on the computer for the pX module. The change to the calibration data points is only finalized when the pH calibration tool is closed.

After clearing all the calibration data points, the plot area is also cleared *(see Figure 113, page 107)*.

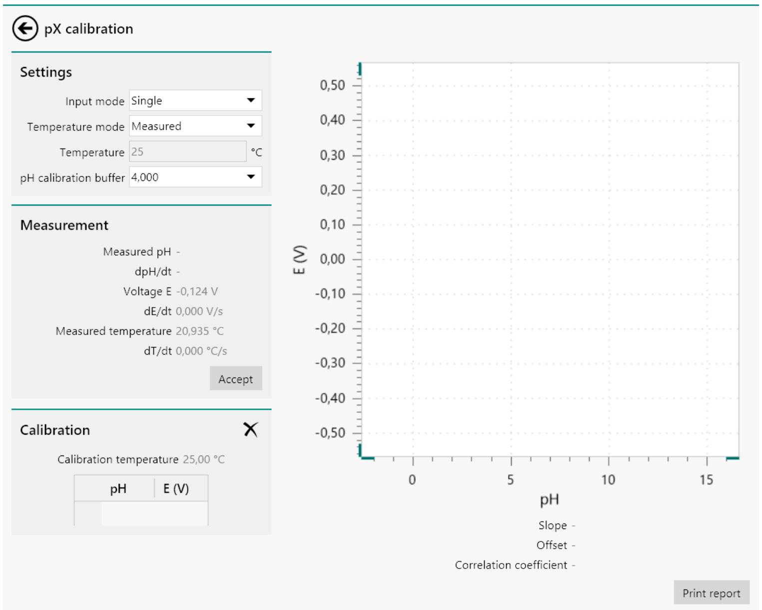


Figure 113 The cleared plot

5.2.2.5.2 Adding calibration points

The calibration of a pH sensor requires at least two data points. It is possible to add data points to the calibration data in two different ways:

- By manually adding data points to the calibration data.
- By measuring the pH of a buffer of known pH value with the connected pH sensor.

To manually add data points to the calibration data, click the first available cell in the **Calibration** panel and directly type the pH and corresponding potential value *(see Figure 114, page 108)*.

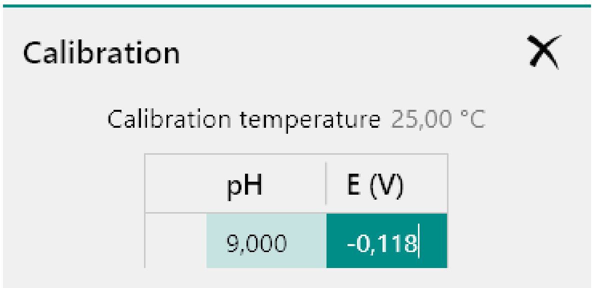


Figure 114 Manually adding calibration data

To add a measured data point to the calibration data, select the required pre-defined pH buffer value from the drop-down list provided in the **Settings** panel *(see Figure 115, page 108)*.

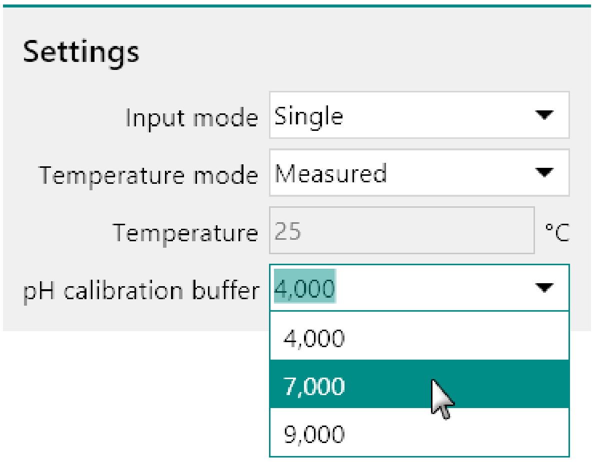


Figure 115 Selecting the pH buffer value

It is also possible to directly type the pH value of the buffer in the **Settings** panel *(see Figure 116, page 108)*.

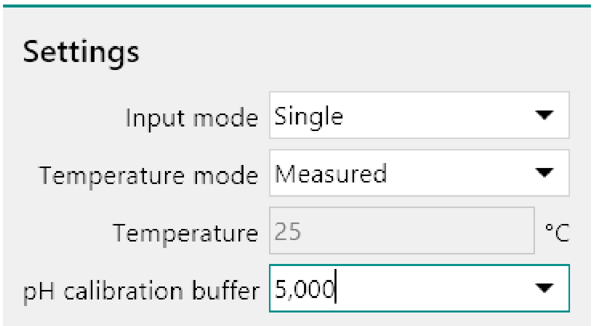


Figure 116 Manually specifying the pH buffer value

With the buffer value specified in the **Settings** panel, click the Accept button in the **Measurement** panel when a stable and correct voltage is displayed *(see Figure 119, page 110)*.

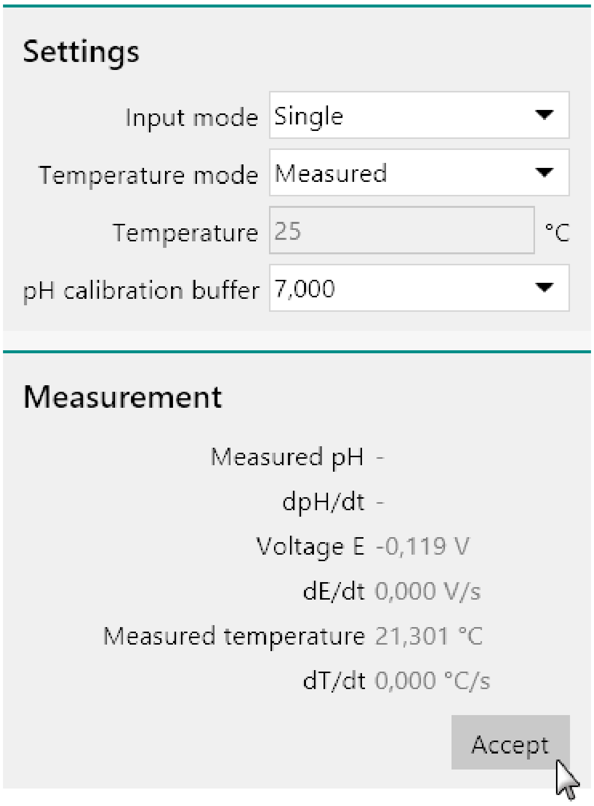


Figure 117 Accepting a calibration point

As soon as the **Accept** button is clicked, the measured value and the specified buffer value are added to the calibration data *(see Figure 118, page 110)*.

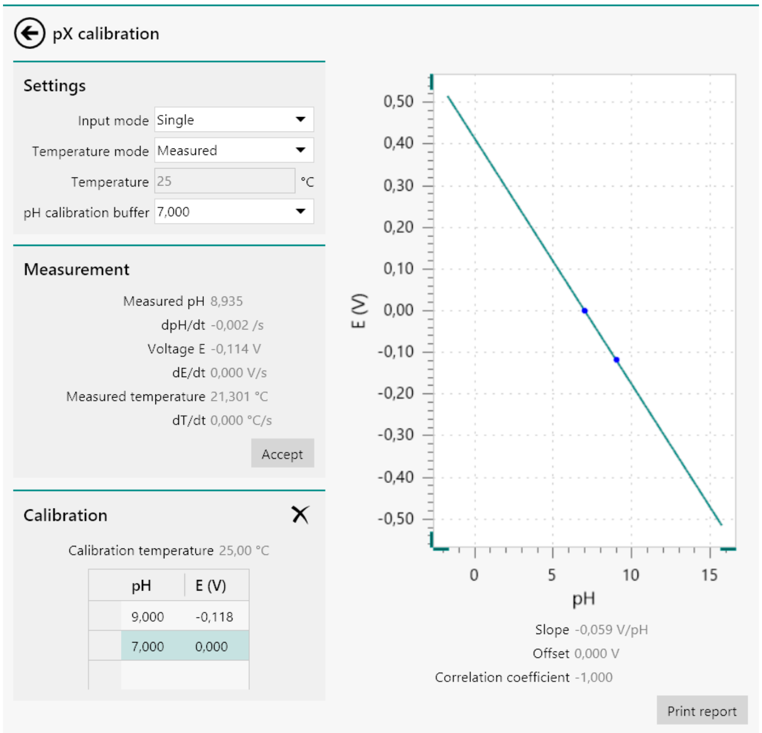


Figure 118 The updated calibration data

NOTE

The calibration data is automatically plotted on the right-hand side of the **Settings** panel when two or more calibration data points are specified.

It is possible to add more calibration points, using the method described above.

If the temperature is measured using the built-in temperature sensor, a validation message may be shown when accepting a new value if the temperature at which this new data point is measured differs by more than 0.5 °C from the existing calibration data *(see Figure 119, page 110)*.

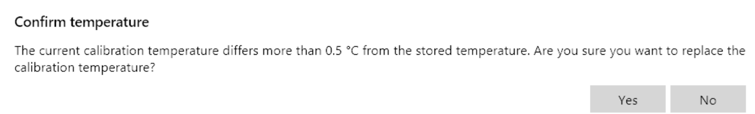


Figure 119 A warning is shown when the temperature deviates by more than 0.5 °C

Clicking the Yes button validates the new data point despite the temperature difference. Clicking the No button cancels the validation of the new calibration point.

5.2.2.5.3 Editing a calibration point

It is possible to manually adjust a measured value. To do this, click a value in the **Calibration** panel and manually edit the value in the table *(see Figure 120, page 111)*.

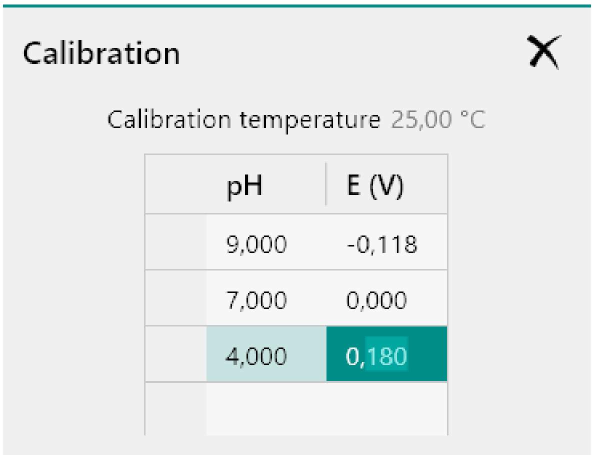


Figure 120 Editing a calibration point

Click away from the value or press the **[Enter]** key on the keyboard to validate the change to the value *(see Figure 121, page 111)*.

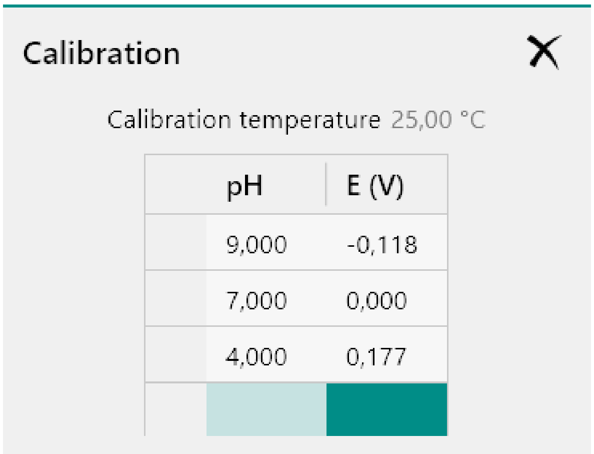


Figure 121 Validating the edited point

5.2.2.5.4 Removing calibration points

It is possible to remove points from the calibration data. To do this, click the row index cell in the **Calibration** panel to select the whole row *(see Figure 122, page 112)*.

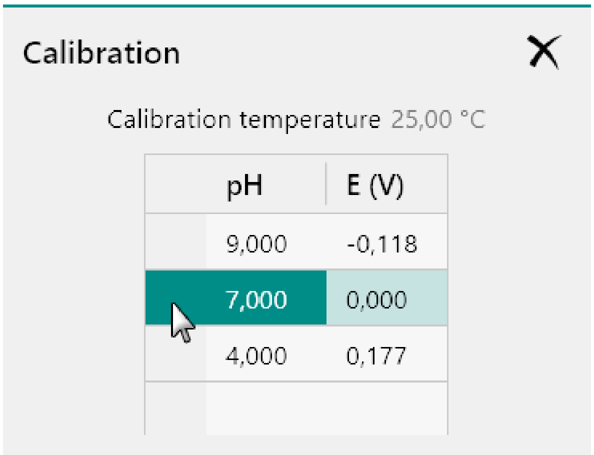


Figure 122 Removing a calibration point

Press the **[Delete]** key on the keyboard to remove the complete row from the calibration data *(see Figure 123, page 112)*.

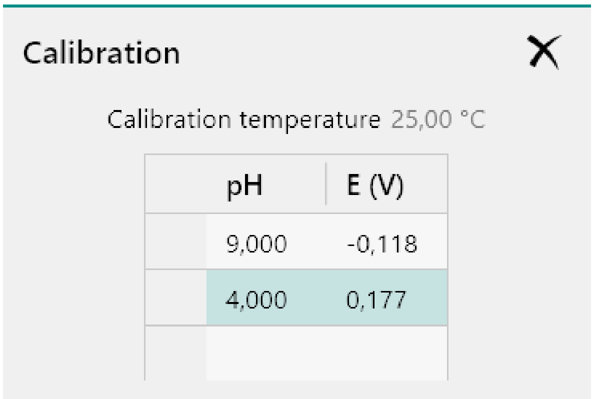


Figure 123 The calibration point is removed

5.2.2.5.5 Printing calibration report

When the calibration is complete, it is possible to generate a printable report for bookkeeping purposes. To do this, click the Print report button located below the plot area *(see Figure 124, page 113)*.

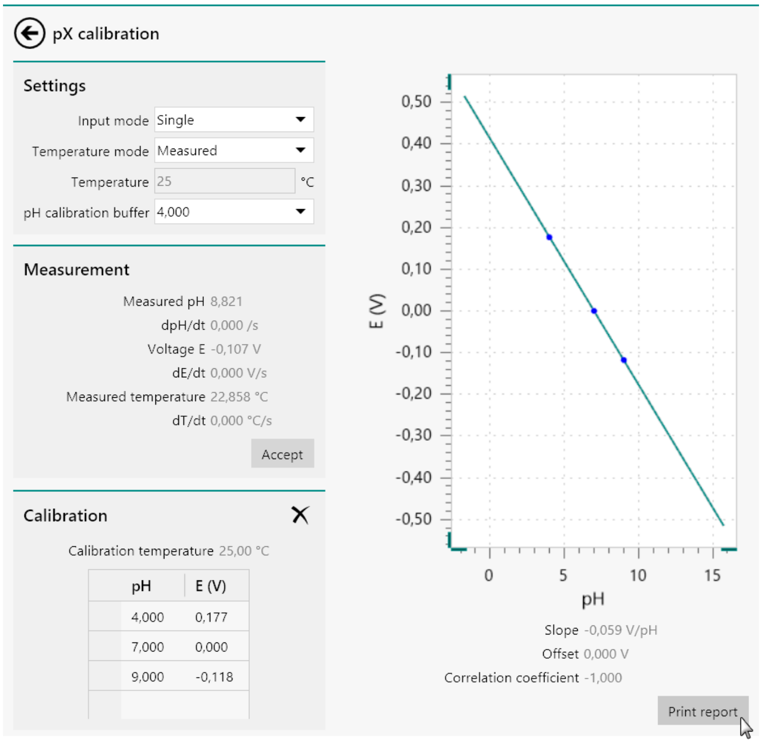


Figure 124 Generating a calibration report

A report will be generated *(see Figure 125, page 114)*.

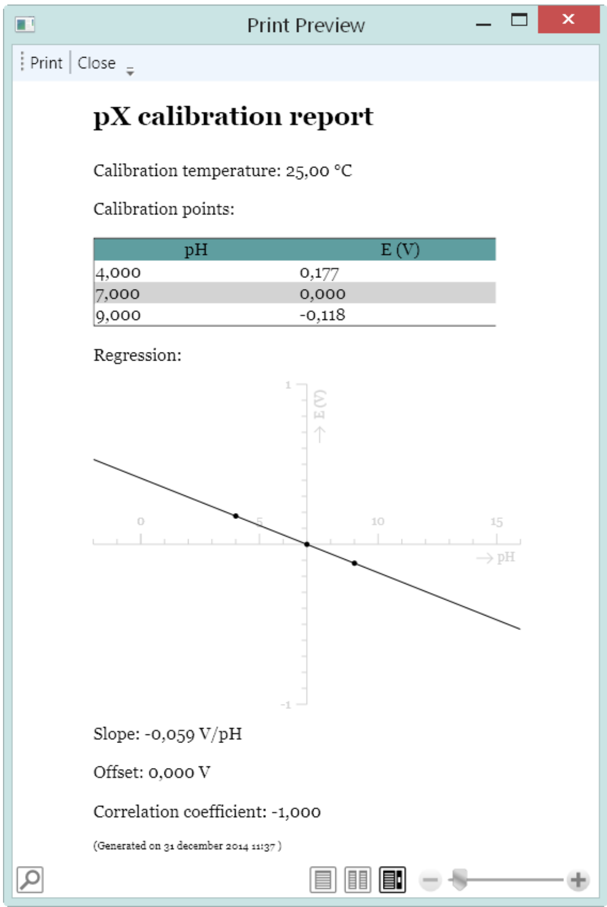


Figure 125 The calibration report

This report contains all the calibration data and regression data, as well as the date of the report.

5.2.2.5.6 Saving calibration data

When the pH calibration is finished, close the pH calibration tool. If the calibration data was modified, you will be prompted to save or discard the data *(see Figure 126, page 114)*.

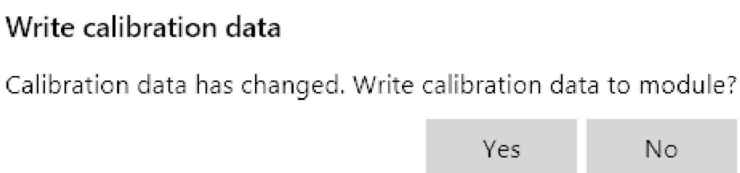


Figure 126 The data can be saved when closing the pH calibration tool

Saving the data will overwrite the existing calibration data stored in the on-board memory of the **pX1000** module or in the calibration file stored on the computer. The previous data can no longer be used after it is overwritten.

5.2.2.6 Reset integrator drift

The Reset integrator drift button can be used to determine and reset the drift of the integrator *(see Figure 127, page 115)*. The integrator drift, in C/s, is the measured charge that accumulates due to the background current. This option will record the drift for the active current range and apply a drift correction for any subsequent measurements involving the integrator.

NOTE

This tool is only available for instruments with an **on-board analog integrator** or for instruments fitted with the optional **FI20** module *(see Chapter 16.3.2.11, page 1061)*.

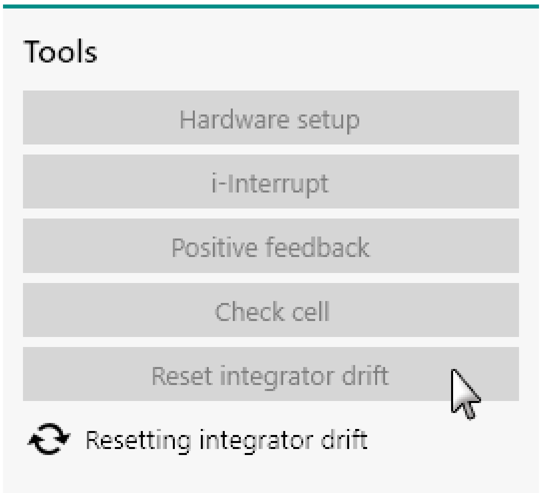


Figure 127 Resetting the integrator drift

While the integrator drift is being measured and reset, the spinning symbol is shown *(see Figure 127, page 115)*. The instrument cannot be used until the drift determination is finished.

NOTE

The determination of the integrator drift can be performed with the electrochemical cell connected to the instrument. The measurement does not affect the connected cell.

> **NOTE**
>
> It is recommended to reset the integrator drift each time a current range is changed.

5.2.3 Autolab display panel

The **Autolab display** panel, shown in *Figure 128*, provides basic manual control of the Autolab and the extension modules that can be manually controlled.

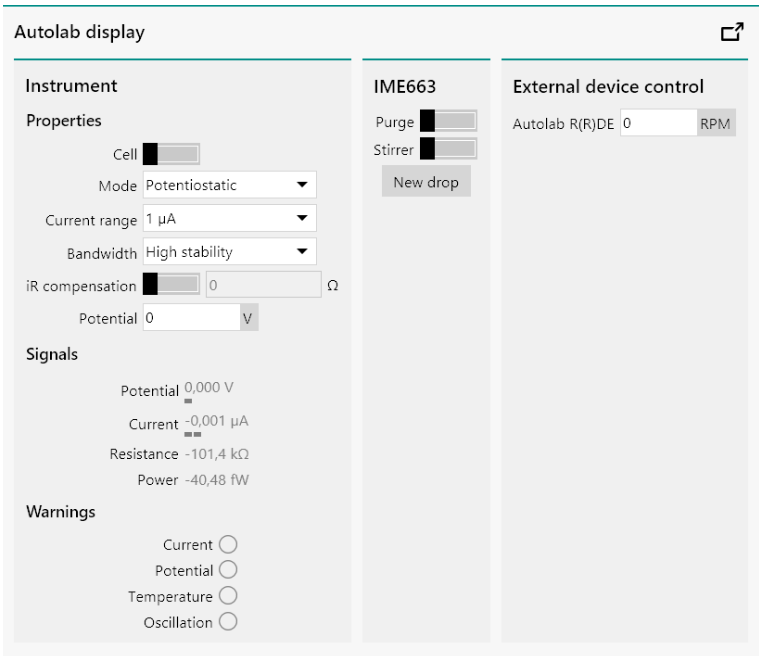


Figure 128 The Autolab display panel

> **NOTE**
>
> The actual content of the **Autolab display** panel depends on the hardware setup. In *Figure 128* manual control of the IME663 module and the R(R)DE electrode is available.

The **Autolab display** panel always shows the **Instrument** panel at the left-most position. This panel provides manual control of the Autolab potentiostat/galvanostat as well as an overview of the real-time values measured by the instrument.

The information is provided in three panels:

- **Properties:** this sub-panel provides basic controls of the Autolab potentiostat/galvanostat.
- **Signals:** this sub-panel provides an overview of the main instrument signals and noise levels observed on these signals. The values are updated in real-time.
- **Warnings:** this sub-panel provides an overview of instrument warnings in real-time.

5.2.3.1 Instrument Properties sub-panel

The **Instrument Properties** sub-panel provides direct control of the Autolab potentiostat/galvanostat *(see Figure 129, page 117)*.

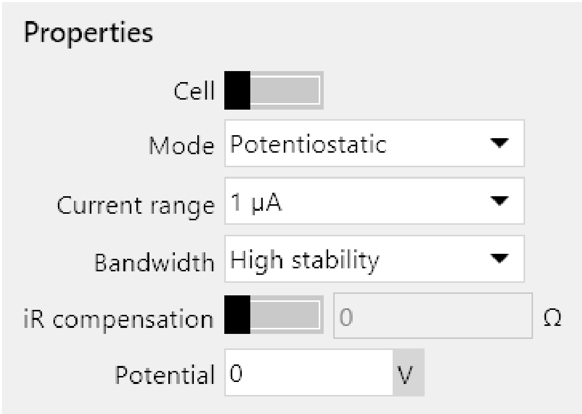


Figure 129 The Instrument Properties sub-panel

The **Instrument Properties** sub-panel provides the following controls of the Autolab instrument:

- **Cell:** a toggle that can be used to switch the cell on or off.
- **Mode:** a drop-down control that can be used to specify the operation mode of the instrument (potentiostatic or galvanostatic).
- **Current range:** a drop-down control that can be used to specify the active current range of the instrument.
- **Bandwidth:** a drop-down control that can be used to specify the bandwidth of the instrument (high stability, high speed or ultra-high speed).
- **iR compensation:** a toggle and a numeric field that can be used to switch the iR compensation circuit on or off and to specify the compensated resistance, in Ω.
- **Potential/Current:** a numeric field that can be used to specify the applied potential (in potentiostatic mode) or the applied current (in galvanostatic mode).

NOTE

The iR compensation control is only available in potentiostatic mode.

5.2.3.2 Instrument Signals sub-panel

The **Instrument Signals** sub-panel provides the real time values measured by the Autolab *(see Figure 130, page 118)*.

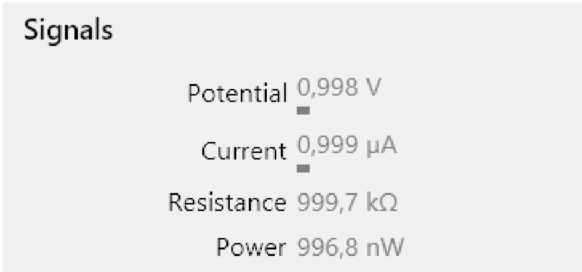


Figure 130 The Instrument Signals sub-panel

The following signals are shown in the **Instrument Signals** sub-panel:

- **Potential:** the potential difference, in V, measured between the reference electrode (RE) and the sense electrode (S) or between the reference electrode (RE) and the working electrode (WE), depending on the type of instrument. This is a measured value and the noise level for this signal is reported using the bars located below the value.
- **Current:** the current, in A, flowing between the counter electrode (CE) and the working electrode (WE). This is a measured value and the noise level for this signal is reported using the bars located below the value.
- **Resistance:** the resistance value of the cell, calculated from the Potential and Current values, in Ω.
- **Power:** the power value of the cell, calculated from the Potential and Current values, in W.

The measured values reported in this sub-panel are shown with a noise level, represented by a number of small bars (between 0 and 8 bars). The bars are determined from the standard deviation, σ. *Table 4* shows the correspondence between noise bars and the standard potential of potential and current (where [CR] is the current range of the instrument).

Table 4 Overview of the noise bars

Number of bars	Potential noise	Current noise
0	$\sigma = 0$	$\sigma = 0$
1	$\sigma < \frac{0.5}{2^7}$	$\sigma < \frac{0.5[CR]}{2^7}$

Number of bars	Potential noise	Current noise
2	$\sigma < \frac{0.5}{2^6}$	$\sigma < \frac{0.5[CR]}{2^6}$
3	$\sigma < \frac{0.5}{2^5}$	$\sigma < \frac{0.5[CR]}{2^5}$
4	$\sigma < \frac{0.5}{2^4}$	$\sigma < \frac{0.5[CR]}{2^4}$
5	$\sigma < \frac{0.5}{2^3}$	$\sigma < \frac{0.5[CR]}{2^3}$
6	$\sigma < \frac{0.5}{2^2}$	$\sigma < \frac{0.5[CR]}{2^2}$
7	$\sigma < \frac{0.5}{2^1}$	$\sigma < \frac{0.5[CR]}{2^1}$
8	$\sigma \geq \frac{0.5}{2^1}$	$\sigma \geq \frac{0.5[CR]}{2^1}$

5.2.3.3 Instrument Warnings sub-panel

The **Instrument Warnings** sub-panel provides the real time instrument warnings *(see Figure 131, page 119)*.

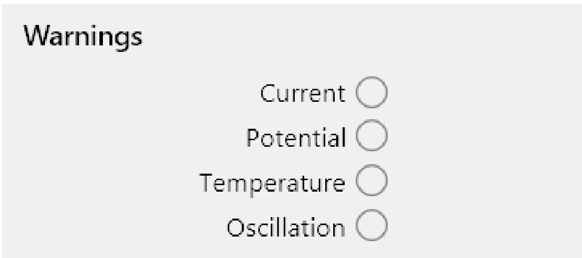


Figure 131 The Instrument Warning sub-panel

The following warnings can be displayed are displayed in the **Instrument Warnings** sub-panel:

- **Current:** this indicator will be lit when a current overload is detected. The current overload warning will be triggered whenever the measured current exceeds the measurable range of the active current range.
- **Potential:** this indicator will be lit when a potential overload is detected. The potential overload warning will be triggered whenever the output potential of the instrument reaches the compliance voltage limit.
- **Temperature:** this will be lit when a temperature overload is detected. The temperature overload warning will be triggered whenever the operating temperature of the instrument exceeds the maximum allowed value.

NOTE

The **PGSTAT204** and **M204** module are fitted with an unrecoverable temperature overload circuit. When this warning is triggered, the instrument needs be switched off completely in order to recover from the temperature overload.

- **Oscillation:** this indicator will be lit when the feedback loop cannot be regulated properly and oscillation is detected.

NOTE

When the oscillation warning is triggered, the cell is automatically switched off for safety reasons. On instruments fitted with a **Cell enable** button on the front panel, this button must be engaged in order to recover from the oscillation warning.

5.2.3.4 Docking and undocking Autolab display panels

For convenience, it is possible to undock the **Autolab display** panel and display the its content in a separate window. To do this, click the button in the top right corner of the **Autolab display** panel *(see Figure 132, page 120)*.

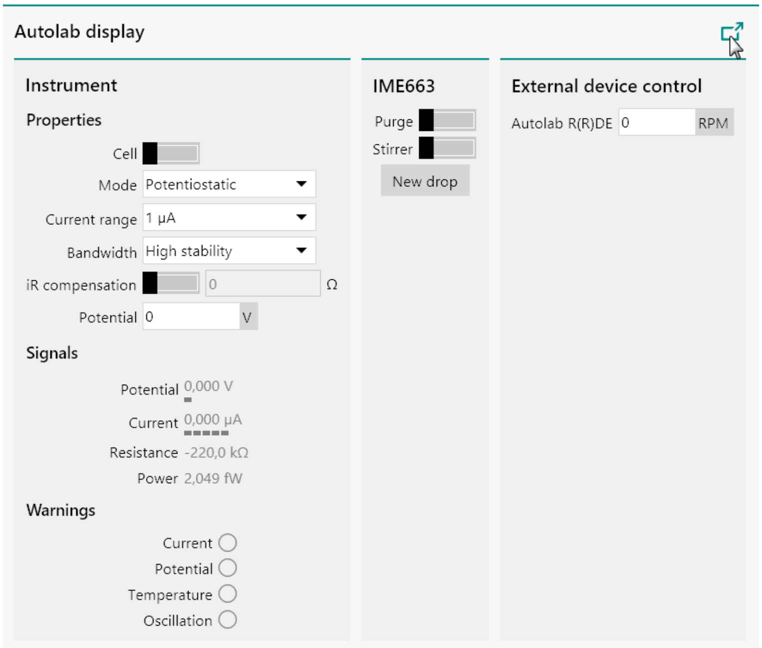


Figure 132 Undocking the Autolab display panel

The content of the **Autolab display** panel will be duplicated in a new window on top of the main NOVA software window. This new window can be moved next to the main NOVA window, or to another computer display if available *(see Figure 133, page 121)*.

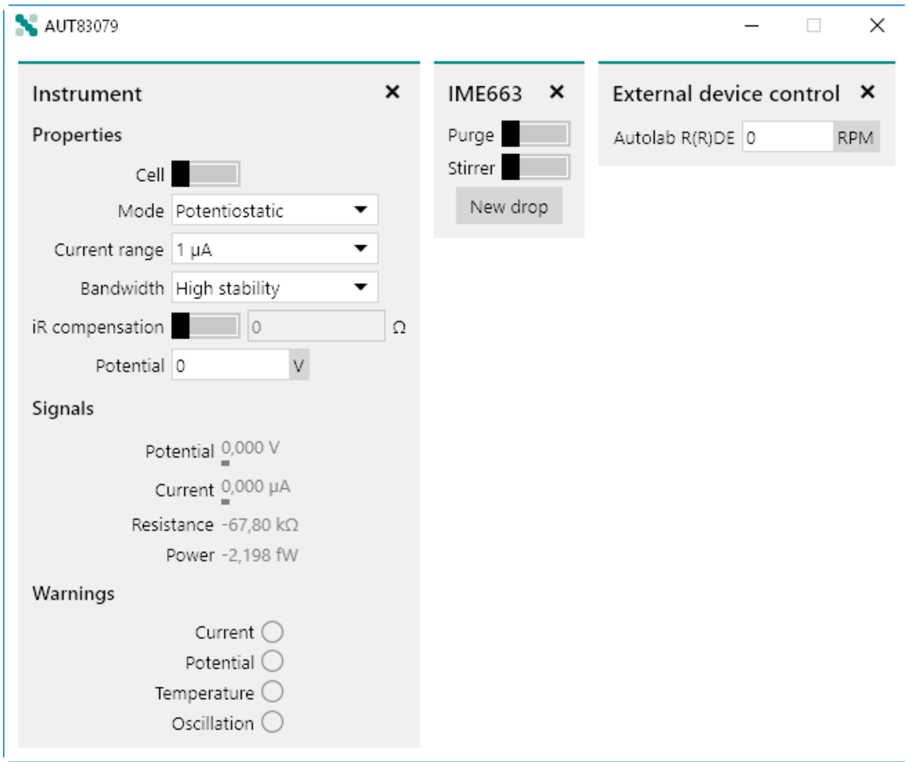


Figure 133 The undocked Autolab display panel window

NOTE

The undocked window is always identified by the serial number of the instrument.

NOTE

It is possible to undock the **Autolab display** panel as many times as required. It is also possible to have undocked **Autolab display** panels from different instruments undocked at any time.

Unnecessary parts of the undocked **Autolab display** panel can be closed by clicking the ✖ button in the top right part of each sub-panel *(see Figure 134, page 122)*.

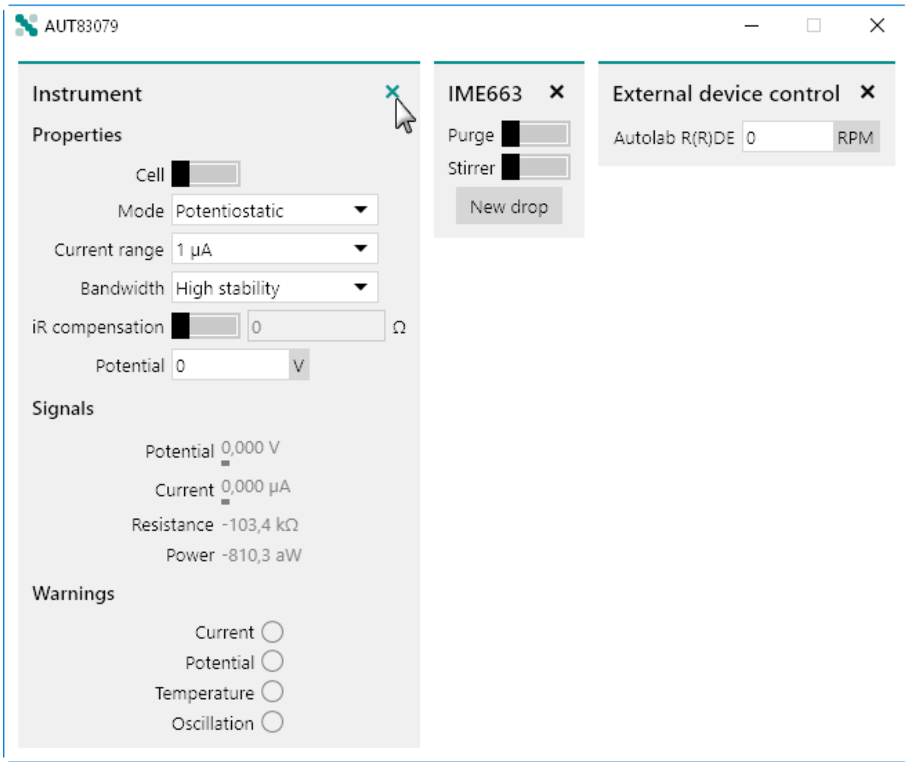


Figure 134 Closing parts of the undocked Autolab display panel window

The closed sub-panel will be removed from the window and the window will be resized, if applicable *(see Figure 135, page 122)*.

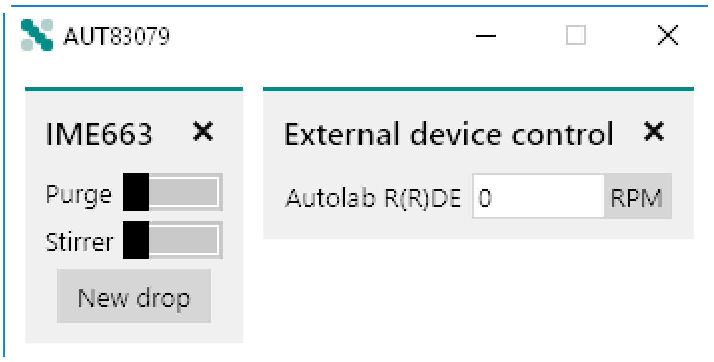


Figure 135 The updated Autolab display panel window

5.3 Autolab RHD Microcell HC control panel

Double clicking an Autolab RHD Microcell HC tile in the **Instruments** panel opens the **Autolab RHD Microcell HC control** panel in a new **tab**, as shown in *Figure 136*.

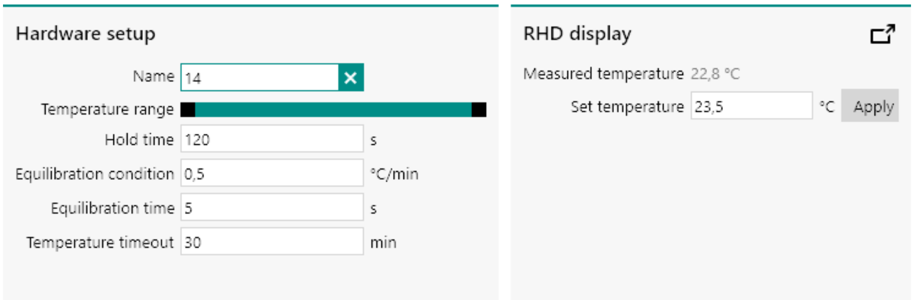


Figure 136 The Autolab RHD Microcell HC control panel

The **Autolab RHD Microcell HC control** panel has two sub-panels:

- **Hardware setup panel:** this panel displays instrument settings for the Autolab RHD Microcell HC controller.
- **RHD display panel:** this panel provides manual control of the Autolab RHD Microcell HC controller.

5.3.1 Autolab RHD Microcell HC hardware setup

The **Hardware setup** panel of the Autolab RHD Microcell HC can be used to specify the settings of the Autolab RHD Microcell HC controller *(see Figure 137, page 123)*.

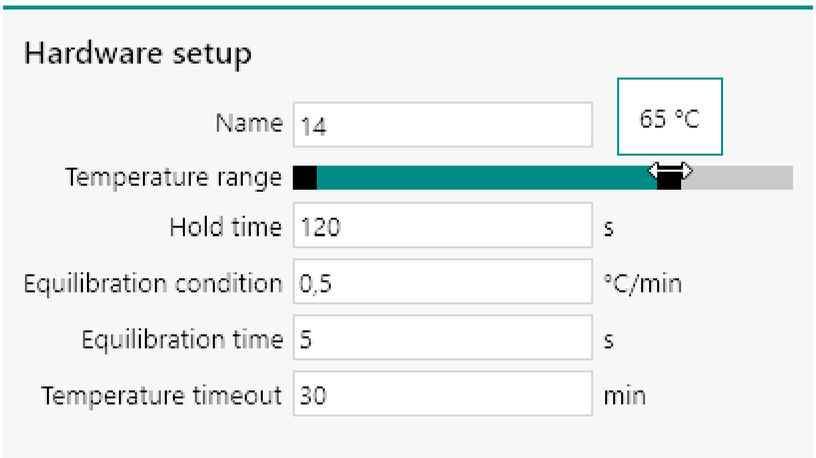


Figure 137 The Hardware setup panel

The following properties are available:

- **Name:** an input field which can be used to give a dedicated name to the instrument. By default, the name of the instrument corresponds to the three digits of the instrument serial number.

- **Temperature range:** this slider can be used to specify the minimum and maximum allowed temperature for the Autolab RHD Microcell HC controller. The default values are -50 °C and 100 °C by default. By clicking and dragging the black ends of the slider, the minimum and maximum temperature can be adjusted, as shown in *Figure 137*.
- **Hold time:** specifies a holding time to use during a measurement after the temperature of the Autolab RHD Microcell HC controller has stabilized. The default value is 120 s.
- **Equilibration condition:** specifies the minimum value of the first derivative of the temperature versus time to reach a stable temperature. The default value is 0.5 °C/min.
- **Equilibration time:** specifies the minimum time during which the equilibration condition must be valid in order to consider the temperature of the Autolab RHD Microcell HC controller stable, in s.
- **Temperature timeout:** specifies a maximum time which is allowed to pass for the Autolab RHD Microcell HC controller to stop adjusting the temperature, in min.

The temperature regulation of the Autolab RHD Microcell HC controller works in the following way:

1. After setting the new temperature, the actual temperature is measured.
2. The temperature is considered stable when the derivative of the temperature versus time is smaller than the **Equilibration condition** for a duration equal or longer than the **Equilibration time**.
3. If no stable temperature can be reached, the controller will stop regulating the temperature after the specified **Temperature timeout**.

5.3.2 Autolab RHD Microcell HC manual control panel

The **Manual control** panel of the Autolab RHD Microcell HC can be used to read the current temperature of the controller and can be used to set a new temperature of the controller *(see Figure 138, page 124)*.

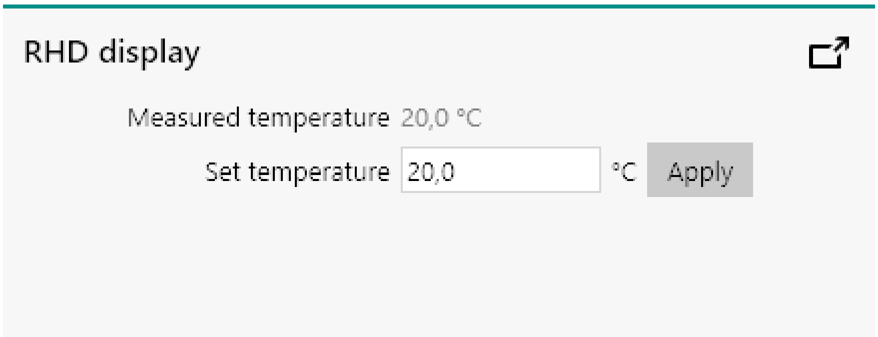


Figure 138 The Autolab RHD Microcell HC Manual control panel

The following properties are available in the **Manual control** panel:

- **Measured temperature:** this read-only value shows the actual temperature of the Autolab RHD Microcell HC controller, in °C.
- **Set temperature:** specifies the new temperature of the Autolab RHD Microcell HC controller, in °C.

To set the temperature of the Autolab RHD Microcell HC controller, specify the value in the **Manual control** panel and click the Apply button *(see Figure 139, page 125)*.

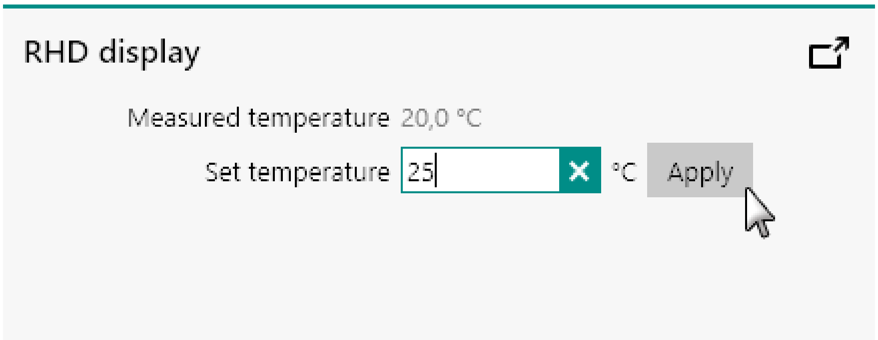


Figure 139 Using the manual control of the Autolab RHD Microcell HC

The new temperature will be set. It is possible to undock the **Manual control** panel by clicking the button. The **Manual control** panel will be undocked in new window *(see Figure 140, page 125)*.

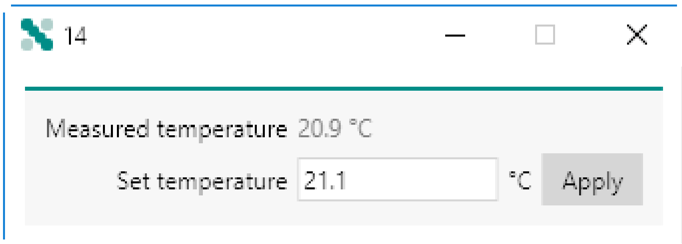


Figure 140 The Manual control panel can be undocked

5.4 Autolab Spectrophotometer control panel

Double clicking an Autolab Spectrophotometer (or Avantes Spectrophotometer) tile in the **Instruments** panel opens the **Spectrophotometer control** panel in a new **tab**, as shown in *Figure 141*.

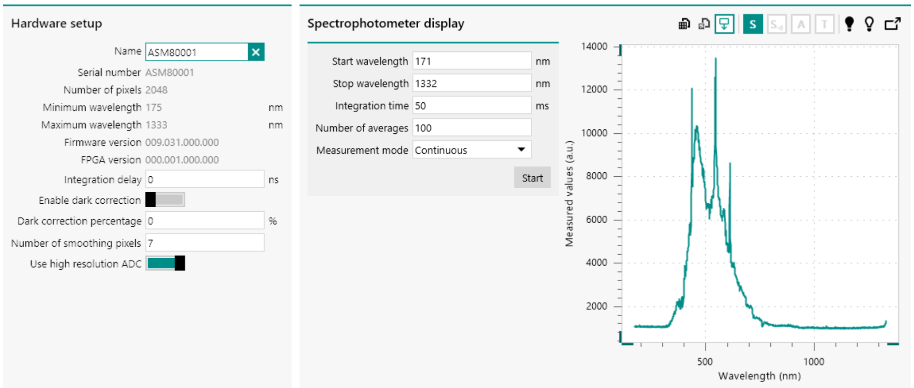


Figure 141 The Spectrophotometer control panel

The **Autolab Spectrophotometer control** panel has two sub-panels:

- **Hardware setup panel:** this panel displays instrument settings for the Autolab Spectrophotometer.
- **Spectrophotometer display panel:** this panel provides manual control of the Autolab Spectrophotometer.

5.4.1 Autolab Spectrophotometer hardware setup

The configuration of the connected **Spectrophotometer** can be adjusted in the **Hardware setup** panel *(see Figure 142, page 126)*.

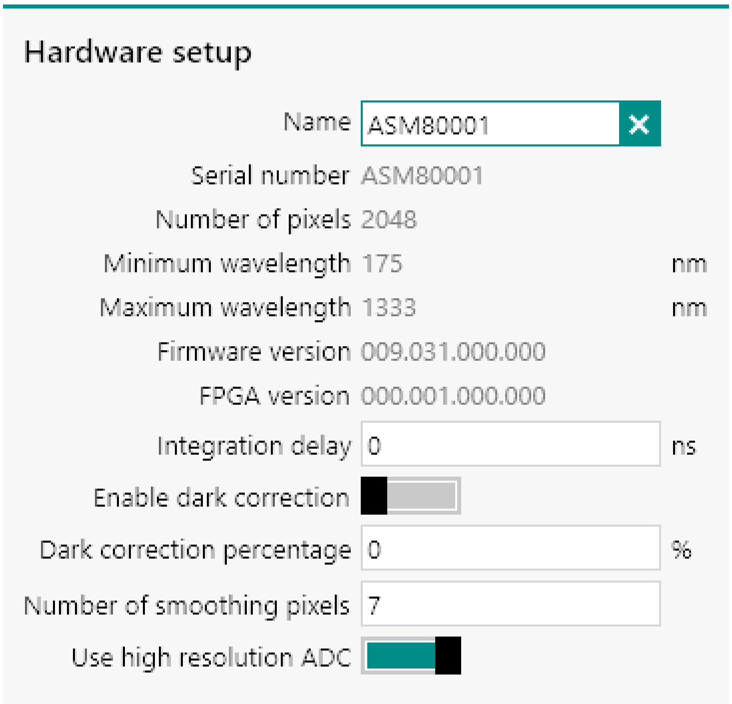


Figure 142 The Spectrophotometer Hardware setup panel

The **Hardware setup** panel displays information or properties of the connected Spectrophotometer. The following properties are available:

- **Name:** an input field which can be used to give a dedicated name to the instrument. By default, the name of the instrument corresponds to the instrument serial number.
- **Serial number:** a read-only field that provides the serial number of the instrument.
- **Number of pixels:** a read-only field that provides the number of pixels of the detector of the instrument.
- **Minimum wavelength:** a read-only field that provides the lowest measurable wavelength of the detector the instrument.
- **Maximum wavelength:** a read-only field that provides the highest measurable wavelength of the detector the instrument.
- **Firmware version:** a read-only field that provides the firmware version of the instrument.
- **FPGA version:** a read-only field that provides the FPGA version of the instrument.
- **Integration delay:** an input field which can be used to specify the integration delay in ms.
- **Enable dark correction:** a toggle which can be used to enable or disables the dark correction (default OFF).
- **Dark correction percentage:** an input field which can be used to specifies the percentile value of dark correction (0-100 %).
- **Number of smoothing pixels:** an input field which can be used to specify the number of pixels used in the smoothing algorithm. When this value is set to 0, no smoothing is used. The optimal value depends on the fiber diameter, pixel size and type of spectrophotometer.
- **Use high resolution ADC:** a toggle which can be used to enables or disables the high resolution ADC of the spectrometer. When enabled, the measured values are resolved using a 16 Bit ADC, when disabled a 14 Bit ADC is used instead (default ON).

NOVA supports all Autolab Spectrophotometers and all Avantes USB 2.0 AvaSpec Spectrophotometers with a suitable firmware installed. The following firmware versions are supported:

- **000.031.000.000** or **009.031.000.000:** these two versions of the firmware support all options provided in NOVA.
- **009.028.000.000:** this firmware version supports all options provided in NOVA except spectrum averaging. When this firmware is detected, a warning symbol is shown in the **Hardware setup** panel, with an indication that an outdated firmware is detected, as shown in *Figure 143*.

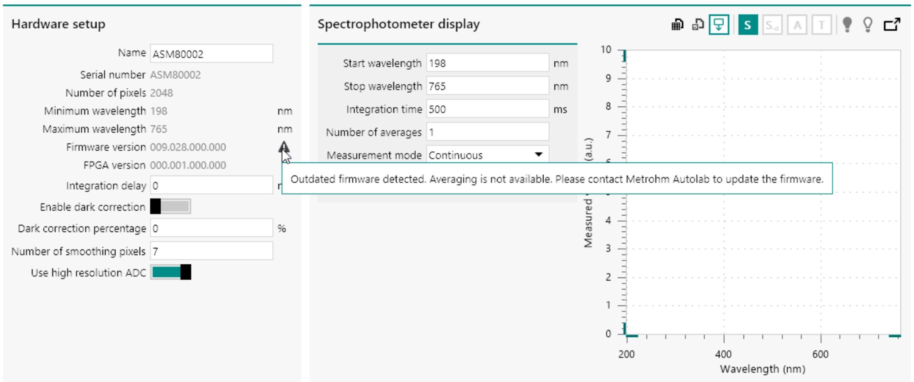


Figure 143 A warning is shown when an outdated firmware is detected

- **Other versions:** all other firmware versions are not supported in NOVA. When an unsupported firmware is detected, a warning error symbol is shown in the **Hardware setup** panel, with an indication that an unsupported firmware is detected, as shown in *Figure 144*. In this case, the spectrophotometer cannot be used.

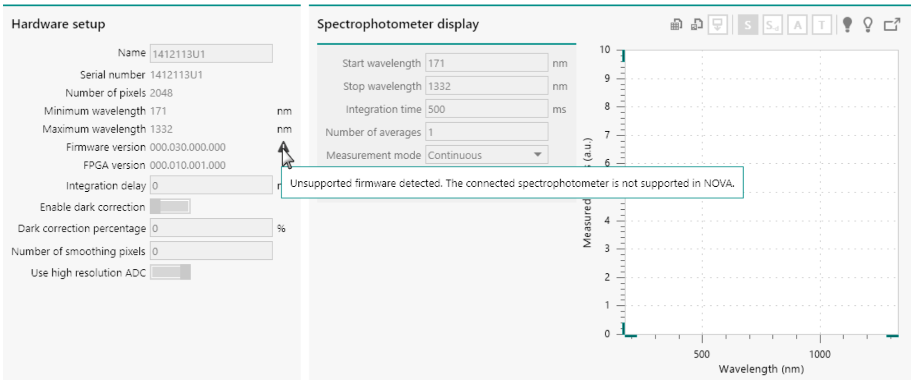


Figure 144 An error is shown when an unsupported firmware is detected

NOTE

Please contact Metrohm Autolab for information on the update process of the installed firmware.

Table 5 provides an overview of the optimal number of **Smoothing pixels** for the Autolab spectrophotometers in function of the fiber diameter.

Table 5 Optimal smooth pixel settings for the different optical fiber diameters

Fiber diameter (µm)	Optimal smoothing pixels
10	0

Fiber diameter (µm)	Optimal smoothing pixels
25	1
50	2
100	3
200	7
400	14
500	17
600	21

NOTE

For information on the optimal number of **Smoothing pixels** for compatible Avantes spectrophotometer, please refer to the Avantes user manual

5.4.2 Autolab Spectrophotometer manual control panel

The **Spectrophotometer display** panel provides manual control of the connected Autolab (or Avantes) spectrophotometer *(see Figure 145, page 129)*.

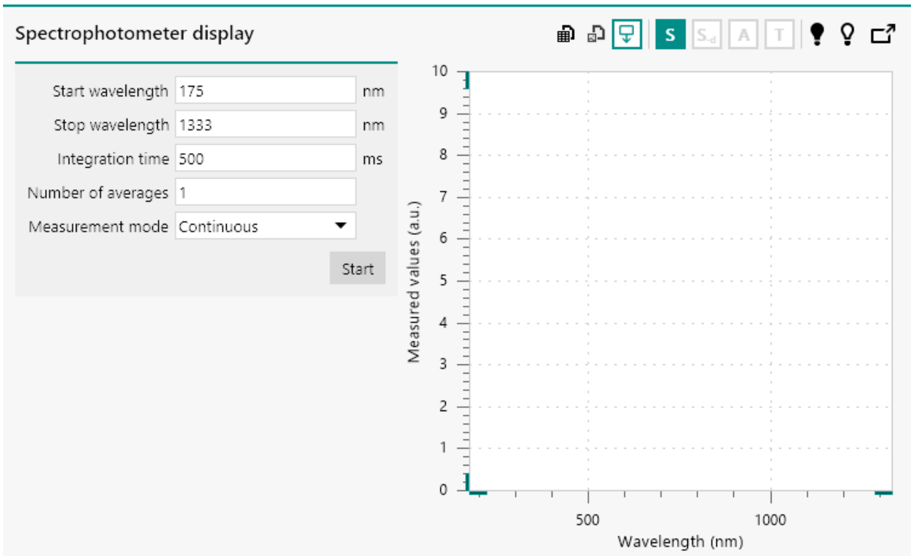


Figure 145 The Spectrophotometer display panel

The following properties are available *(see Figure 146, page 130)*:

- **Start wavelength:** an input field which can be used to specify the start wavelength used in the measurement, in nm.

- **Stop wavelength:** an input field which can be used to specify the stop wavelength used in the measurement, in nm.
- **Integration time:** an input field which can be used to specify the integration time, in ms. The smallest integration time depends on the type of spectrophotometer used. For the Autolab Spectrophotometer instruments, the smallest possible value is 1.05 ms.
- **Number of averages:** an input field which can be used to specify the number of averages, as an integer.
- **Measurement mode:** a drop-down control that can be used to specify the measurement mode (continuous or single). In continuous mode, the spectrophotometer will acquire spectra until stopped by the user. In single mode, the spectrophotometer will acquire a single spectrum.

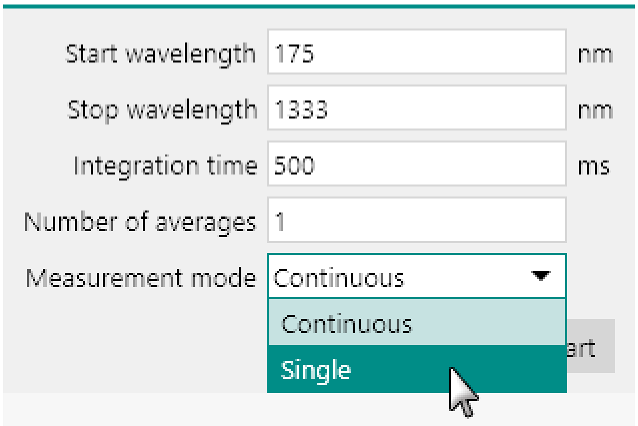


Figure 146 The measurement properties

To start the acquisition of a spectrum, the Start button can be pressed. Depending on the Measurement mode property, the spectrophotometer will acquire one or more spectra and display the measured data in the plot on the right hand side *(see Figure 147, page 131)*.

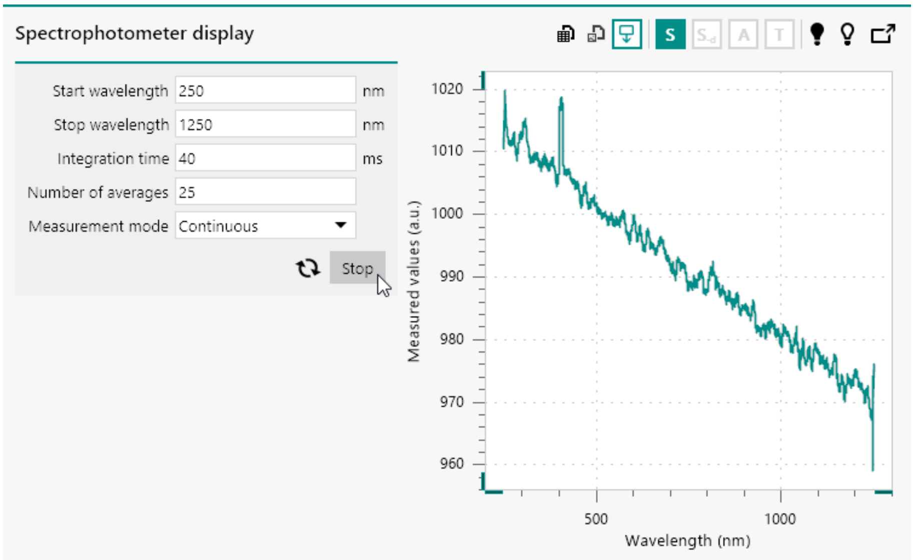


Figure 147 Measured spectra are displayed in the plot on the right-hand side

NOTE

The measured data is displayed in arbitrary units.

If needed, the measurement properties can be adjusted while spectra are acquired *(see Figure 148, page 131)*.

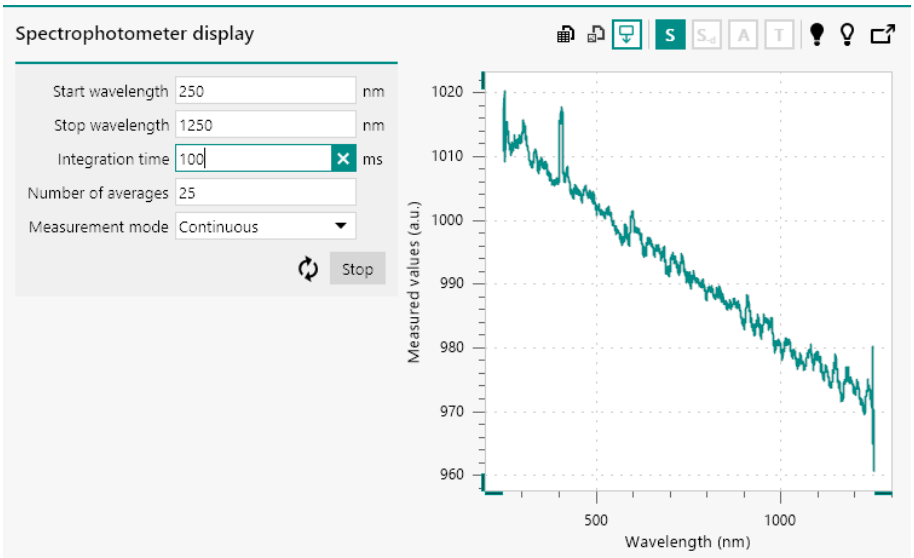


Figure 148 Measurement properties can be adjusted

NOTE

While spectra are being acquired, the **Hardware setup** of the spectrophotometer cannot be adjusted.

In continuous measurement mode, the acquisition of data can be stopped by pressing the Stop button.

After stopping the acquisition, it is possible to save the last measured spectrum as a *Dark spectrum* or as a *Reference spectrum*, by clicking the button or button, respectively *(see Figure 149, page 132)*.

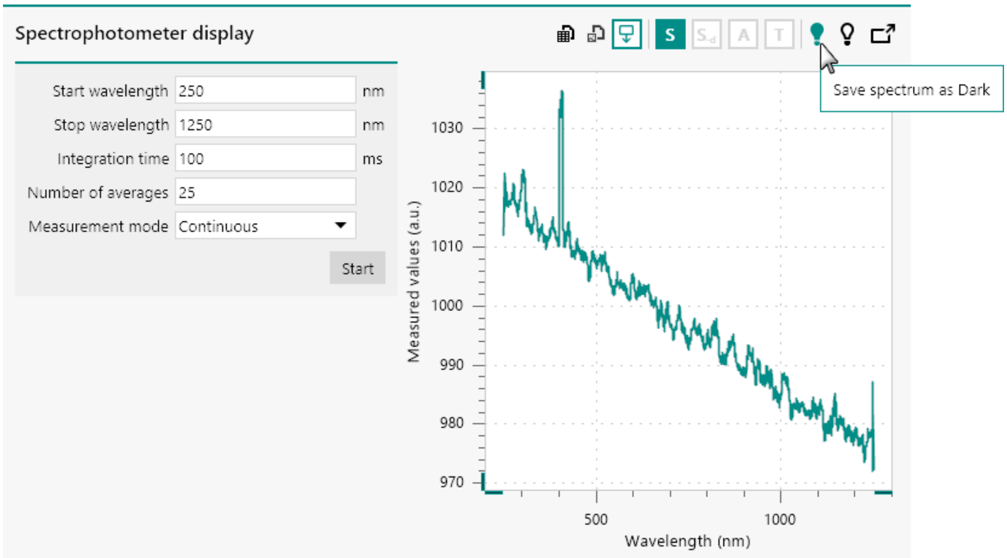


Figure 149 Saving a measured spectrum as Dark spectrum

When a *Dark* or *Reference* spectrum is saved, a check mark (or) will be visible in the top right corner of the **Spectrophotometer display** window *(see Figure 150, page 133)*.

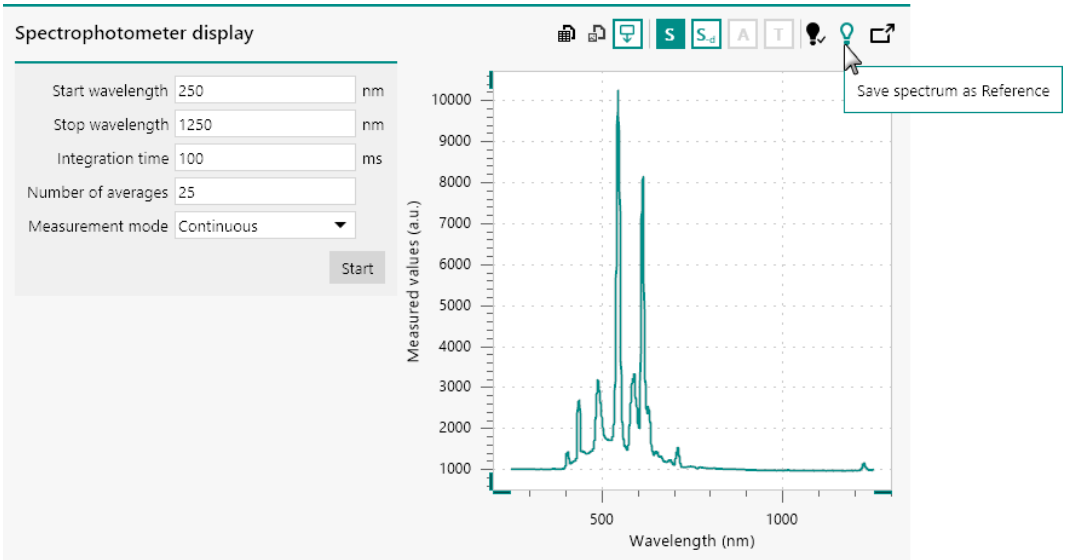


Figure 150 Saved spectra are indicated by a check mark

NOTE

It is possible to overwrite the saved *Dark* or *Reference* spectrum by clicking the associated buttons again.

NOTE

Changing the acquisition properties will discard the saved *Dark* and *Reference* spectrum.

5.4.2.1 Display modes

The **Spectrophotometer display** panel provides the possibility to toggle between different display modes, using the buttons (S, S-d, A, T) located in the top right corner *(see Figure 151, page 134)*.

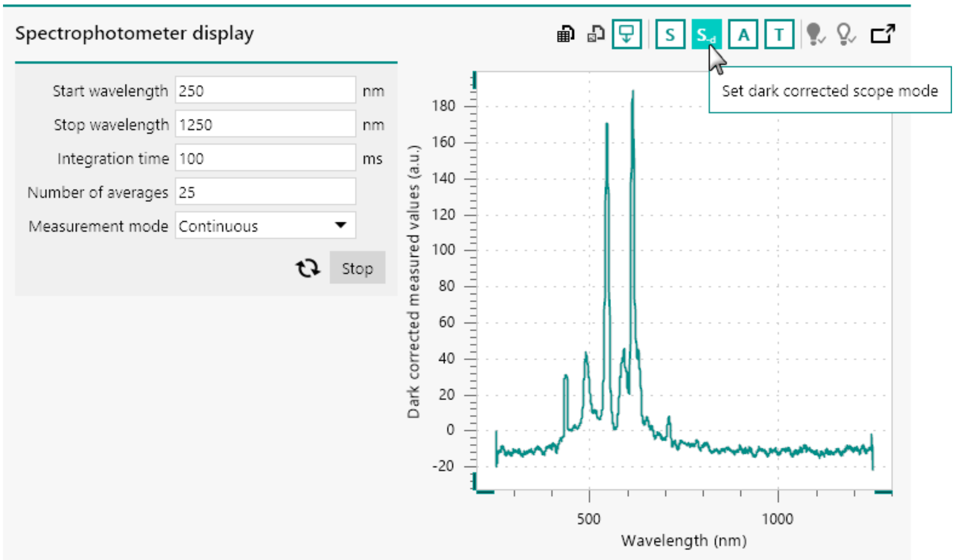


Figure 151 Controlling the display mode of the measured data

The following display modes are available:

- **Scope mode** (S): this mode shows the raw data from the spectrophotometer is arbitrary units. This display mode is always available.
- **Dark corrected scope mode** (S_{-d}): this mode shows the raw data ($S_{Measured}$) from the spectrophotometer corrected with the stored *Dark* spectrum (S_{Dark}), in arbitrary units. This display mode is only available if a *Dark* spectrum is saved. The dark corrected scope data is calculated according to:

$$S_{-d} = S_{Measured} - S_{Dark}$$

- **Absorbance mode** (A): this mode shows the absorbance values calculated from the measured data ($S_{Measured}$), the stored *Dark* spectrum (S_{Dark}) and the stored *Reference* spectrum ($S_{Reference}$). This display mode is only available if a *Dark* and a *Reference* spectrum are saved.

$$A = -\log\left(\frac{S_{Measured} - S_{Dark}}{S_{Reference} - S_{Dark}}\right)$$

- **Transmittance mode** (T): this mode shows the transmittance values calculated from the measured data ($S_{Measured}$), the stored *Dark* spectrum (S_{Dark}) and the stored *Reference* spectrum ($S_{Reference}$). This display mode is only available if a *Dark* and a *Reference* spectrum are saved.

$$T = 100 \cdot \left(\frac{S_{Measured} - S_{Dark}}{S_{Reference} - S_{Dark}}\right)$$

NOTE

The modes can be toggled while spectra are acquired.

5.4.2.2 Step through data

The **Spectrophotometer display** panel provides the possibility to toggle the *Step through data* mode on or off using the the button in the top right corner *(see Figure 152, page 135)*.

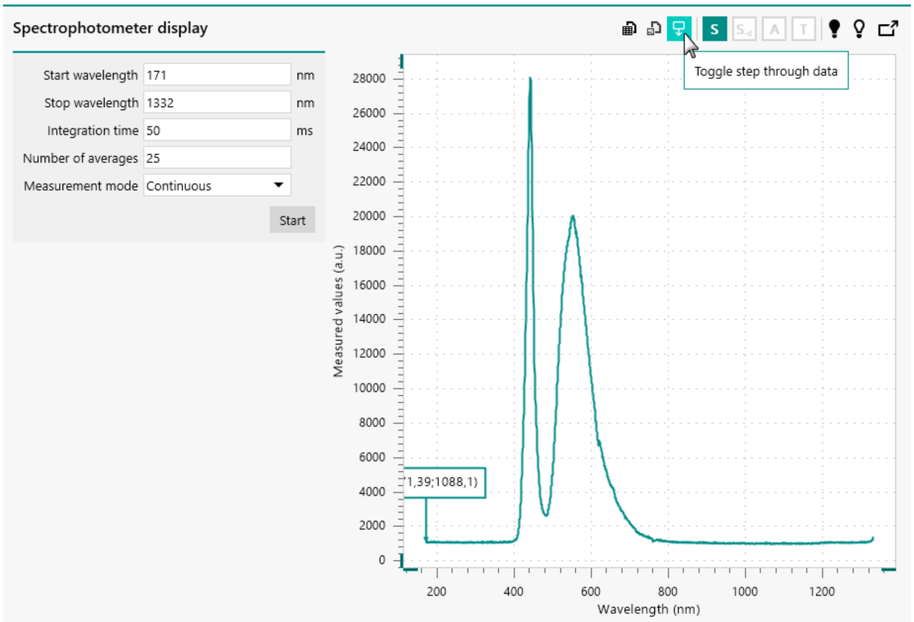


Figure 152 The Step through data option can be used in the Spectrophotometer control panel

When the *Step through data* mode is on, an additional indicator is added to the plot, showing the X and Y coordinates of the point indicated by the arrow, as shown in *Figure 152*.

NOTE

The indicator is always shown for the first data point of the plot.

It is possible to relocate the indicator in the following ways *(see Figure 153, page 136)*:

- By clicking anywhere in the plot area: the indicator is relocated to the closest data point of the plot.
- Using the **[←]**/**[→]**: the indicator can be moved by 1 point at a time.

- Using the **[←]/[→]** and **[CTRL]**: the indicator can be moved by 10 points at a time.
- **[←]/[→]** and **[CTRL]** and **[SHIFT]**: the indicator can be moved by 100 points at a time.

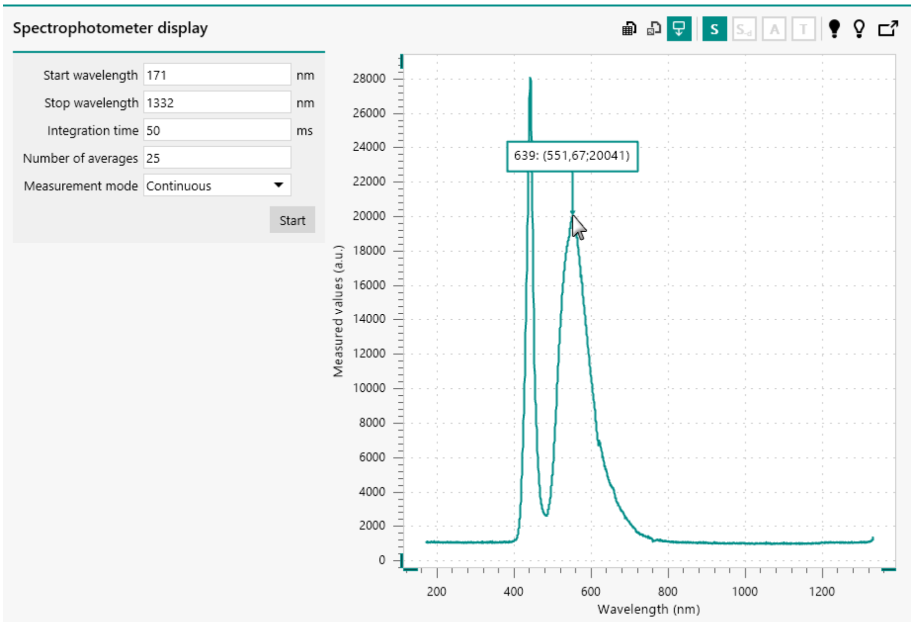


Figure 153 It is possible to relocate the indicator using the mouse or keyboard

5.4.2.3 Export data and plot

The **Spectrophotometer display** panel provides the possibility to export the measured data. Measured value can either be exported to ASCII or Excel format or as an image, using the provided and buttons in the top right corner of the panel *(see Figure 154, page 137)*.

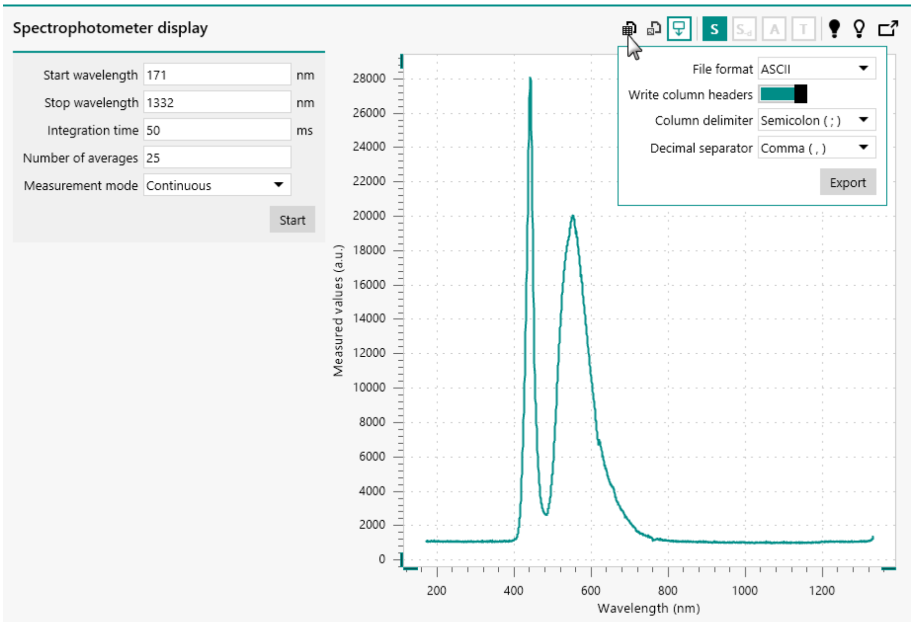


Figure 154 The measured data can be exported

Clicking the button displays a pop-out menu providing controls of the format of the exported file *(see Figure 155, page 137)*.

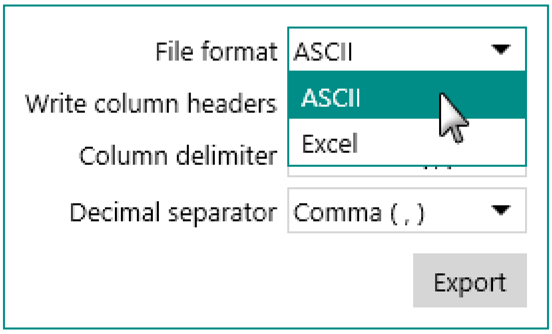


Figure 155 The data can be exported to ASCII or Excel

The data can be exported as ASCII or to Excel. The following properties can be specified:

- **File format:** specifies the format of the output file (ASCII or Excel), using the provided drop-down list.
- **Write column headers:** a toggle that can be used to indicate if the names of the signals need to be added to the output file.
- **Column delimiter:** specifies the symbol used as a column separator, using the provided drop-down list. This property is only available for ASCII output.
- **Decimal separator:** specifies the decimal separator symbol used in the output file, using the provided drop-down list. This property is only available for ASCII output.

Clicking the Export button displays a save dialog window which can be used to specify the filename and location *(see Figure 156, page 138)*.

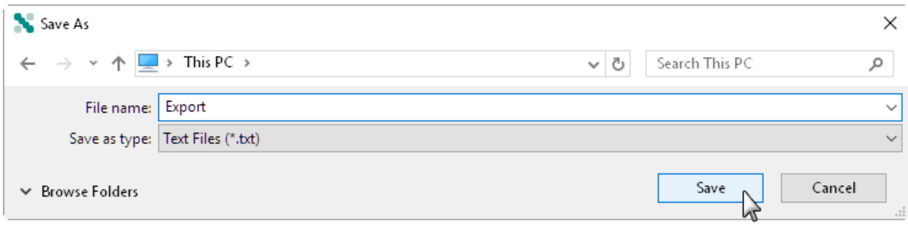


Figure 156 Specifying the filename and location

NOTE

All of the available data is exported to the file.

Clicking the button displays a pop-out menu providing controls of the format the size of the exported image file *(see Figure 157, page 138)*.

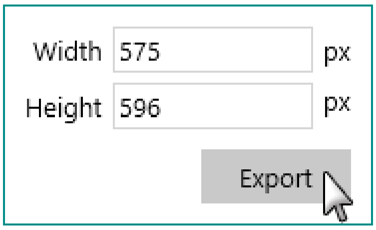


Figure 157 Exporting the data as image

Two types of image types can be used when exporting plots:

- **Pixel based output:** the data is exported to a pixel based file format, with or without compression (*.bmp, *.png, *.jpg, *.tiff, *.gif). When this type is used, the size of the image is specified in pixels.
- **Vector based output:** the data is exported to a vector based file format (*.emf, *.svg, *.wmf). When this type is used, the size of the image is specified in arbitrary units.

Clicking the Export button displays a Windows explorer dialog which can be used to specify the path, name and file type used to create the output image file *(see Figure 158, page 138)*.

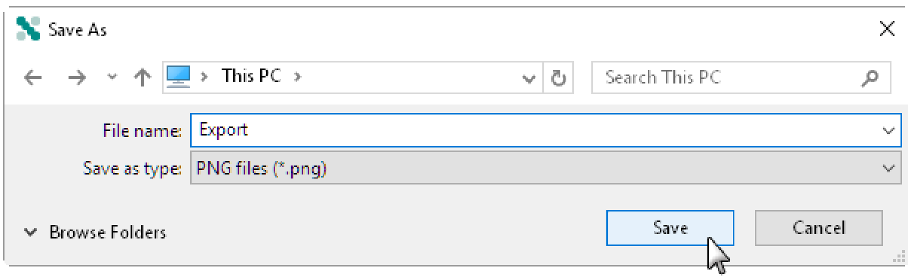


Figure 158 Specifying the name, location and type of output file

5.5 Metrohm devices control panel

Double clicking a Metrohm device tile in the **Instruments** panel opens the **Metrohm device control** panel in a new **tab**. Depending on the type of device, the content of the Metrohm device control panel will be different. Four categories of Metrohm devices are supported in NOVA:

- Metrohm 800 Dosino with 807 Dosing Cylinder
- Metrohm 814, 815 and 858 Sample Processor
- Metrohm 801 Magnetic Stirrer and Metrohm 804 Titration Stand with 741 Magnetic Stirrer or 802 Rod Stirrer
- Metrohm 6.2148.010 Remote Box

5.5.1 Metrohm Dosino control panel

The **Metrohm Dosino control** panel opens in a new tab when a **Dosino** tile, shown in the **Instruments** panel, is double clicked *(see Figure 159, page 139)*.

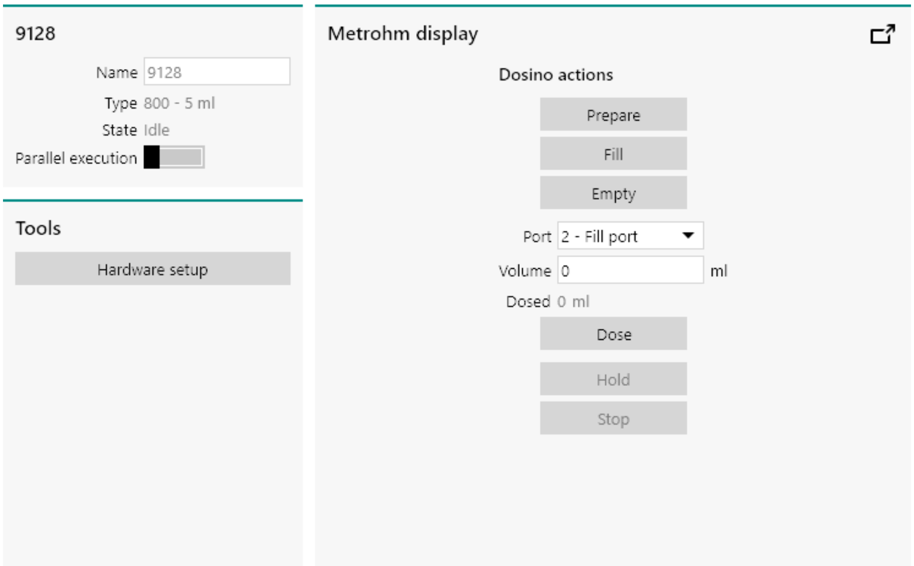


Figure 159 The Metrohm Dosino control panel

The **Metrohm Dosino control** panel shows three different sub-panels:

- **Dosino information panel:** this panel displays information about the Dosino.
- **Tools:** this panel provides quick access to the hardware setup.
- **Metrohm display:** this panel provides manual control of the Dosino.

5.5.1.1 Dosino information panel

The **Dosino information** panel shown in the **Instrument control** panel provides information on the selected instrument *(see Figure 160, page 140)*.

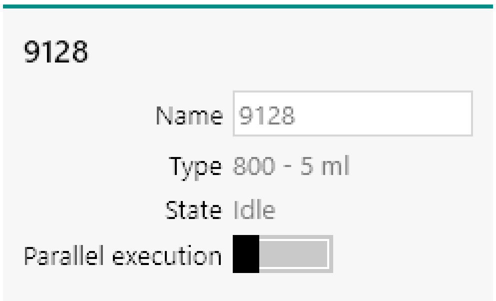


Figure 160 The Instrument information panel for the Metrohm Dosino

The following items and controls are listed:

- **Name:** an input field which can be used to give a dedicated name to the instrument. By default, the name of the instrument corresponds to the last four or five digits of the instrument serial number.
- **Type:** indicates the type of instrument. For the **Dosino**, the type is 800, followed by the volume of the Dosing Cylinder, in ml. The value can be 0, 2, 5, 10, 20 or 50. A value of 0 means that no Dosing Cylinder is detected.
- **State:** indicates the state of the instrument.
- **Parallel execution:** a toggle that can be used to specify if the parallel execution is allowed for this device (off by default).

The **Name** can be edited if required. This is convenient for identifying the **Dosino** in NOVA. If a specific name is provided, this name will be used throughout the whole NOVA application to identify the **Dosino** *(see Figure 161, page 140)*.

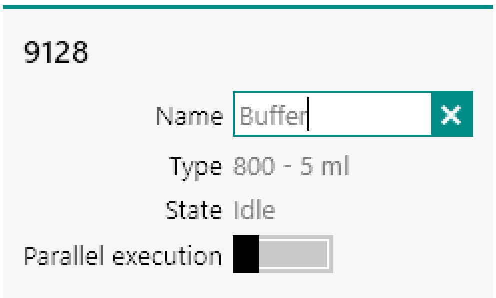


Figure 161 The Dosino name can be modified if required

NOTE

The **Name** must be unique.

5.5.1.2 Dosino hardware setup

The configuration of the connected **Dosino** can be adjusted in the Hardware setup. To open the Hardware setup, click the dedicated button in the **Tools** panel *(see Figure 162, page 141)*.

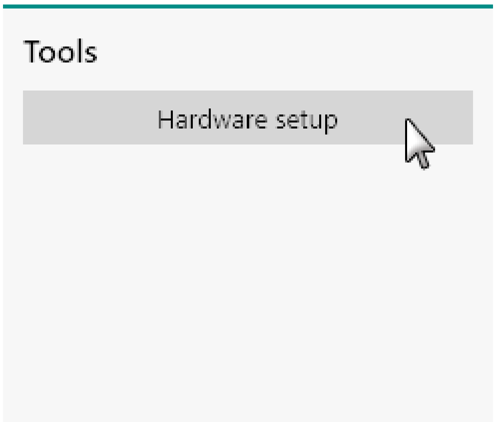


Figure 162 Click the Hardware setup button in the Tools panel to open the Dosino Hardware setup

A new window will be displayed, showing the settings for the selected **Dosino** *(see Figure 163, page 142)*.

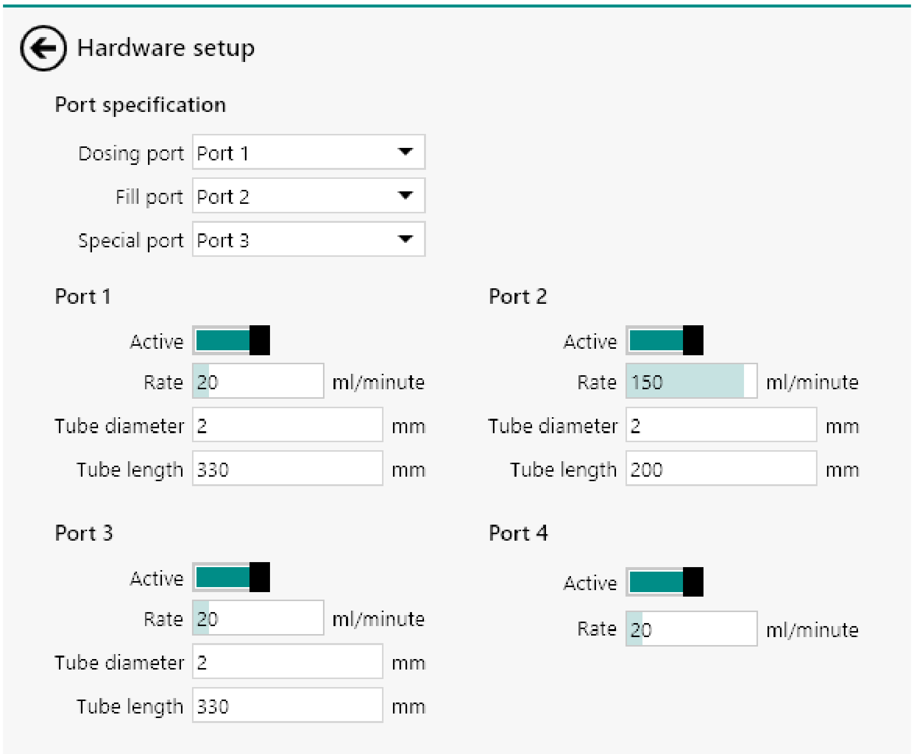


Figure 163 The Metrohm 800 Dosino hardware setup

The following settings can be specified for each Dosino:

- **Port specification:** assigns a specific role to the Dosino ports. Three roles can be defined (Dosing port, Fill port and Special port). The ports are assigned to a specific role using the provided drop-down list.
  - **Dosing port:** this is the port used for dosing by the **Dosino**. The default is Port 1.
  - **Fill port:** this the port for filling the **Dosino**. The default is Port 2.
  - **Special port:** this is the alternative dosing port used by the **Dosino**. The default is Port 3
- **Port 1-3**
  - **Active:** specifies if the port is active or not, using the provided  toggle. Ports are active by default. If the active state is set to off, the port will be skipped by **Prepare** and **Empty** actions executed on the Dosino.
  - **Rate:** the rate used by the port, in ml/minute. The maximum value is 150 ml/minute.
  - **Tube diameter:** the diameter of the tube connected to the port, in mm.
  - **Tube length:** the length of the tube connected to the port, in mm.

- **Port 4**
  - **Active:** specifies if the port is active or not, using the provided toggle. Port 4 is active by default. The active state is set to off, the port will be skipped by **Prepare** and **Empty** actions executed on the Dosino.
  - **Rate:** the rate used by the port, in ml/minute. The maximum value is 150 ml/minute.

5.5.1.3 Dosino manual control

The **Metrohm display** panel provides controls which can be used to manually operate the selected **Dosino**. These controls can be used at any time *(see Figure 164, page 143)*.

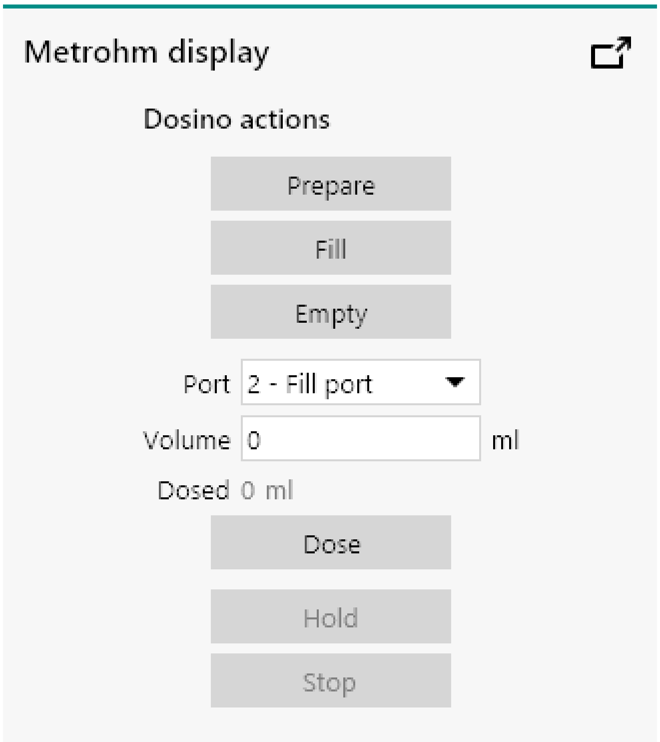


Figure 164 The manual controls of the Dosino

The following controls are provided:

- **Prepare:** starts a single prepare cycle on the Dosino.
- **Fill:** fill the Dosing cylinder completely, using the specified fill port.
- **Empty:** starts an empty cycle on the Dosino.
- **Port:** selects the active port, using the provided drop-down list.
- **Volume:** an input field which can be used to specify a volume to manually dose using the Dosino, in ml.
- **Dosed:** a read-only field which shows the dosed volume.
- **Dose:** starts a dosing action, using the specified *Volume* and using the selected *Port*.

- **Hold/Continue:** holds the current action, if possible. This button is only enabled when the Dosino is not idle and when the action carried out by the Dosino can be held. When the Dosino is held, the **Hold** button switches to a **Continue** button which can be clicked again to resume the action.
- **Stop:** stops the current action, if possible. This button is only enabled when the Dosino is not idle and when the action carried out by the Dosino can be stopped.

NOTE

The controls shown in the **Metrohm display** panel depend on the hardware setup of the selected **Dosino**.

For convenience, it is possible to undock the **Metrohm display** panel and display the its content in a separate window. To do this, click the button in the top right corner of the **Metrohm display** panel *(see Figure 165, page 144)*.

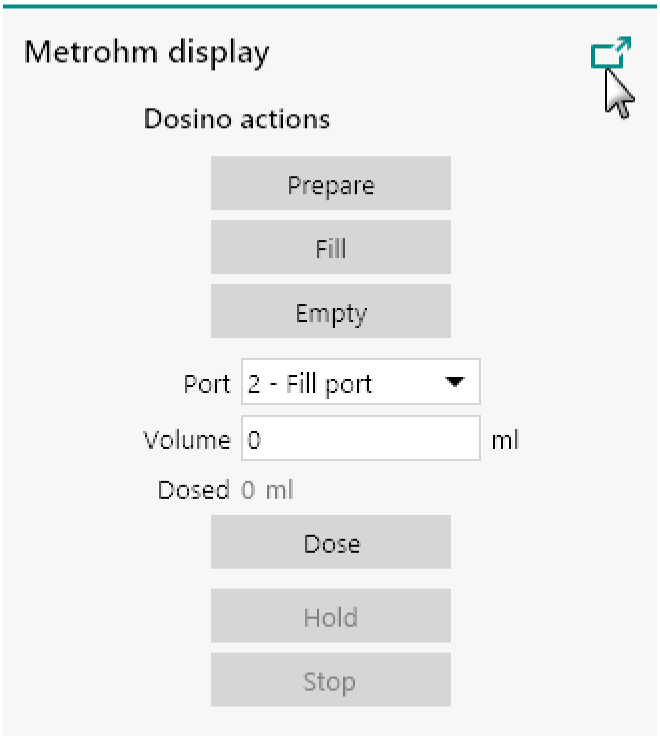


Figure 165 Undocking the Metrohm display panel for Dosino control

The content of the **Metrohm display** panel will be duplicated in a new window on top of the main NOVA software window. This new window can be moved next to the main NOVA window, or to another computer display if available *(see Figure 166, page 145)*.

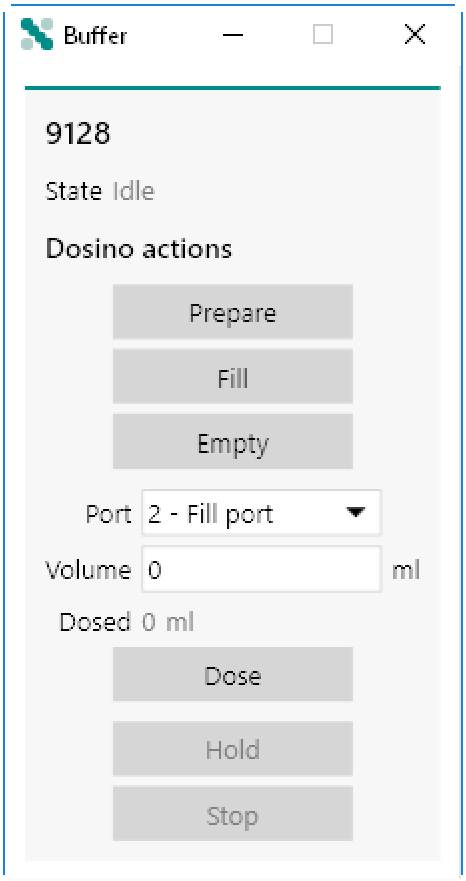


Figure 166 The undocked Metrohm display panel window

5.5.2 Metrohm Sample Processor control panel

The **Metrohm Sample Processor control** panel opens in a new tab when a **Sample Processor** tile, shown in the **Instruments** panel, is double clicked *(see Figure 167, page 146)*.

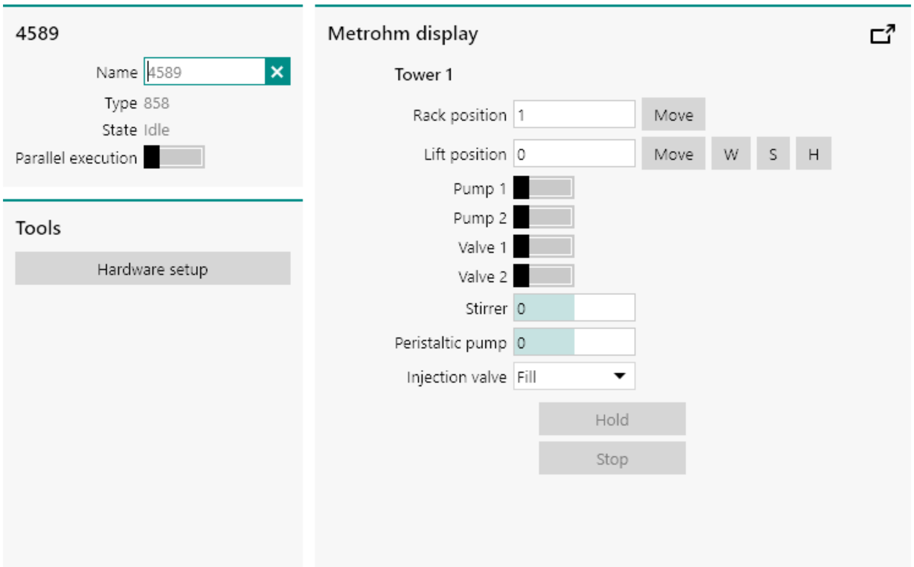


Figure 167 The Metrohm Sample Processor control panel

The **Metrohm Sample Processor control** panel shows three different sub-panels:

- **Sample Processor information panel:** this panel displays information about the Sample Processor.
- **Tools:** this panel provides quick access to the hardware setup.
- **Metrohm display:** this panel provides manual control of the Sample Processor.

CAUTION

The controls provided for the **Metrohm Sample Processor** depend on the configuration and the options installed on the device. The hardware setup needs to be adjusted in order to match the instrument configuration. For more information, please refer to the user manual provided with the instrument.

5.5.2.1 Sample Processor information panel

The **Sample Processor information** panel shown in the **Instrument control** panel provides information on the selected instrument *(see Figure 168, page 147)*.

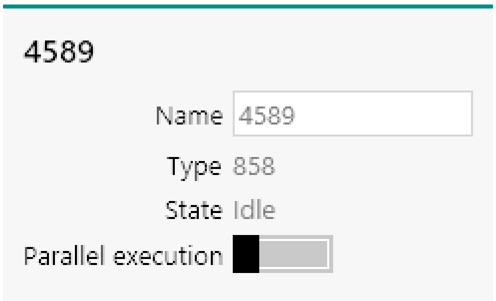


Figure 168 The Instrument information panel for the Metrohm Sample Processor

The following items and controls are listed:

- **Name:** an input field which can be used to give a dedicated name to the instrument. By default, the name of the instrument corresponds to the last four or five digits of the instrument serial number.
- **Type:** indicates the type of instrument. For the **Sample Processor**, the type can be 814, 815 or 858.
- **State:** indicates the state of the instrument.
- **Parallel execution:** a toggle that can be used to specify if the parallel execution is allowed for this device (off by default).

The **Name** can be edited if required. This is convenient for identifying the **Sample Processor** in NOVA. If a specific name is provided, this name will be used throughout the whole NOVA application to identify the **Sample Processor** *(see Figure 169, page 147)*.

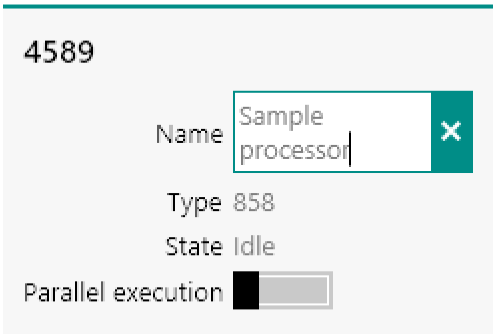


Figure 169 The Sample Processor name can be modified if required

NOTE

The **Name** must be unique.

5.5.2.2 Sample Processor hardware setup

The configuration of the connected **Sample Processor** can be adjusted in the Hardware setup. To open the Hardware setup, click the dedicated button in the **Tools** panel *(see Figure 170, page 148)*.

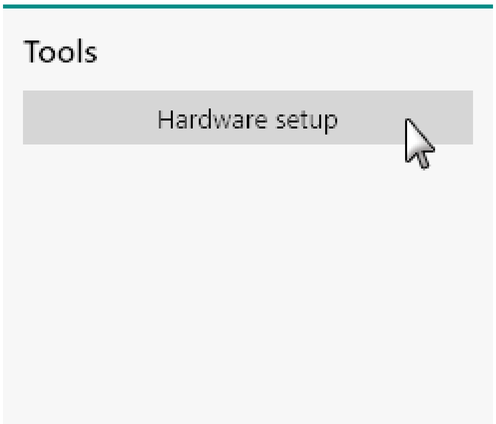


Figure 170 Click the Hardware setup button in the Tools panel to open the Sample Processor Hardware setup

A new window will be displayed, showing the settings for the selected **Sample Processor** *(see Figure 171, page 148)*.

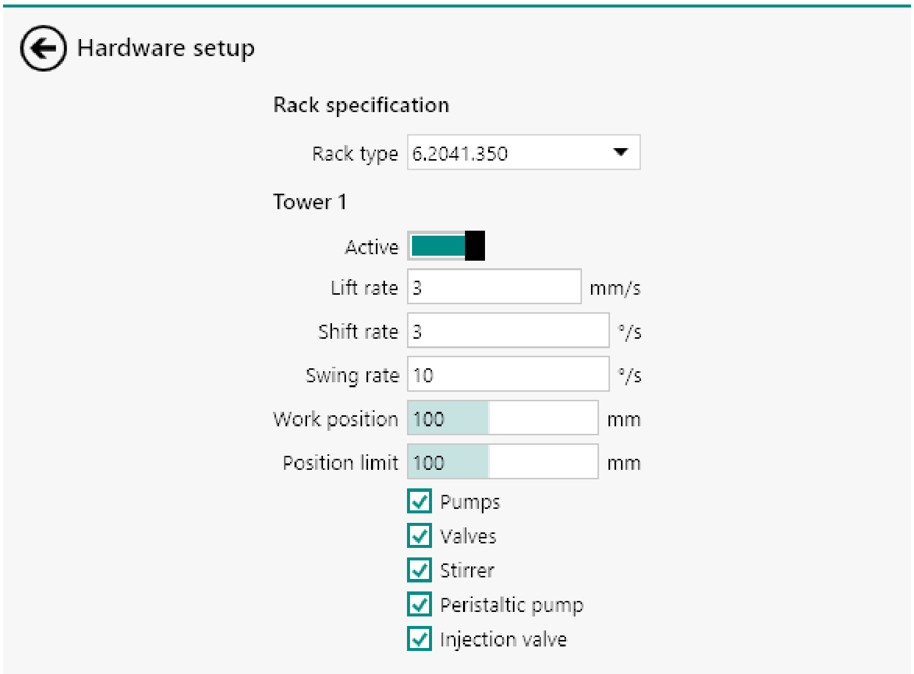


Figure 171 The Metrohm Sample Processor hardware setup

The following settings can be specified for each **Sample Processor**:

- **Rack specification:** defines the sample rack mounted on the **Sample Processor**, using the provided drop-down list. The available sample racks are identified by their Metrohm part numbers.

- **Tower 1**
  - **Active:** specifies if the tower of the **Sample processor** is active or not, using the provided toggle. Towers are active by default. If the active state is set to off, the tower will not be available for use.
  - **Lift rate:** the rate used by the lift, in mm/s. The maximum value is 25 mm/s.
  - **Shift rate:** the rotation rate used by the sample rack, in °/s. The maximum value is 20 °/s.
  - **Swing rate:** the swing rate used by the swing arm, if available, in °/s. The maximum is 55 °/s.
  - **Work position:** the work position used by the lift of the tower, in mm with respect to the top of the tower. The maximum value is 235 mm.
  - **Position limit:** the maximum position used by the lift of the tower, in mm with respect to the top of the tower. The maximum value is 235 mm. The position limit must always be larger or equal to the work position.
  - **Pumps:** specifies if pumps are installed on the back of the tower or connected to the back of the tower.
  - **Valves:** specifies if valves are installed on the back of the tower.
  - **Stirrer:** specifies if a stirrer is connected to the back of the tower.
  - **Peristaltic pump:** specifies if a peristaltic pump is installed on the side of the tower. This option is only available with the **Metrohm 858 Professional Sample Processor**.
  - **Injection valve:** specifies if an injection valve is installed on the side of the tower. This option is only available with the **Metrohm 858 Professional Sample Processor**.

- **Tower 2**
  - **Active:** specifies if the tower of the Sample processor is active or not, using the provided toggle. Towers are active by default. If the active state is set to off, the tower will not be available for use.
  - **Lift rate:** the rate used by the lift, in mm/s. The maximum value is 25 mm/s.
  - **Shift rate:** the rotation rate used by the sample rack, in °/s. The maximum value is 20 °/s.
  - **Swing rate:** the swing rate used by the swing arm, if available, in °/s. The maximum is 55 °/s.
  - **Work position:** the work position used by the lift of the tower, in mm with respect to the top of the tower. The maximum value is 235 mm.
  - **Position limit:** the maximum position used by the lift of the tower, in mm with respect to the top of the tower. The maximum value is 235 mm. The position limit must always be larger or equal to the work position.
  - **Pumps:** specifies if pumps are installed on the back of the tower or connected to the back of the tower.
  - **Valves:** specifies if valves are installed on the back of the tower.
  - **Stirrer:** specifies if a stirrer is connected to the back of the tower.

NOTE

Tower 2 is only available with the **Metrohm 814** and **815 Sample Processors**.

NOTE

The settings defined in the **Sample Processor** hardware setup affect the controls provided in the **Metrohm display**.

5.5.2.3 Sample Processor manual control

The **Metrohm display** panel provides controls which can be used to manually operate the selected **Sample Processor**. These controls can be used at any time *(see Figure 172, page 151)*.

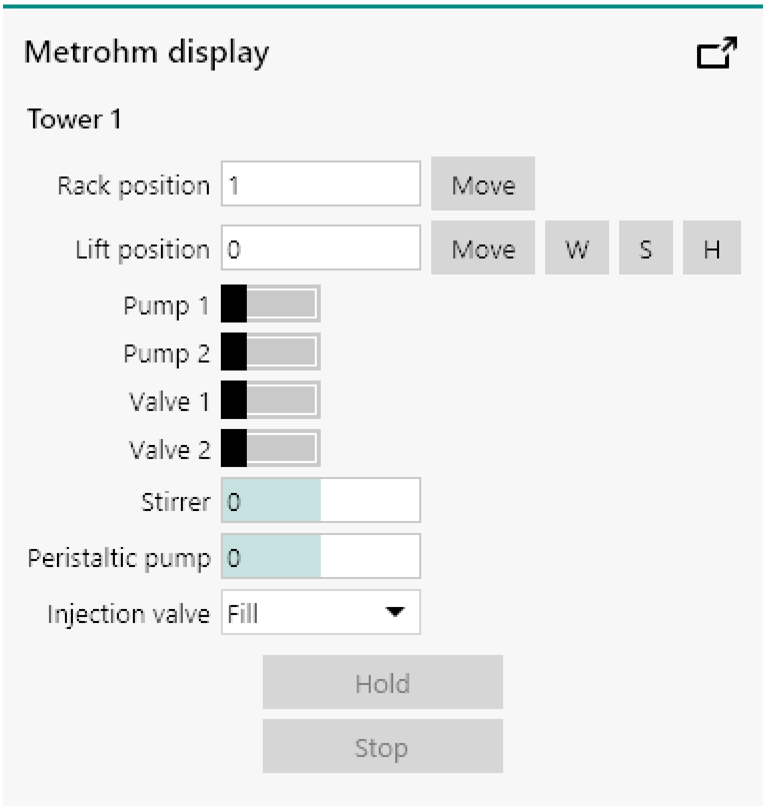


Figure 172 The manual controls of the Sample Processor

The following controls are provided:

- **Tower 1**
  - **Rack position:** sets the position of the sample rack with respect to tower 1.
  - **Lift position:** sets the position of the lift on tower 1, in mm with respect to the top of the tower. Shortcut buttons are provided for the *Work* position (W), *Shift* position (S) and *Home* position (H).
  - **Pump 1:** switches pump 1 on or off using the provided toggle.
  - **Pump 2:** switches pump 2 on or off using the provided toggle.
  - **Valve 1:** switches valve 1 on or off using the provided toggle.
  - **Valve 2:** switches valve 2 on or off using the provided toggle.
  - **Stirrer:** sets the rotation rate of the stirrer, from -15 to 15. When the rotation rate is set to 0, the stirrer will stop.
  - **Peristaltic pump:** sets the rotation rate of the peristaltic pump, from -15 to 15. When the rotation rate is set to 0, the pump will stop.
  - **Injection valve:** sets the state of the injection valve, using the provided drop-down list (*Fill* or *Inject*).
- **Tower 2**
  - **Rack position:** sets the position of the sample rack with respect to tower 2.
  - **Lift position:** sets the position of the lift on tower 2, in mm with respect to the top of the tower. Shortcut buttons are provided for the *Work* position (W), *Shift* position (S) and *Home* position (H).
  - **Pump 1:** switches pump 1 on or off using the provided toggle.
  - **Pump 2:** switches pump 2 on or off using the provided toggle.
  - **Valve 1:** switches valve 1 on or off using the provided toggle.
  - **Valve 2:** switches valve 2 on or off using the provided toggle.
  - **Stirrer:** sets the rotation rate of the stirrer, from -15 to 15. When the rotation rate is set to 0, the stirrer will stop.
- **Hold/Continue:** holds the current action, if possible. This button is only enabled when the Sample processor is not idle and when the action carried out by the Sample processor can be held. When the Sample processor is held, the **Hold** button switches to a **Continue** button which can be clicked again to resume the action.

- **Stop:** stops the current action, if possible. This button is only enabled when the Sample processor is not idle and when the action carried out by the Sample processor can be stopped.

NOTE

The controls shown in the **Metrohm display** panel depend on the hardware setup of the selected **Sample Processor**.

For convenience, it is possible to undock the **Metrohm display** panel and display the its content in a separate window. To do this, click the button in the top right corner of the **Metrohm display** panel *(see Figure 173, page 153)*.

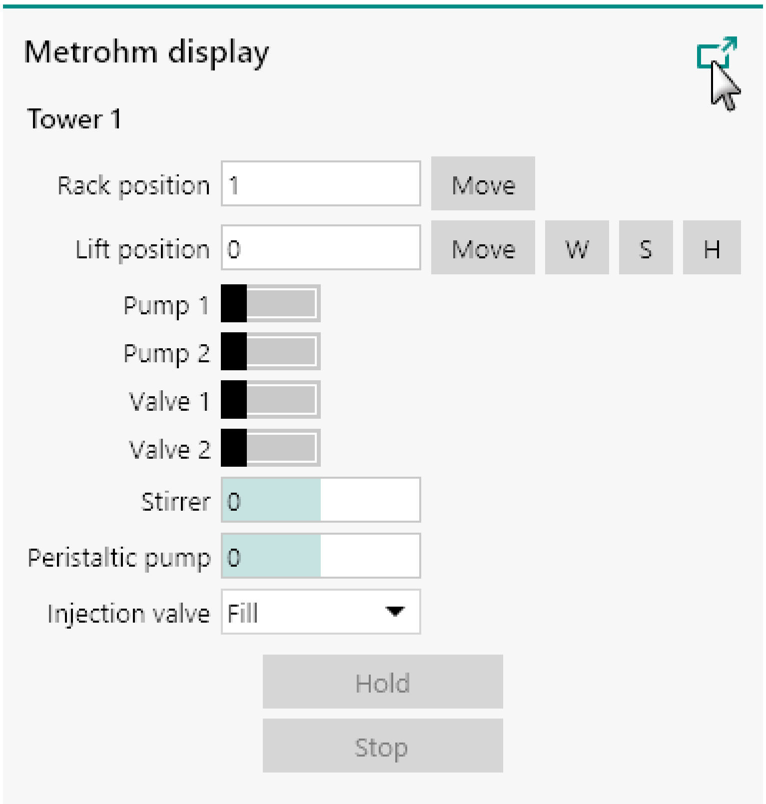


Figure 173 Undocking the Metrohm display panel for Sample Processor control

The content of the **Metrohm display** panel will be duplicated in a new window on top of the main NOVA software window. This new window can be moved next to the main NOVA window, or to another computer display if available *(see Figure 174, page 154)*.

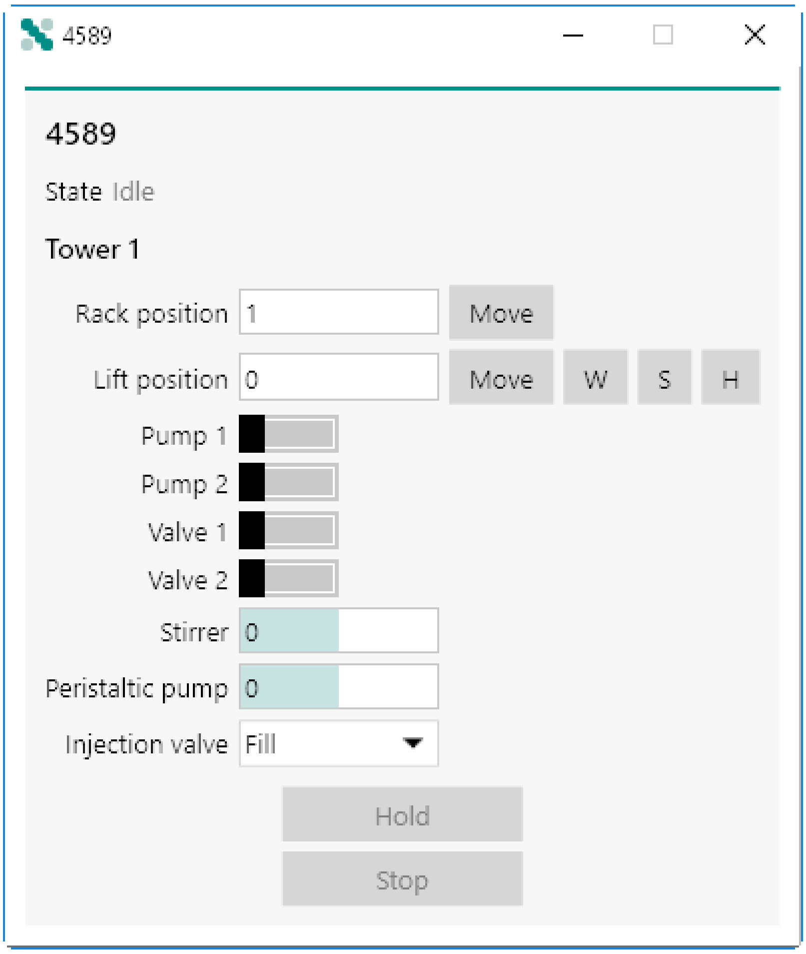


Figure 174 The undocked Metrohm display panel window

5.5.3 Metrohm Stirrer control panel

The **Metrohm Stirrer control** panel opens in a new tab when a **Stirrer** tile, shown in the **Instruments** panel, is double clicked *(see Figure 175, page 155)*.

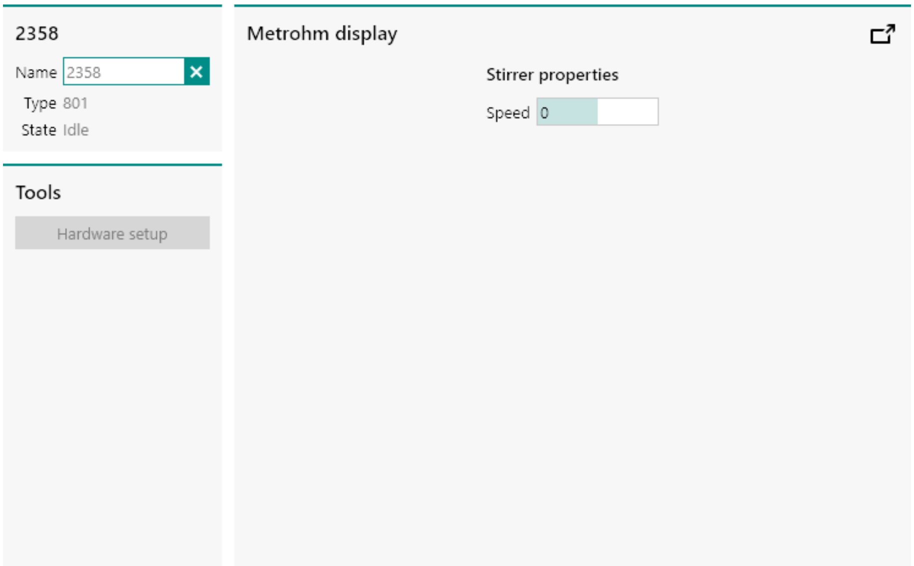
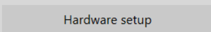


Figure 175 The Metrohm Stirrer control panel

The **Metrohm Stirrer control** panel shows three different sub-panels:

- **Stirrer information panel:** this panel displays information about the stirrer.
- **Metrohm display:** this panel provides manual control of the stirrer.

NOTE

The  button, located in the **Tools** panel, is disabled for the Metrohm Stirrer.

5.5.3.1 Stirrer information panel

The **Stirrer information** panel shown in the **Instrument control** panel provides information on the selected instrument *(see Figure 176, page 155)*.

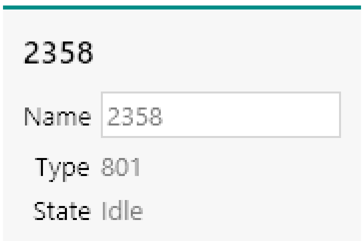


Figure 176 The Instrument information panel for the Metrohm Stirrer

The following items and controls are listed:

- **Name:** an input field which can be used to give a dedicated name to the instrument. By default, the name of the instrument corresponds to the last four or five digits of the instrument serial number.

- **Type:** indicates the type of instrument. For the **Stirrer**, the type can be 801, 802 or 741, depending on the type of device.
- **State:** indicates the state of the instrument.

The **Name** can be edited if required. This is convenient for identifying the **Stirrer** in NOVA. If a specific name is provided, this name will be used throughout the whole NOVA application to identify the **Stirrer** *(see Figure 177, page 156)*.

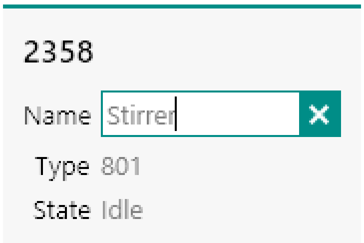


Figure 177 The Stirrer name can be modified if required

NOTE

The **Name** must be unique.

5.5.3.2 Stirrer manual control

The **Metrohm display** panel provides a control which can be used to manually operate the selected **Stirrer**. This control can be used at any time *(see Figure 178, page 156)*.

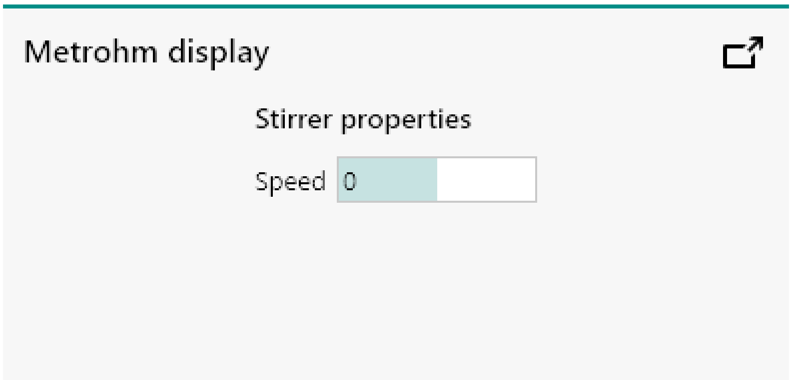


Figure 178 The manual controls of the Stirrer

The following control is provided:

- **Speed:** specifies the rotation rate of the Stirrer.

The **Speed** value can be adjusted between -15 and 15, with integral increments. Negative values will force the stirrer to rotate in the anti-clockwise direction while positive values will translate into a clockwise rotation. The higher the value, the higher the rotation rate. The speed value can be specified in two ways:

- **Numerically:** by typing the value directly in the **Metrohm display** panel *(see Figure 179, page 157)*.

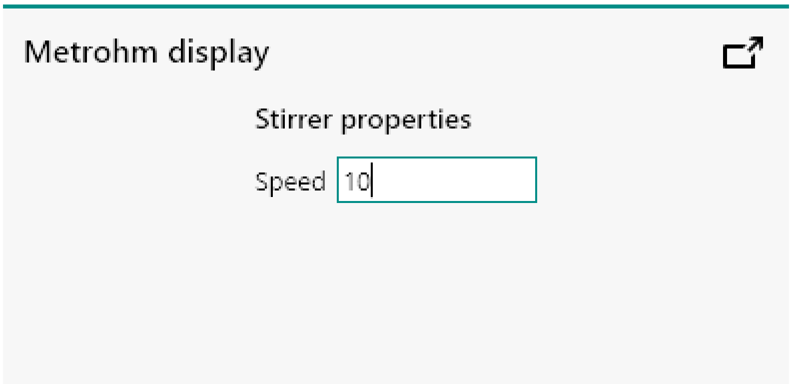


Figure 179 Adjusting the Speed numerically

- **Slider:** by clicking and dragging the slider control provided in the **Metrohm display** panel *(see Figure 180, page 157)*.

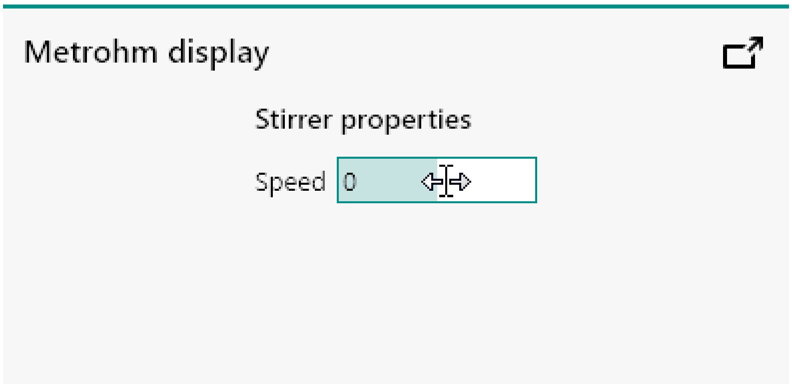


Figure 180 Adjusting the Speed with the slider

In both cases, the **Metrohm display** panel will be updated after the **Speed** value is adjusted *(see Figure 181, page 157)*.

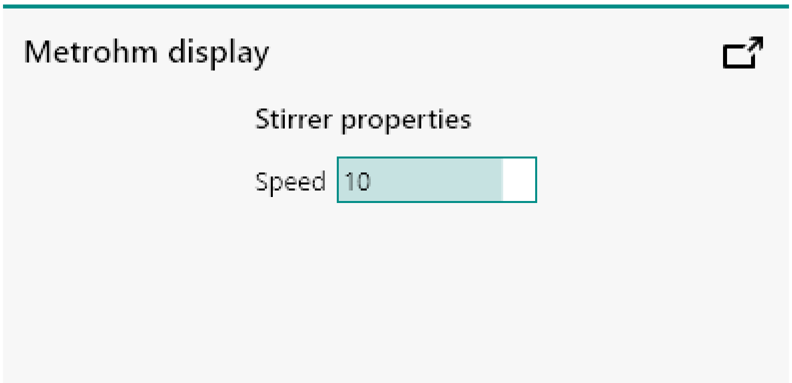


Figure 181 The Speed value is adjusted

NOTE

Setting the rotation rate to a value of 0 will force the Stirrer to stop.

NOTE

The actual rotation rate, in RPM, depends on the type of stirrer. For more information on the conversion of rotation rate steps provided in NOVA and the actual rotation rate, please refer to the Metrohm documentation supplied with each type of supported stirrer.

For convenience, it is possible to undock the **Metrohm display** panel and display the its content in a separate window. To do this, click the button in the top right corner of the **Metrohm display** panel *(see Figure 182, page 158)*.

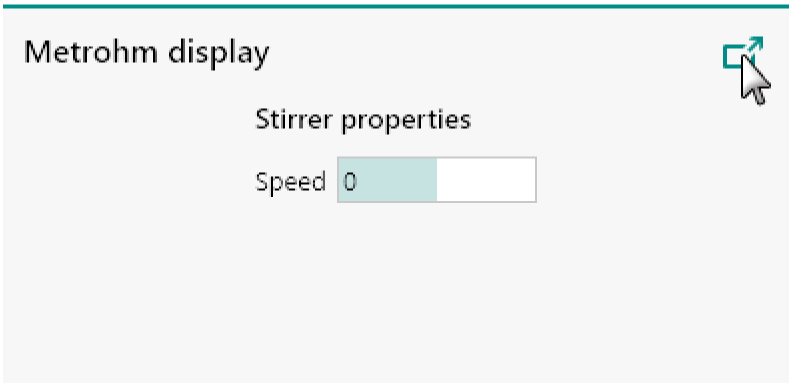


Figure 182 Undocking the Metrohm display panel for Stirrer control

The content of the **Metrohm display** panel will be duplicated in a new window on top of the main NOVA software window. This new window can be moved next to the main NOVA window, or to another computer display if available *(see Figure 183, page 159)*.

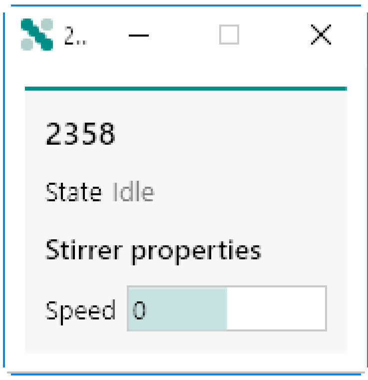


Figure 183 The undocked Metrohm display panel window

5.5.4 Metrohm Remote box control panel

The **Metrohm Remote box control** panel opens in a new tab when a **Remote box** tile, shown in the **Instruments** panel, is double clicked *(see Figure 184, page 159)*.

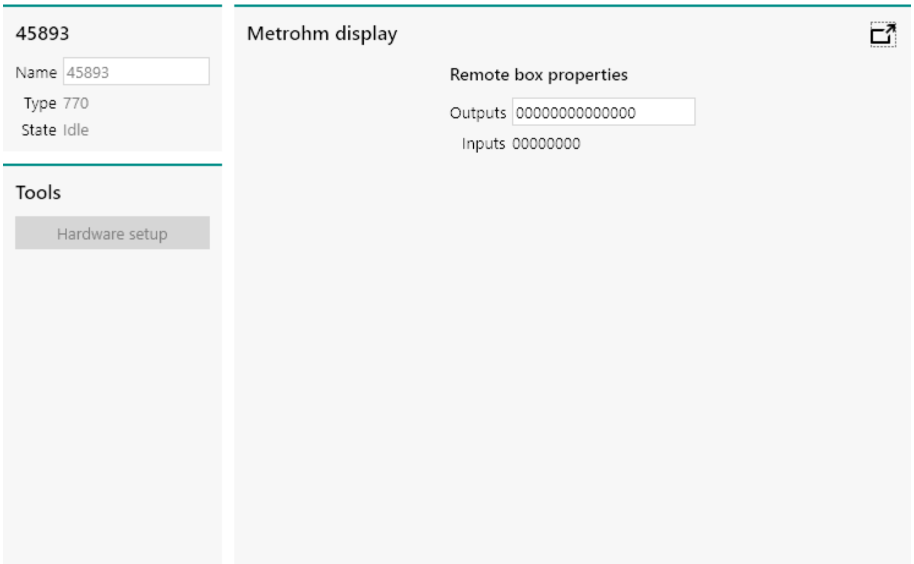


Figure 184 The Metrohm Remote box control panel

The **Metrohm Remote box control** panel shows three different sub-panels:

- **Control box information panel:** this panel displays information about the control box.
- **Metrohm display:** this panel provides manual control of the control box.

NOTE

The Hardware setup button, located in the **Tools** panel, is disabled for the Metrohm Control box.

5.5.4.1 Remote box information panel

The **Remote box information** panel shown in the **Instrument control** panel provides information on the selected instrument *(see Figure 185, page 160)*.

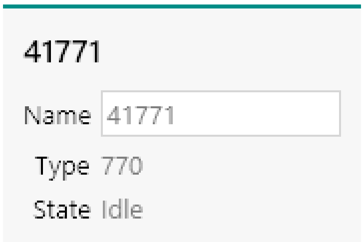


Figure 185 The Instrument information panel for the Metrohm Remote box

The following items and controls are listed:

- **Name:** an input field which can be used to give a dedicated name to the instrument. By default, the name of the instrument corresponds to the last four or five digits of the instrument serial number.
- **Type:** indicates the type of instrument. For the **Remote box**, the type is 770.
- **State:** indicates the state of the instrument.

The **Name** can be edited if required. This is convenient for identifying the **Remote box** in NOVA. If a specific name is provided, this name will be used throughout the whole NOVA application to identify the **Remote box** *(see Figure 186, page 160)*.

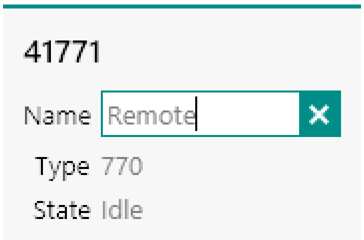


Figure 186 The Remote box name can be modified if required

NOTE

The **Name** must be unique.

5.5.4.2 Remote box manual control

The **Metrohm display** panel provides controls which can be used to manually operate the selected **Remote box**. These controls can be used at any time *(see Figure 187, page 161)*.

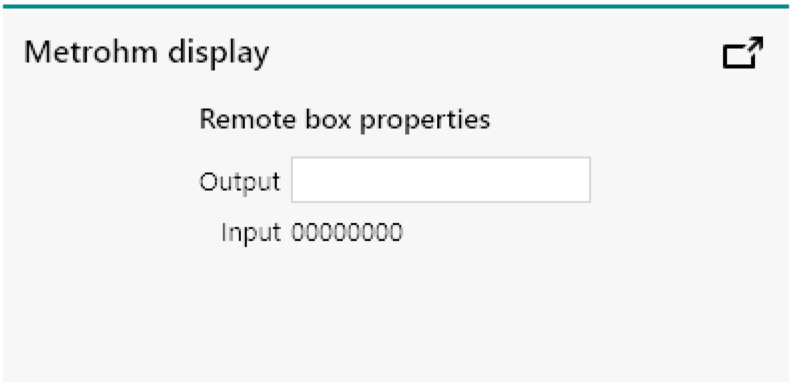


Figure 187 The manual controls of the Remote box

The following controls are provided:

- **Output:** specifies the state of the 14 output lines (numbered OUT13 to OUT0) of the Remote box. The state of each output line can set to either *low* or *high* state, represented by a **0** or a **1**, respectively. The state of the 14 output lines is specified as a 14 character string, consisting of **0** and **1**, representing the state of the output lines, from **OUT13 to OUT0**.
- **Input:** specifies the state of the 8 input lines (numbered IN7 to IN0). The state of each input line can be either *low* or *high*, represented by a **0** or a **1**, respectively. This is a read-only control.

For convenience, it is possible to undock the **Metrohm display** panel and display the its content in a separate window. To do this, click the button in the top right corner of the **Metrohm display** panel *(see Figure 165, page 144)*.

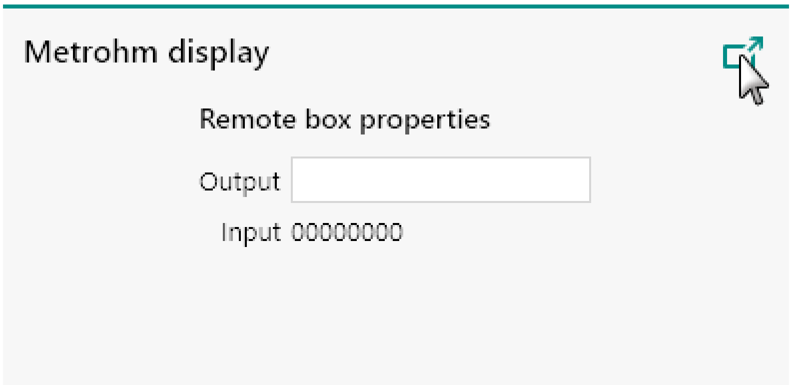


Figure 188 Undocking the Metrohm display panel for Remote box control

The content of the **Metrohm display** panel will be duplicated in a new window on top of the main NOVA software window. This new window can be moved next to the main NOVA window, or to another computer display if available *(see Figure 166, page 145)*.

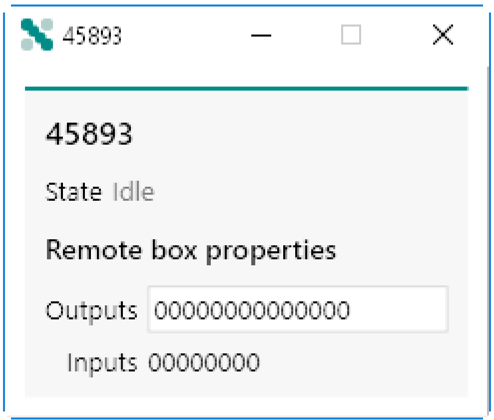


Figure 189 The undocked Metrohm display panel window

6 Library

The **Library** provides an interface to NOVA procedures, data and schedules. The Library can be accessed at any time from the **Dashboard**, by clicking the Open library button *(see Figure 190, page 163)*.

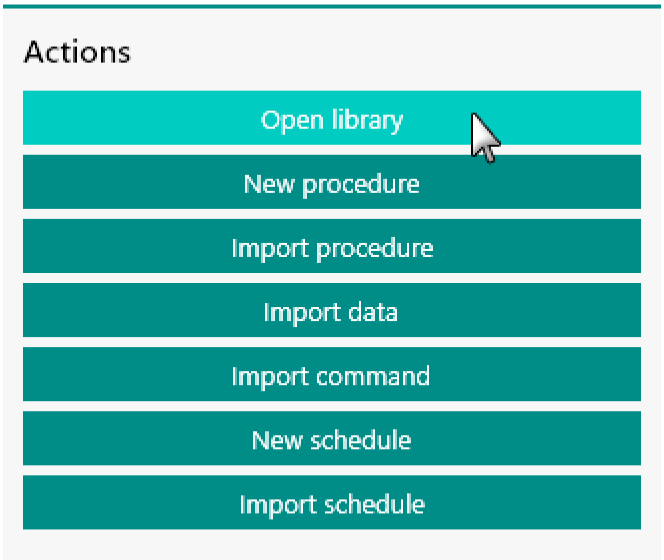


Figure 190 Opening the Library

NOTE

The **Recent items** panel in the **Dashboard** shows the most recent procedures (Recent procedures) and data files (Recent data) when NOVA is used *(see Chapter 4.2, page 75)*. Both lists are updated whenever a new procedure or data file is saved or updated.

The **Library** opens in a new tab, represented by the symbol. This tab is always located to the immediate right of the **Dashboard** tab (Home tab,), as shown in *Figure 191*.

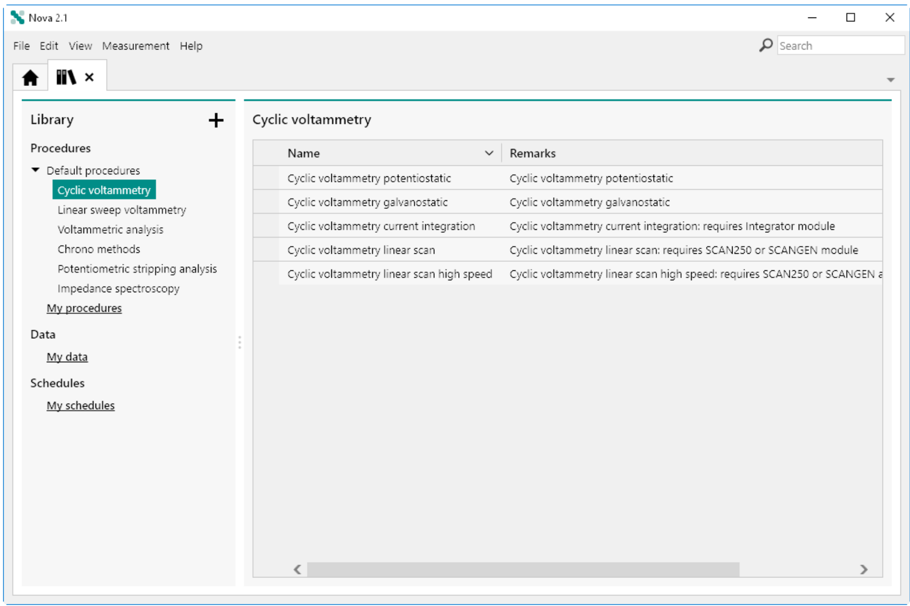


Figure 191 The Library tab is opened to the right of the Dashboard tab

The **Library** tab contains two panels. The panel on the left-hand side is a navigation panel that provides the possibility to select a **location** in which procedure, data or schedule files are located. The panel on the right-hand side lists the available procedure, data or schedule files for the selected location.

Four locations are always visible in the navigation panel:

- **Default procedures:** this location provides all the factory default procedures installed with NOVA. These procedures cannot be deleted or modified. They can be loaded and modified in the procedure editor and saved as new procedures.
- **My procedures:** this location contains all the user-defined procedures. This location maps all the procedure files located in the \My Documents\NOVA 2.1\Procedures folder.
- **My data:** this location contains all the user generated data files. This location maps all the procedure files located in the \My Documents \NOVA 2.1\Data folder.
- **My schedules:** this location contains all the user generated schedules. This location maps all the schedule files located in the \My Documents \NOVA 2.1\Schedules folder.

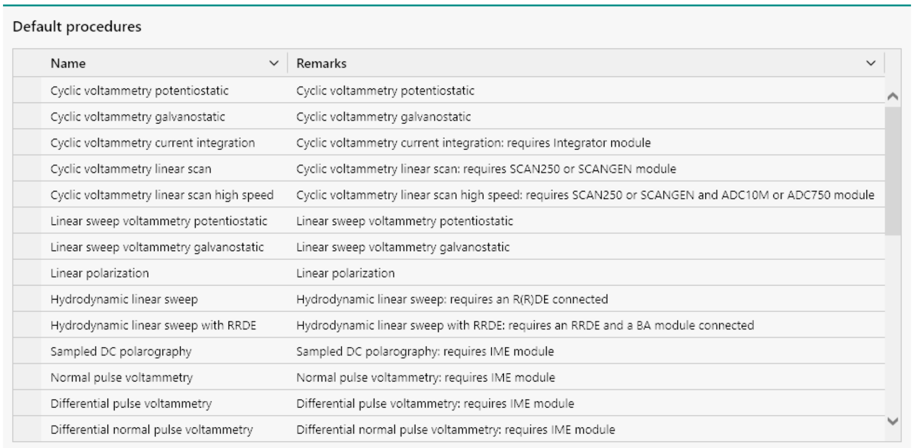
6.1 Default procedures

The **Default procedures** location is always visible in the **Library** panel. This location contains a series of factory default procedures. These procedures are intended to perform simple measurements and can be used for routine experiments or as templates for more elaborate procedures. These procedures are provided as read-only examples. They cannot be deleted or modified but it is possible to open these procedures and to save them as a modified version in one of the user-accessible locations.

NOTE

The procedures located in the **Default procedures** location are generated by the NOVA software. None of these procedures is available as an individual file on the computer.

The **Default procedures** are listed in the panel on the right-hand side *(see Figure 192, page 165)*.


Default procedures

Name	Remarks
Cyclic voltammetry potentiostatic	Cyclic voltammetry potentiostatic
Cyclic voltammetry galvanostatic	Cyclic voltammetry galvanostatic
Cyclic voltammetry current integration	Cyclic voltammetry current integration: requires Integrator module
Cyclic voltammetry linear scan	Cyclic voltammetry linear scan: requires SCAN250 or SCANGEN module
Cyclic voltammetry linear scan high speed	Cyclic voltammetry linear scan high speed: requires SCAN250 or SCANGEN and ADC10M or ADC750 module
Linear sweep voltammetry potentiostatic	Linear sweep voltammetry potentiostatic
Linear sweep voltammetry galvanostatic	Linear sweep voltammetry galvanostatic
Linear polarization	Linear polarization
Hydrodynamic linear sweep	Hydrodynamic linear sweep: requires an R(R)DE connected
Hydrodynamic linear sweep with RRDE	Hydrodynamic linear sweep with RRDE: requires an RRDE and a BA module connected
Sampled DC polarography	Sampled DC polarography: requires IME module
Normal pulse voltammetry	Normal pulse voltammetry: requires IME module
Differential pulse voltammetry	Differential pulse voltammetry: requires IME module
Differential normal pulse voltammetry	Differential normal pulse voltammetry: requires IME module

Figure 192 The Default procedures

It is possible to expand the **Default procedures** location in the panel on the left-hand side to show groups of procedures that use the same type of experimental conditions *(see Figure 193, page 166)*.

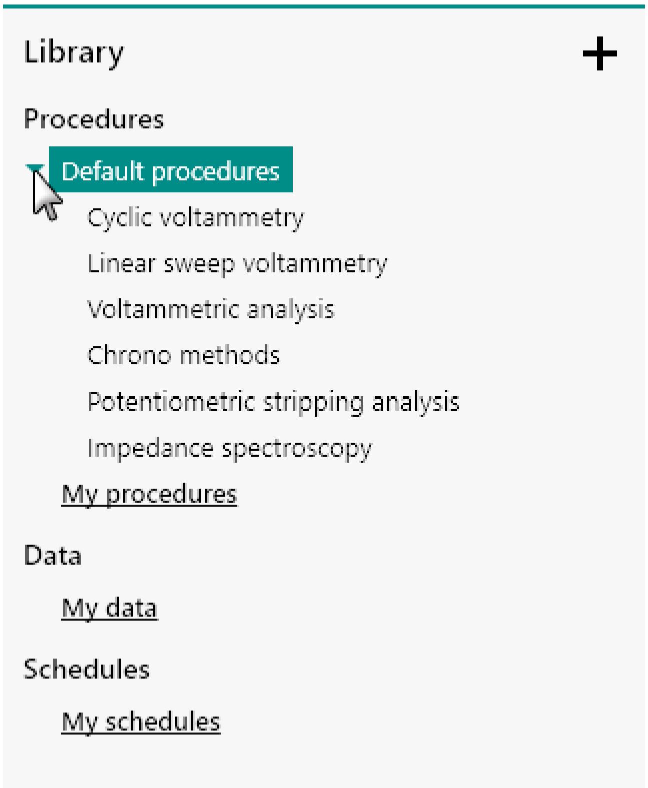


Figure 193 Expanding the Default procedures

Selecting one of the groups in the **Default procedures** will reduce the number of procedures shown in the panel on the right-hand side *(see Figure 194, page 166)*.

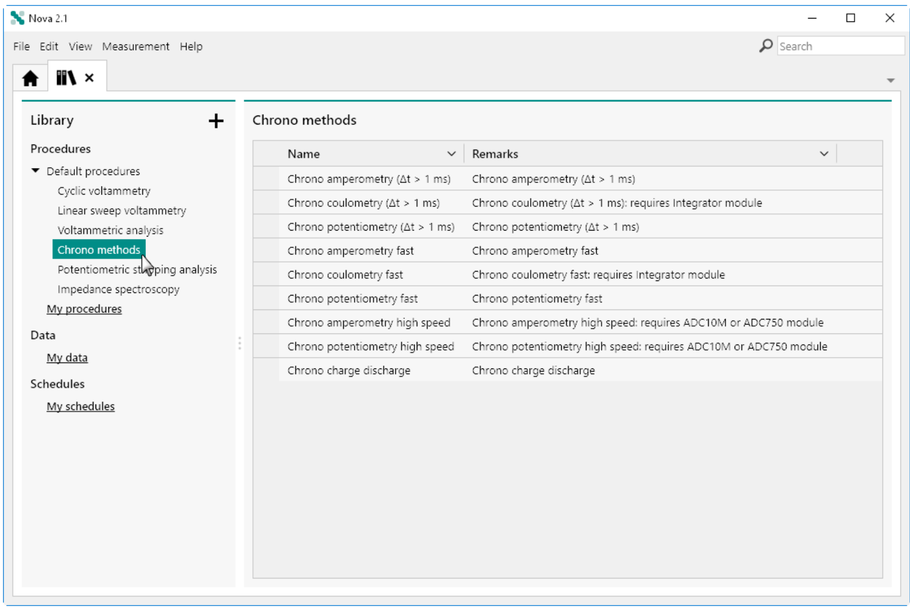


Figure 194 Selecting a group in the Default procedures reduces the number of procedures displayed

6.2 Add location

It is possible to add one or more locations for either procedures, data or schedules by clicking the + button and selecting the required type of location (procedure or data) from the popout menu *(see Figure 195, page 167)*.

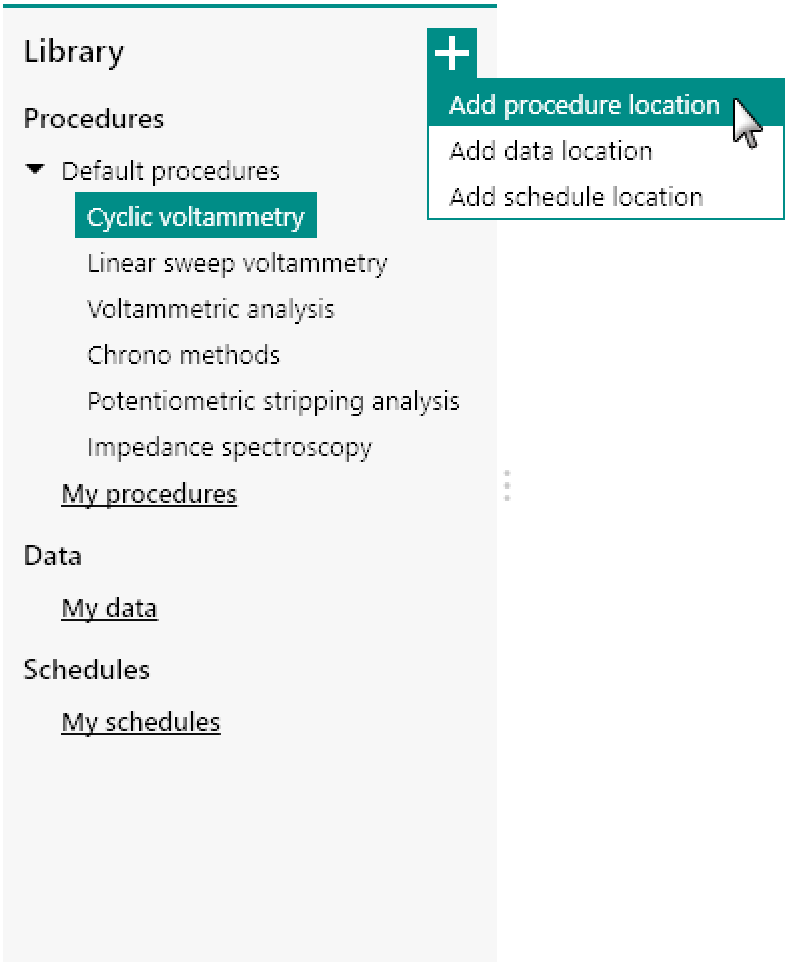


Figure 195 Adding a location to the Library

A Windows folder selection window will be displayed. Using this control, it is possible to navigate to the folder to be added to the list of locations *(see Figure 196, page 168)*.

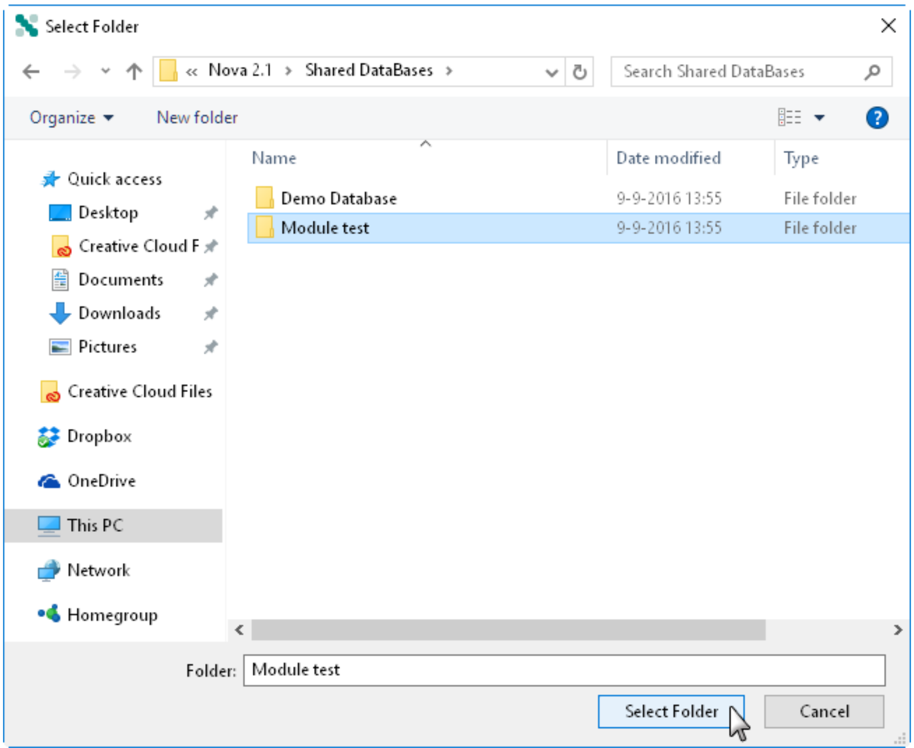


Figure 196 Any folder can be added to the list of locations

The new location will be added to the **Library** and the content of folder will be displayed in the frame on the right-hand side *(see Figure 197, page 168)*.

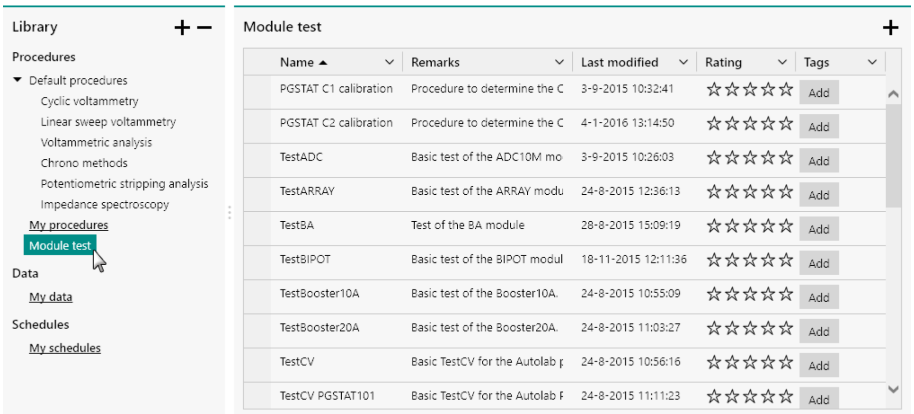


Figure 197 The Module test folder is added as a location to the Library

NOTE

It is possible to add as many new locations as needed.

NOTE

Any sub-folder containing NOVA procedures, data or schedules located in the folder added as location in NOVA will be displayed in the **Library**.

6.3 Default save Location

When more than one location is specified in the **Library**, one of these locations will be used as the default save location. This will be used to save procedures or data unless otherwise specified. By default, the **My procedures**, **My data** and **My schedules** locations are used. It is possible to assign another location as the default save location by selecting it in the **Library** panel and right-clicking it. The context menu can be used to set the selected location as the new default location *(see Figure 198, page 169)*.

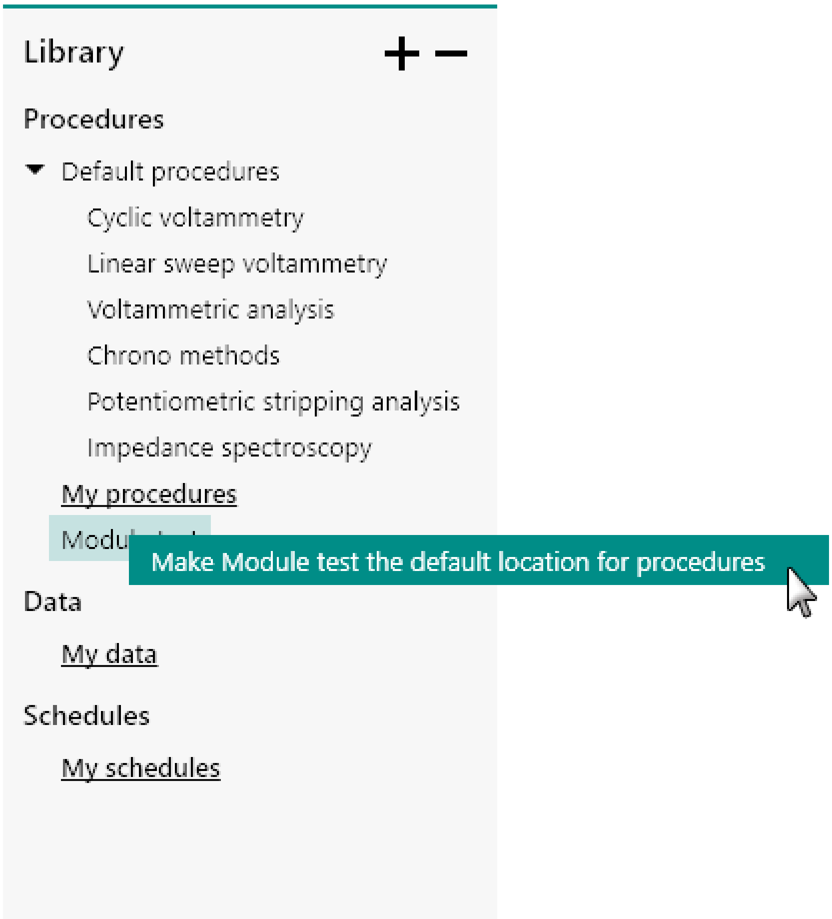


Figure 198 Defining the default save location

The new location will be used as default save location.

NOTE

The default save locations are indicated with an underline font in the **Library** panel.

6.4 Moving files to a new location

When two or more locations are specified in the **Library** panel, it is possible to move files from one location to another by selecting the files and drag and dropping them from the source location to the destination location *(see Figure 199, page 170)*.

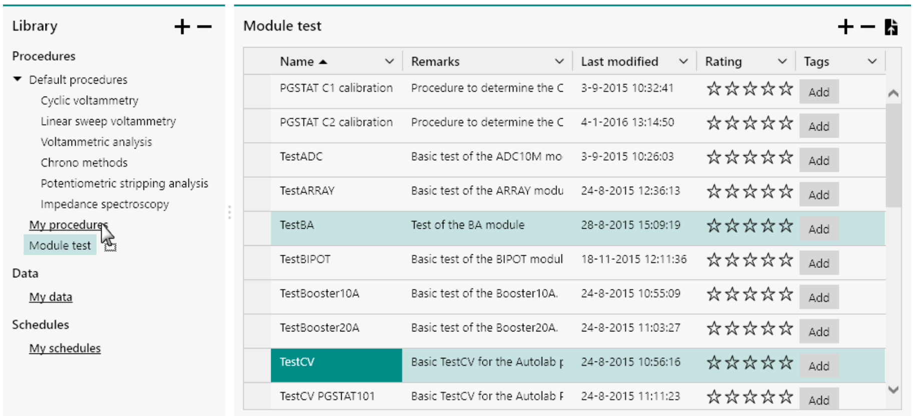


Figure 199 Moving files to a new location using the drag and drop method

The moved files will be removed from the source location and copied to the destination location.

6.5 Remove location

It is possible to remove a location from the **Library**, by clicking the **━** button, located in the top right corner of the **Library** panel. This will remove the highlighted location from the **Library** *(see Figure 200, page 171)*.

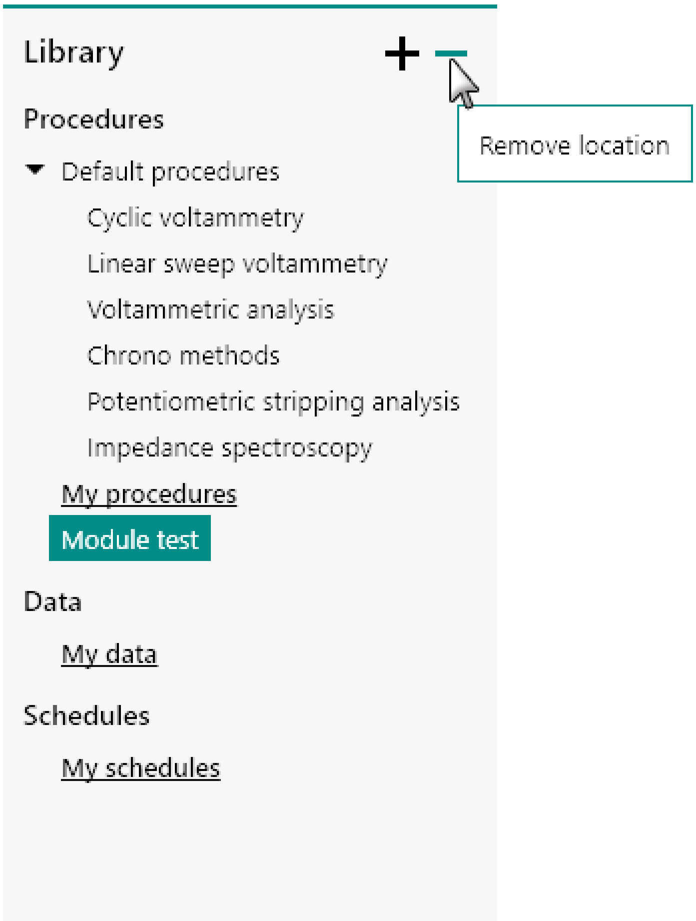


Figure 200 Click the remove button to remove a location

A confirmation message is displayed before the location is removed. Clicking the Yes button removes the location. Clicking the No button cancels the remove action *(see Figure 201, page 171)*.

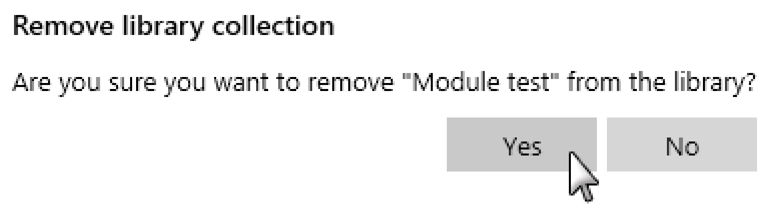


Figure 201 A confirmation message is shown when a Location is removed from the Library

NOTE

Only the location is removed from the **Library**. The content of the folder associated with this location is not deleted from the computer.

6.6 Load from Library

It is possible to open any item from the **Library** by double clicking the corresponding entry in the panel on the right-hand side or by selecting the item and clicking the button in the top-right corner of this panel *(see Figure 202, page 172)*.

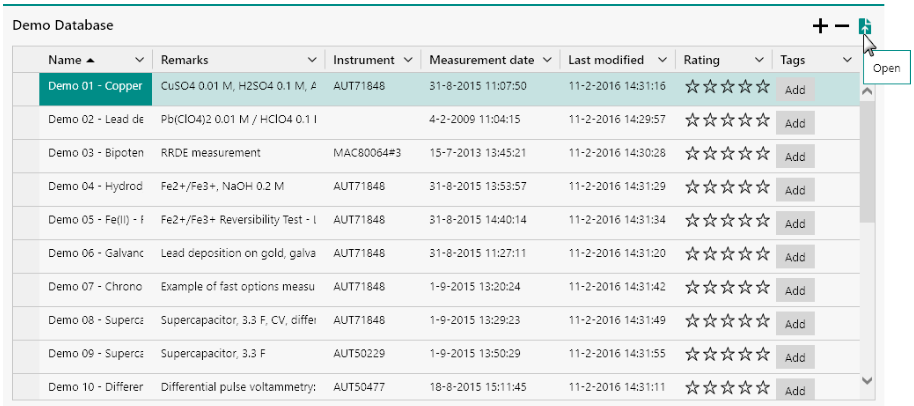


Figure 202 Loading an entry from the Library

The selected item will be opened in a new tab *(see Figure 203, page 172)*.

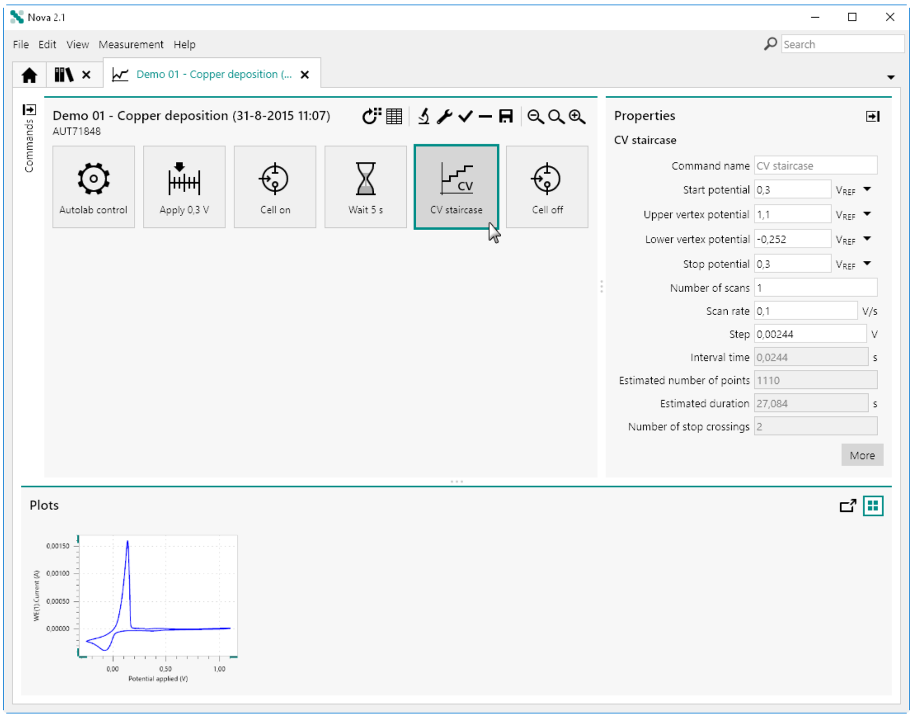


Figure 203 The selected item is opened in a new tab

NOTE

It is possible to open more than one item at the same time by using the multi selection method and clicking the button.

6.7 Edit name and remarks

Using the **Library**, it is possible to change the name and remarks of an item. To change the name or remarks, click the cell to be edited in the table shown in the right-hand side panel of the **Library** to select it and then click it again to go in edit mode *(see Figure 204, page 173)*.

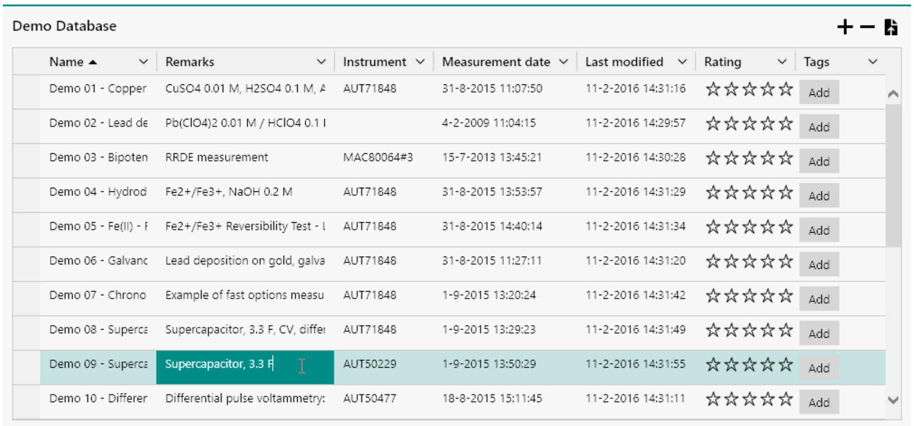


Figure 204 Editing the name or remarks

Edit the name or remarks and click away from the cell or press the **[Enter]** key to validate the change.

CAUTION

Changing the name of an item in the **Library** only changes the display name. The name of the file on the computer remains unchanged.

6.8 Rating and tagging

It is possible to directly edit the rating and tags for items in the **Library**. To specify the rating of an entry in the **Library**, click the highest star in the rating field *(see Figure 205, page 174)*.

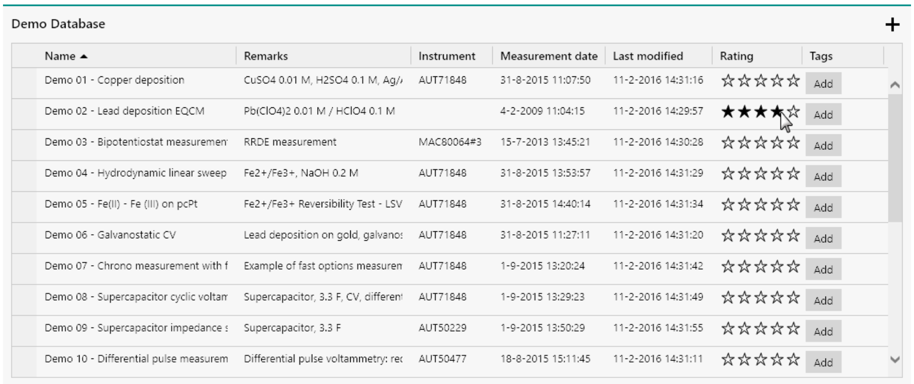
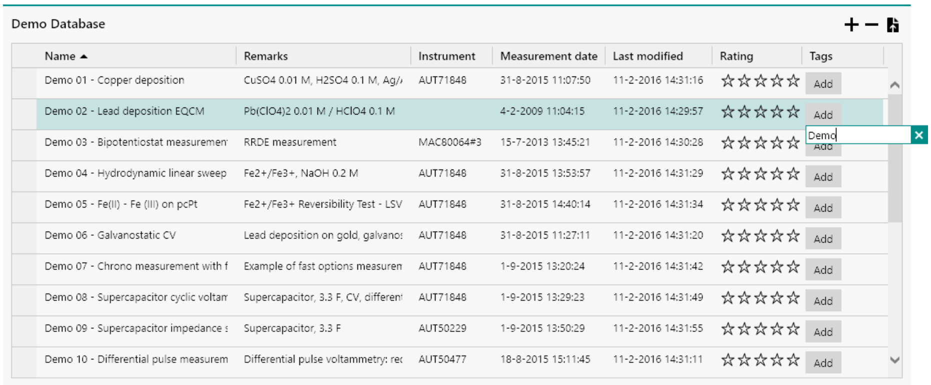


Figure 205 Rating data or procedure items in the Library

It is also possible to edit the tags for a **Library** item. To add a tag, click the Add button and specify the tag to add to the item *(see Figure 206, page 174)*.

Figure 206 Adding tags to data or procedure items in the Library

A popout field will be displayed, allowing specification of a text used for tagging the data or procedure item in the Library *(see Figure 207, page 175)*.

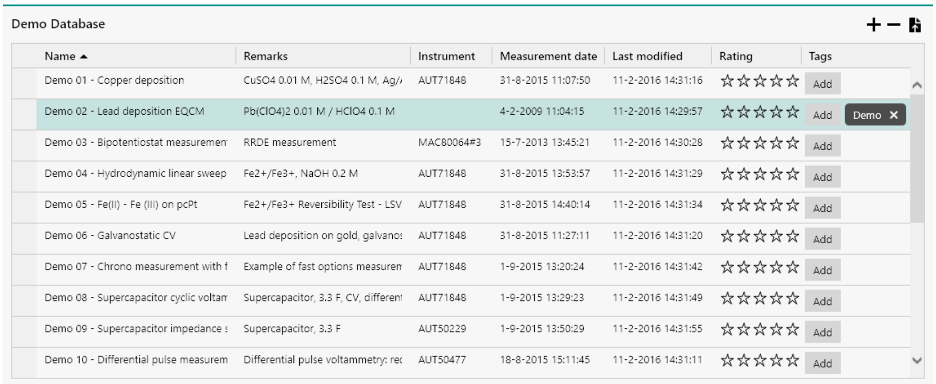


Demo Database

Name ▲	Remarks	Instrument	Measurement date	Last modified	Rating	Tags
Demo 01 - Copper deposition	CuSO4 0.01 M, H2SO4 0.1 M, Ag/	AUT71848	31-8-2015 11:07:50	11-2-2016 14:31:16	☆☆☆☆☆	Add
Demo 02 - Lead deposition EQCM	Pb(ClO4)2 0.01 M / HClO4 0.1 M		4-2-2009 11:04:15	11-2-2016 14:29:57	☆☆☆☆☆	Add
Demo 03 - Bipotentiostat measuremen	RRDE measurement	MAC80064#3	15-7-2013 13:45:21	11-2-2016 14:30:28	☆☆☆☆☆	Demo
Demo 04 - Hydrodynamic linear sweep	Fe2+/Fe3+, NaOH 0.2 M	AUT71848	31-8-2015 13:53:57	11-2-2016 14:31:29	☆☆☆☆☆	Add
Demo 05 - Fe(II) - Fe (III) on pcPt	Fe2+/Fe3+ Reversibility Test - LSV	AUT71848	31-8-2015 14:40:14	11-2-2016 14:31:34	☆☆☆☆☆	Add
Demo 06 - Galvanostatic CV	Lead deposition on gold, galvano:	AUT71848	31-8-2015 11:27:11	11-2-2016 14:31:20	☆☆☆☆☆	Add
Demo 07 - Chrono measurement with f	Example of fast options measuren	AUT71848	1-9-2015 13:20:24	11-2-2016 14:31:42	☆☆☆☆☆	Add
Demo 08 - Supercapacitor cyclic voltan	Supercapacitor, 3.3 F, CV, differen	AUT71848	1-9-2015 13:29:23	11-2-2016 14:31:49	☆☆☆☆☆	Add
Demo 09 - Supercapacitor impedance :	Supercapacitor, 3.3 F	AUT50229	1-9-2015 13:50:29	11-2-2016 14:31:55	☆☆☆☆☆	Add
Demo 10 - Differential pulse measurem	Differential pulse voltammetry: rec	AUT50477	18-8-2015 15:11:45	11-2-2016 14:31:11	☆☆☆☆☆	Add

Figure 207 The tag can be specified in the popout field

The tag will be added to the Tags column in the Library *(see Figure 208, page 175)*.



Demo Database

Name ▲	Remarks	Instrument	Measurement date	Last modified	Rating	Tags
Demo 01 - Copper deposition	CuSO4 0.01 M, H2SO4 0.1 M, Ag/	AUT71848	31-8-2015 11:07:50	11-2-2016 14:31:16	☆☆☆☆☆	Add
Demo 02 - Lead deposition EQCM	Pb(ClO4)2 0.01 M / HClO4 0.1 M		4-2-2009 11:04:15	11-2-2016 14:29:57	☆☆☆☆☆	Add Demo ✕
Demo 03 - Bipotentiostat measuremen	RRDE measurement	MAC80064#3	15-7-2013 13:45:21	11-2-2016 14:30:28	☆☆☆☆☆	Add
Demo 04 - Hydrodynamic linear sweep	Fe2+/Fe3+, NaOH 0.2 M	AUT71848	31-8-2015 13:53:57	11-2-2016 14:31:29	☆☆☆☆☆	Add
Demo 05 - Fe(II) - Fe (III) on pcPt	Fe2+/Fe3+ Reversibility Test - LSV	AUT71848	31-8-2015 14:40:14	11-2-2016 14:31:34	☆☆☆☆☆	Add
Demo 06 - Galvanostatic CV	Lead deposition on gold, galvano:	AUT71848	31-8-2015 11:27:11	11-2-2016 14:31:20	☆☆☆☆☆	Add
Demo 07 - Chrono measurement with f	Example of fast options measuren	AUT71848	1-9-2015 13:20:24	11-2-2016 14:31:42	☆☆☆☆☆	Add
Demo 08 - Supercapacitor cyclic voltan	Supercapacitor, 3.3 F, CV, differen	AUT71848	1-9-2015 13:29:23	11-2-2016 14:31:49	☆☆☆☆☆	Add
Demo 09 - Supercapacitor impedance :	Supercapacitor, 3.3 F	AUT50229	1-9-2015 13:50:29	11-2-2016 14:31:55	☆☆☆☆☆	Add
Demo 10 - Differential pulse measurem	Differential pulse voltammetry: rec	AUT50477	18-8-2015 15:11:45	11-2-2016 14:31:11	☆☆☆☆☆	Add

Figure 208 The tag is added to the item in the Library

NOTE

It is possible to remove a tag by clicking the small X next to tag Demo ✕.

6.9 Preview plot

All data items in the **Library** provide a plot preview in the tooltip *(see Figure 209, page 176)*.

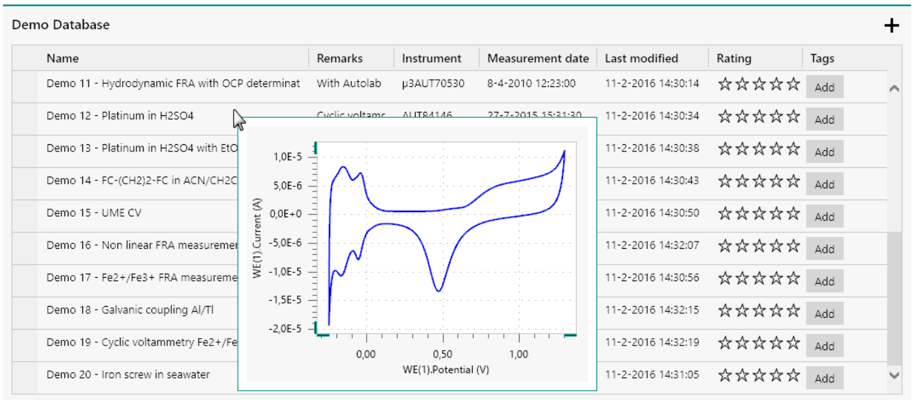


Figure 209 A plot preview is displayed in a tooltip

The plot preview is automatically generated when the data set is saved. By default, the first plot of the data set is used to create the plot preview, however if needed the preview plot can be edited.

NOTE

Measurements performed with older versions of NOVA do not have a preview plot. This plot can be generated when changes to older files are saved in the current version of NOVA.

NOTE

More information on specifying the preview plot can be found in *Chapter 11.7*.

6.10 Column visibility

For each type of location, the visibility of the columns shown in the **Library** can be edited. To hide a visible column in the **Library**, right-click the column header and select the column to hide from the context menu *(see Figure 210, page 177)*.

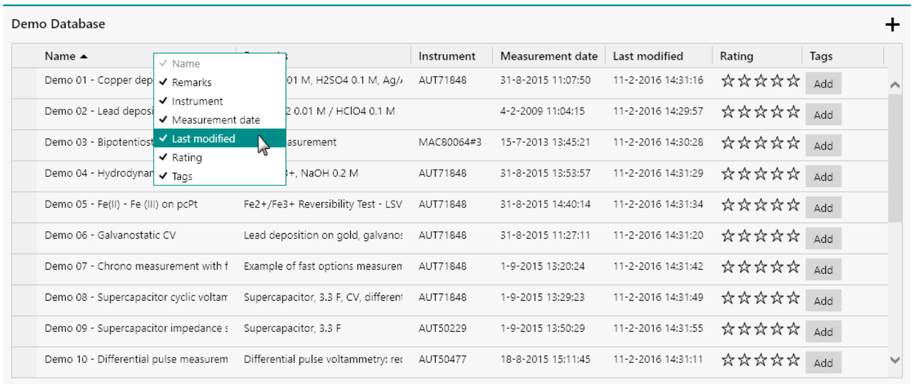


Figure 210 Right-click the column header to hide a visible column

The column will be hidden *(see Figure 211, page 177)*.

Figure 211 The column is hidden

To make a hidden column visible again, right-click the column header and select the hidden column from the context menu *(see Figure 212, page 178)*.

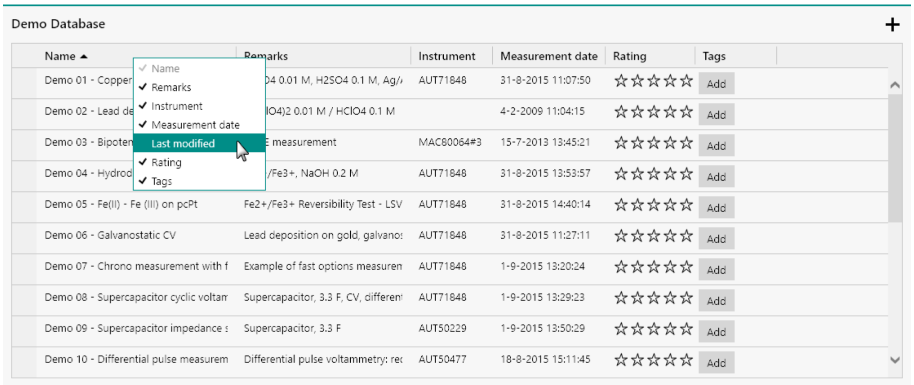


Figure 212 Hidden columns can be displayed again

NOTE

It is not possible to hide the *Name* column.

6.11 Filtering the Library

The columns used to display the items in the **Library** can be used for filtering. To filter content of a column, click the ˅ button located in the right corner of the column header *(see Figure 213, page 178)*.

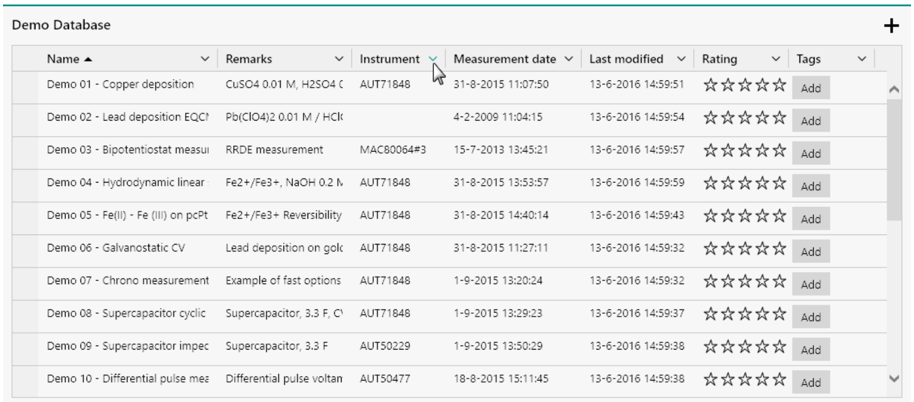


Figure 213 The columns displayed in the Library can be filtered

When the ˅ button is clicked, a menu will appear below the button, providing a list of filters options which can be selected to filter the content of the column based on the specified argument. Four type of filters are available:

- **Alphanumeric filter:** this filter provides the possibility to filter the content of the column based on items that start with a letter or number in the selected bracket(s). This filter is available for the **Name** and **Remarks** columns.

- **Enumeration filter:** this filter provides the possibility to filter the content of the column based on the list of available arguments. *Figure 214* shows an example of an enumeration filter, which displays all the available instrument serial numbers. This filter is available for the **Instrument** and **Tags** columns.
- **Date filter:** this filter provides the possibility to filter the content of the column based on a specific date or timeframe. This type of filter is available for the **Measured date** and **Last modified** columns.
- **Rating filter:** this filter provides the possibility to filter the content of the column based on the assigned rating. This type of filter is available for the **Rating** column.

In the example shown in *Figure 214* a list of instrument serial number is provided.

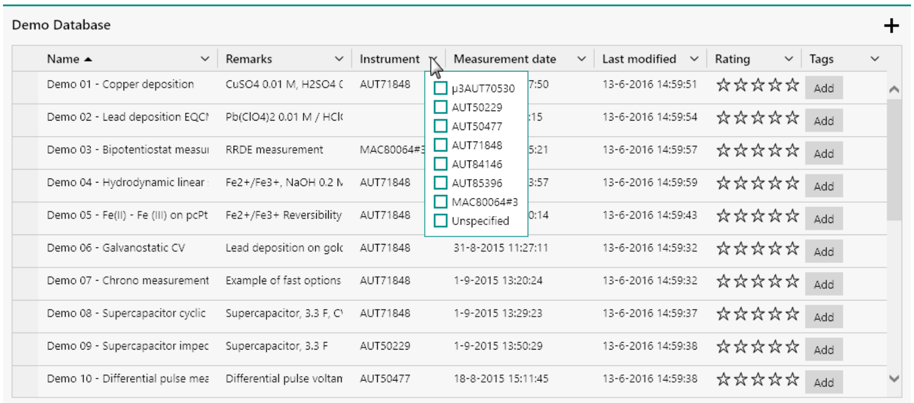


Figure 214 It is possible to filter on the instrument serial number

NOTE

The *Unspecified* filter check box can be used to filter the entries that have no associated value.

Selecting one or more of the available check boxes immediately removes all the entries that do not match the specified filter argument from view, as shown in *Figure 215*.

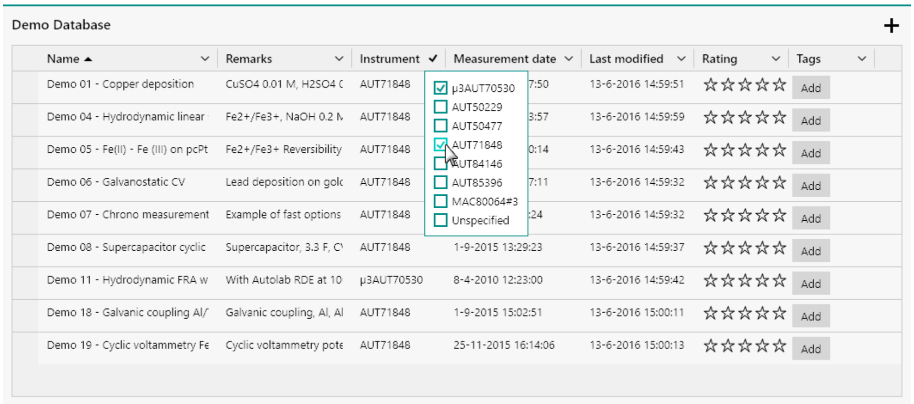


Figure 215 Applying the filter

When a column has a filter active, the ✓ symbol will be shown in the right-hand corner of the column header on which the filter is applied *(see Figure 216, page 180)*.

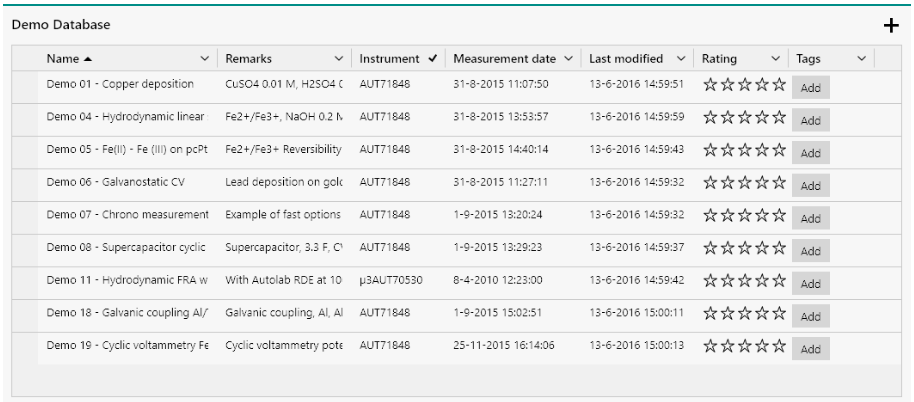


Figure 216 A filtered view of the location

It is possible to adjust the filter at any time by repeating the process described above. Each time a check box is either ticked or unticked, the information displayed in the **Library** will be automatically updated *(see Figure 217, page 180)*.

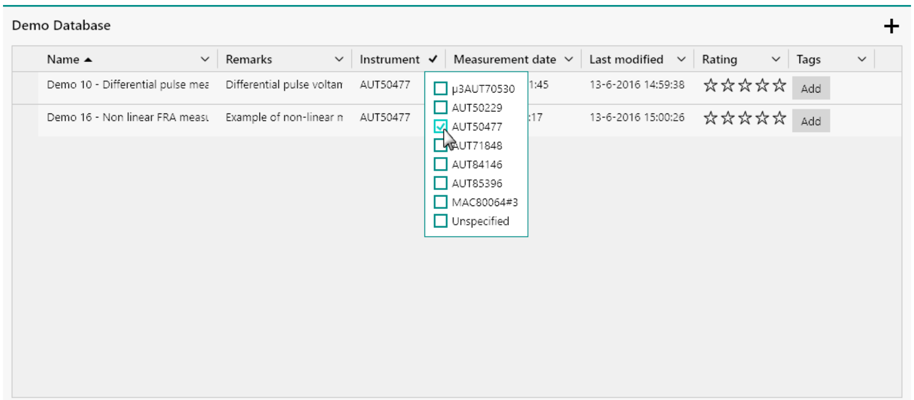


Figure 217 Adjusting the filter

If needed, additional filters can be applied. In that case, the content of the **Library** is adjusted in order to only display the items that match all the filter conditions, as shown in *Figure 218*.

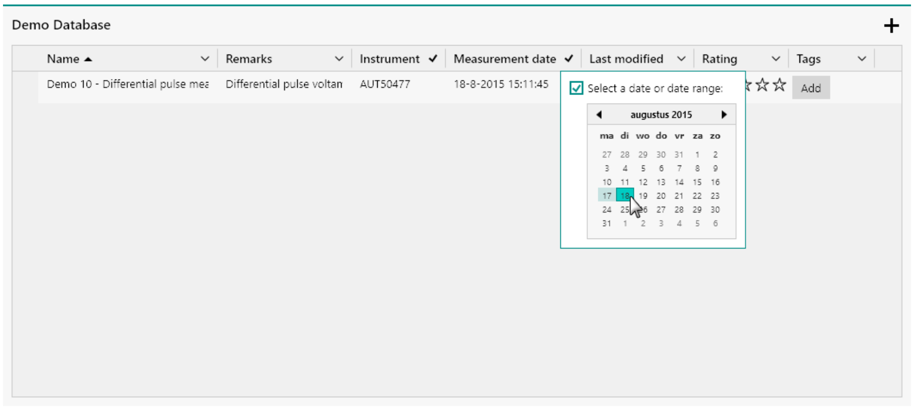


Figure 218 Adding additional filters

NOTE

The specified filter(s) only apply to the active **Location** in the **Library**. For each **Location**, unique filter can be specified.

NOTE

The specified filter(s) remain active until they are cleared or until **NOVA** is closed. To clear an active filter, uncheck all the check boxes in use by this filter.

6.12 Sorting the Library

The columns used to display the items in the **Library** can be used for sorting. To sort the data, click the column header. Clicking the header again toggles from ascending sorting to descending sorting *(see Figure 219, page 182)*.

Figure 219 Sorting the columns in the Library

NOTE

It is possible to sort the Library content using multiple columns by holding the **[SHIFT]** key and clicking the column headers.

6.13 Rearranging Library columns order

If necessary, it is possible to arrange the columns shown in the **Library** in whichever order necessary. To move a column in the **Library**, click the column header and while holding the mouse button, slide the column left or right in the **Library** panel *(see Figure 220, page 182)*.

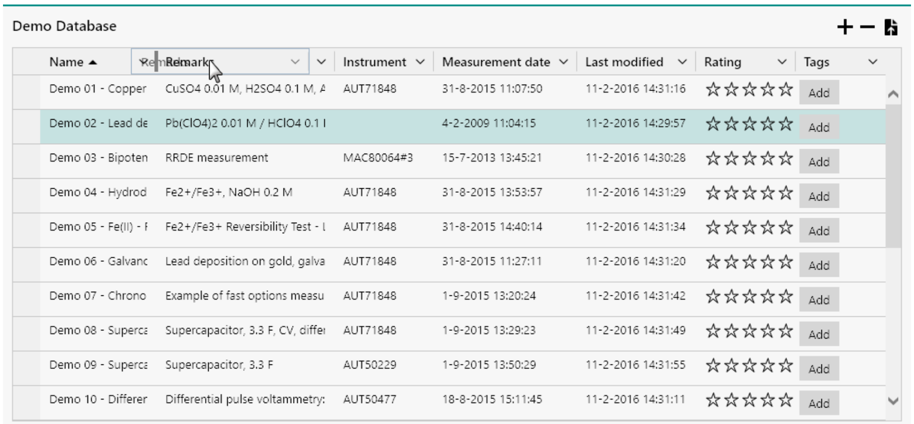


Figure 220 Arranging the column order in the Library panel

Release the mouse button when the column is relocated.

NOTE

The column order can be defined for the Default procedures, Procedures, Data and Schedules locations independently. The order will be used by all locations of the same type.

6.14 Locating files

The **Library** provides the option to quickly locating a file on the computer.

Right-clicking an item in the **Library** displays a context menu that provides the choice to *Show in Windows Explorer*, as shown in *Figure 221*.

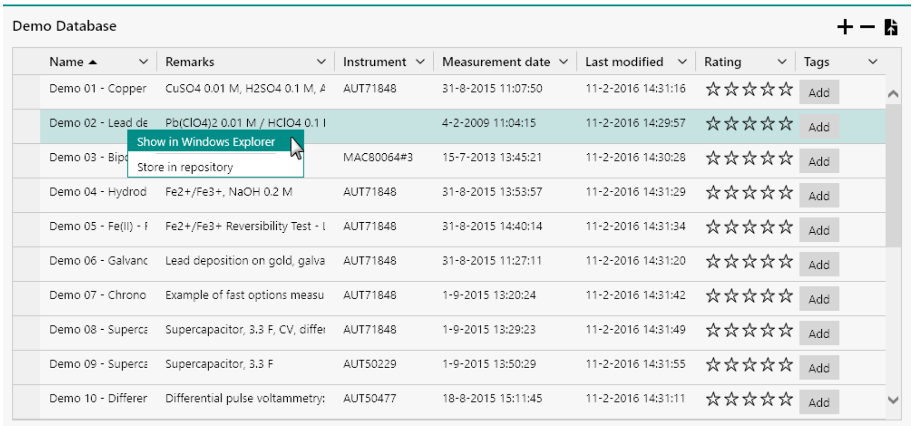


Figure 221 The Show in Windows Explorer option can be used to find a file on the computer

Using this option, a **Windows Explorer** window will be opened, showing the location of the file matching the selected item *(see Figure 222, page 184)*.

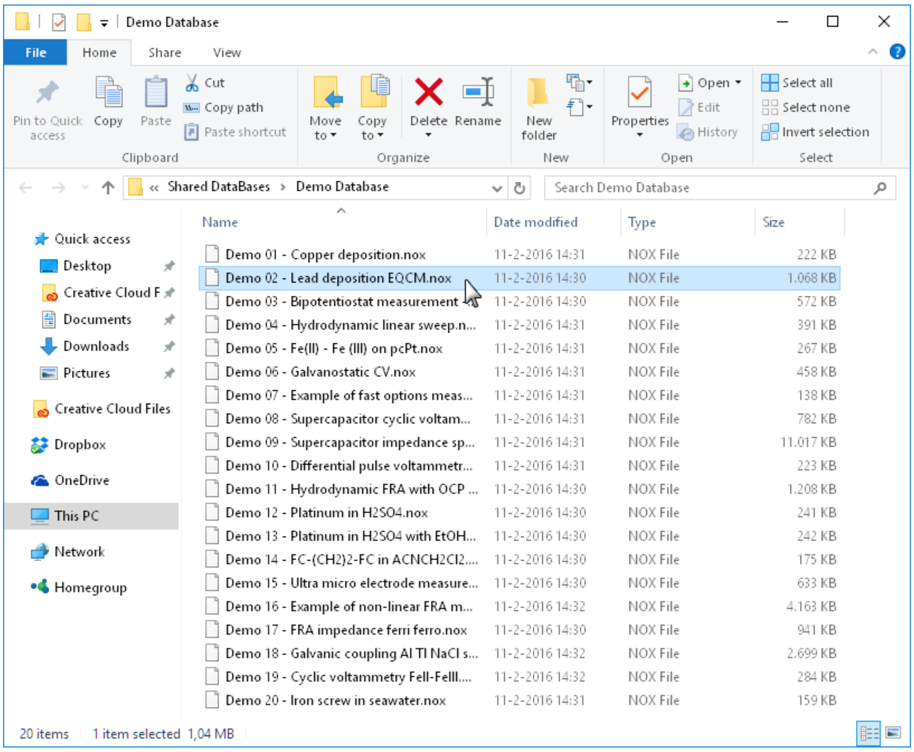


Figure 222 The selected file is shown in Windows Explorer

6.15 Delete files from Library

Through the **Library** interface, it is possible to delete one or more files from a location.

To delete a file from the active location, click the **−** button located in the top right corner of the right-hand side panel *(see Figure 223, page 184)*.

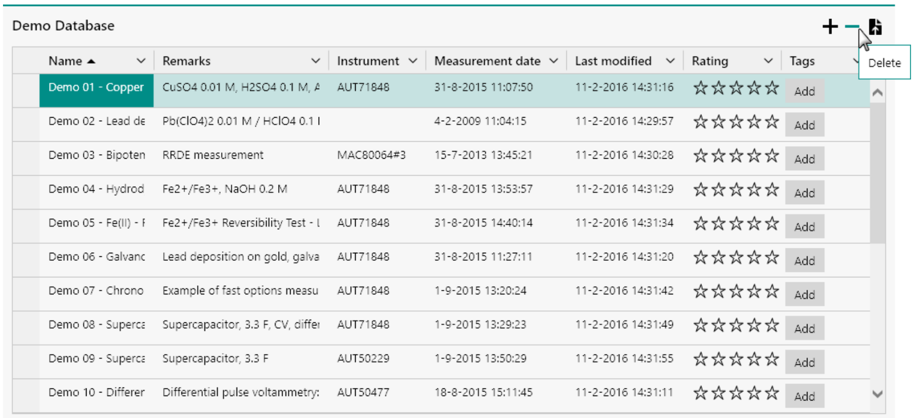


Figure 223 Deleting a file from the Library

A confirmation message is displayed before the file is deleted. Clicking the Yes button deletes the file. Clicking the No button cancels the delete action *(see Figure 224, page 185)*.

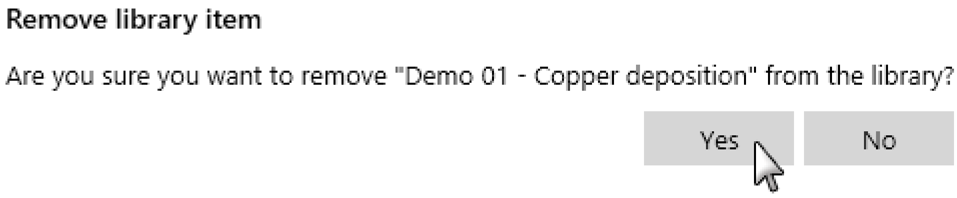


Figure 224 A confirmation is required to delete the file from the Library

CAUTION

Deleting a file from the **Library** also deletes the source file from the computer. The file is moved to the **Recycle Bin** and if needed, it can be restored (if possible).

6.16 The data repository

The **Library** provides access to a data repository. With the repository, it is possible to create one or more internal backups of a data item in the **Library**. This makes it possible to make one or more backups of the original data and recover the original data from one of backups, if required.

NOTE

The repository can only be used for data files.

To store data in the repository, right-click the corresponding item in the **Library** and choose the *Store in repository* option from the context menu *(see Figure 225, page 185)*.

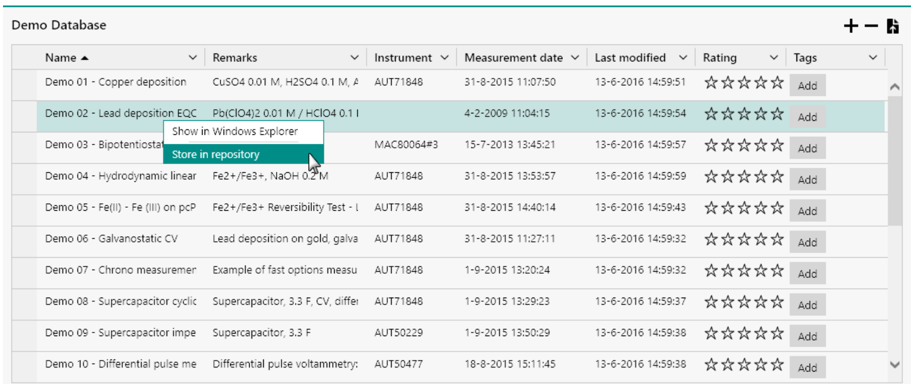


Figure 225 Storing data in the repository

The *Store in repository* option adds a copy of the original data to the **Library** item. This backup is logged with time and date of creation.

NOTE

It is possible to use the *Store in repository* option as many times as required. Each time this option is used, a new backup is added to the **Library** item.

Once a backup has been added to the repository, it is possible to modify the original data set and revert to it at any time by choosing the *Revert from repository* option, available from the context menu *(see Figure 226, page 186)*.

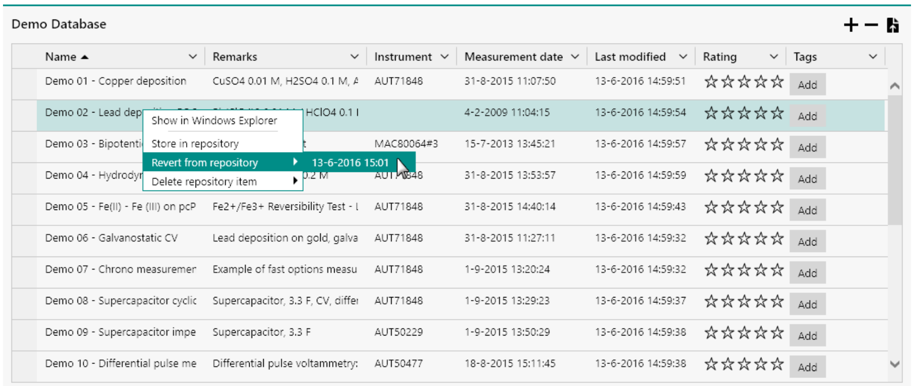


Figure 226 Reverting from the repository

NOTE

In the case of multiple repository backups, the context menu shows all the backups, sorted by time and date.

When repository backups are no longer needed, they can be removed by using the *Delete repository item*, available from the context menu *(see Figure 227, page 187)*.

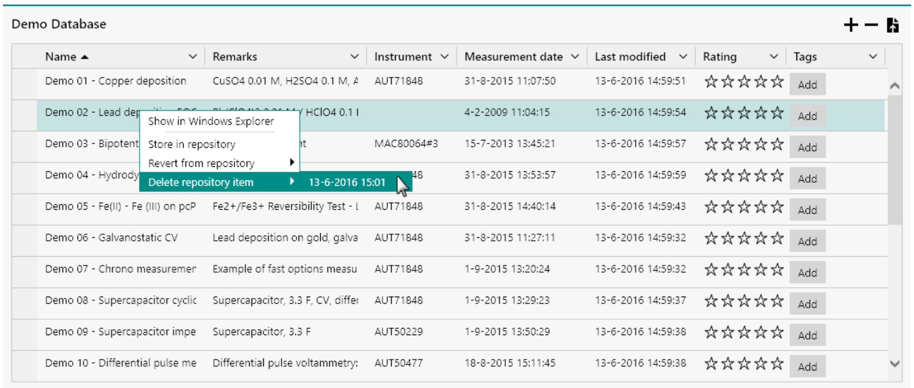


Figure 227 Deleting a repository backup

NOTE

Deleting a repository backup does not remove the source data from **Library**.

6.17 Merge data

An advanced feature of the **Library** provides the means of merging items. When **Library** items are merged, a new item containing the procedures and the data from the merged items will be copied to the new **Library** item. This can be used to involve the data from two or more different measurements in a calculation or other data handling steps. This option also provides the means to merge different procedures into a single one.

NOTE

It is only possible to merge items located in the same **Library** location.

To merge two or more items, select them by holding the **[CTRL]** key and clicking the items to merge *(see Figure 228, page 188)*.

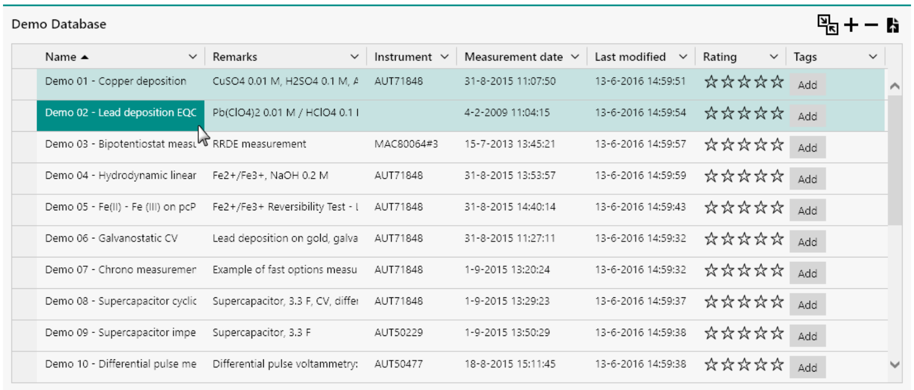


Figure 228 Select two or more items

With two or more items selected, click the button in the top right corner *(see Figure 229, page 188)*.

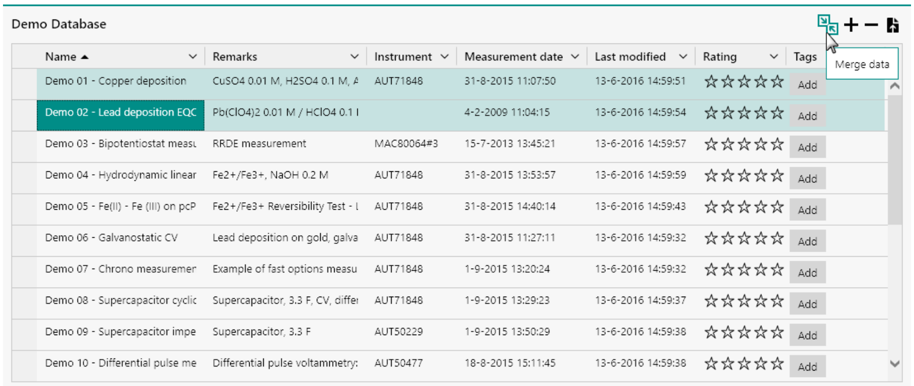


Figure 229 Merging the selected items

NOTE

The button is only visible when two or items are selected in the **Library**.

A message will be displayed, showing the following information *(see Figure 230, page 189)*:

- **Name:** the name of the merged item. By default, NOVA will generate the [MERGED] 'Name of the first selected item' automatically as the name of the merged item.
- **Remarks:** the remarks for the merged item. By default, NOVA fills this input field with the remarks of all the selected items.

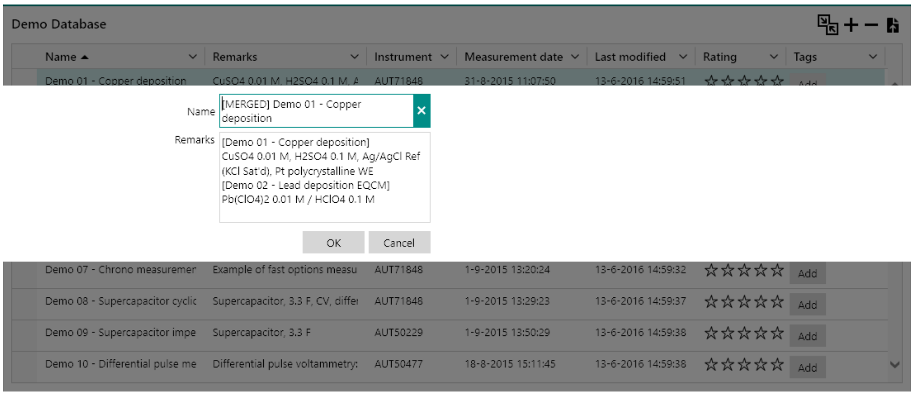


Figure 230 Default name and remarks are generated automatically

It is possible to specify a Name and Remarks for the merged file *(see Figure 231, page 189)*.

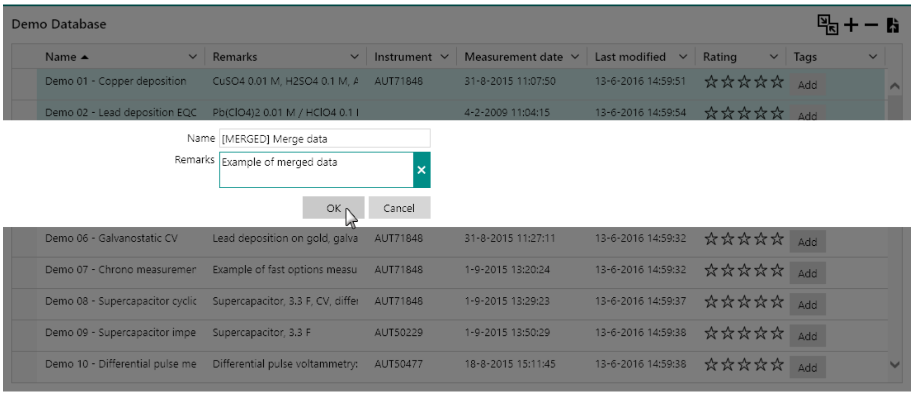


Figure 231 Specifying name and remarks

Click the OK button to merge the items. A new **Library** item will be added to the current location *(see Figure 232, page 189)*.

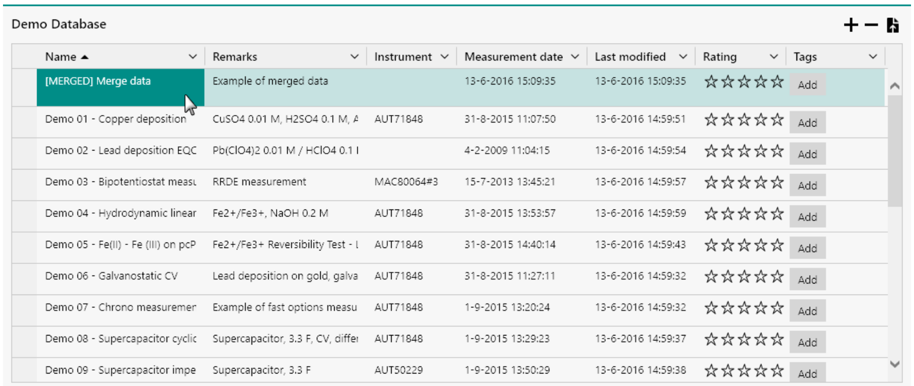


Figure 232 The merged item is added to the Library

NOTE

The source items are not affected by the merging.

It is possible to load the merged item like any other **Library** item. The data or procedure will be loaded in the procedure editor. If data is available, it will be plotted in the **Plots** frame *(see Figure 233, page 190)*.

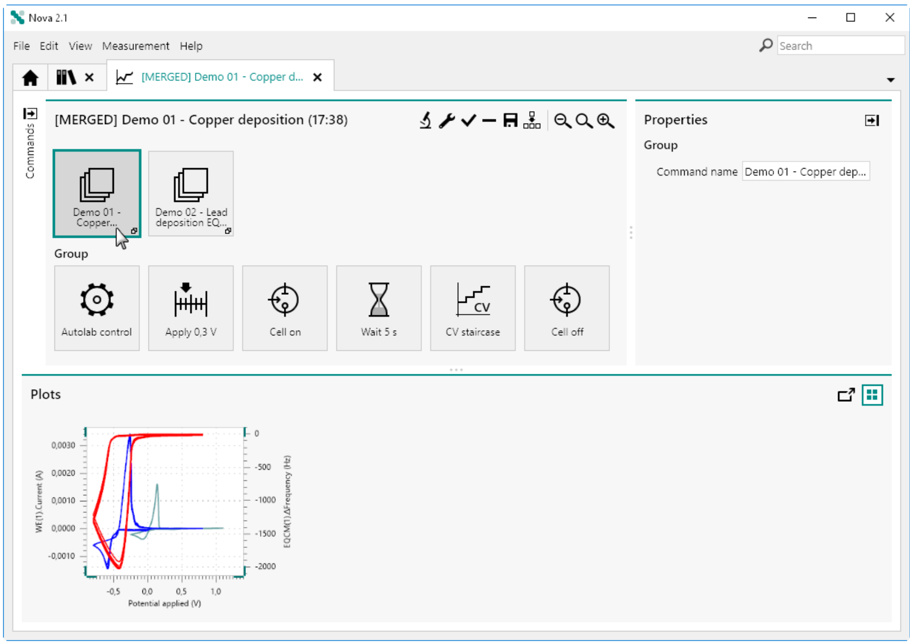


Figure 233 The merged item can be used as data or procedure

6.18 Search function

NOVA provides the means to search for Procedure, Data and Schedule items in the all the *Locations* specified in the **Library**, except the Default procedures. The search function is based on a context insensitive string.

NOTE

The search function can find items based on the **Name** and **Remarks** fields.

To use the search function, a string can be specified in the dedicated input field, located in the top right corner of NOVA *(see Figure 234, page 191)*.

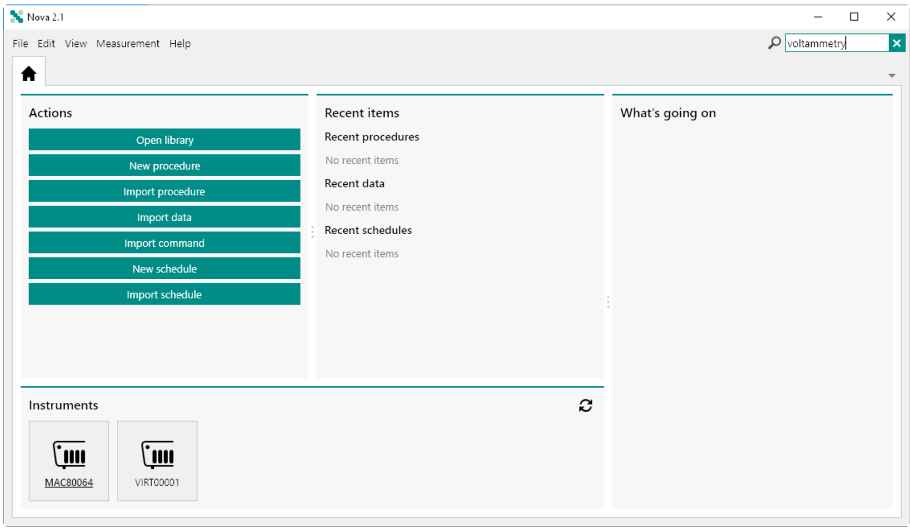


Figure 234 Using the search function

After specifying the search string, press the **[Enter]** key to trigger the search. All Procedure, Data or Schedule items matching the search criteria will be displayed on a separate tab *(see Figure 235, page 191)*.

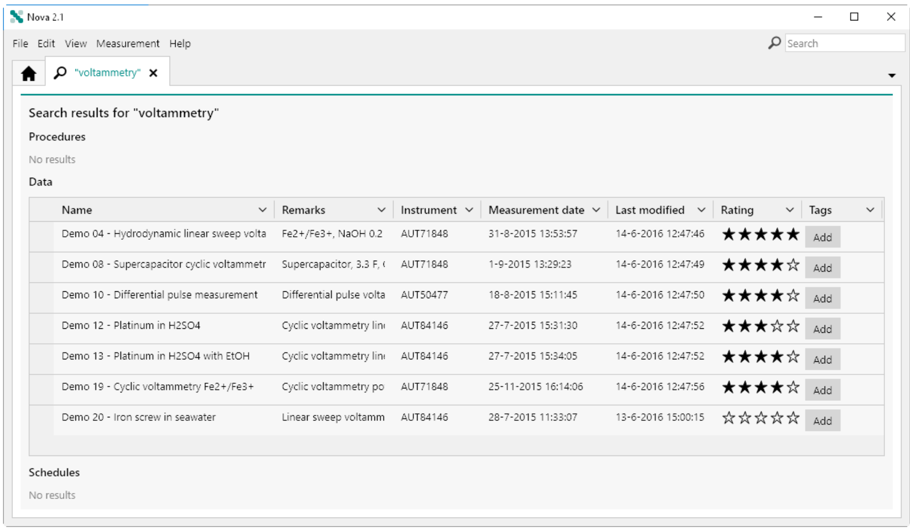


Figure 235 The procedures, data and schedule items matching the specified search string are shown in a dedicated tab

Figure 234 and *Figure 235* illustrate how the search function can be used to find all items that contain the word **voltammetry** in the **Name** or **Remarks**.

NOTE

The results are grouped by type in the results tab.

If needed, wildcards (*) can be used. A wildcard indicates that any word can replace it in the search string. In the example shown in *Figure 236*, two wildcards are used: *** in * with EtOH**. This search string format indicates any word can be replace the * when the search is executed.

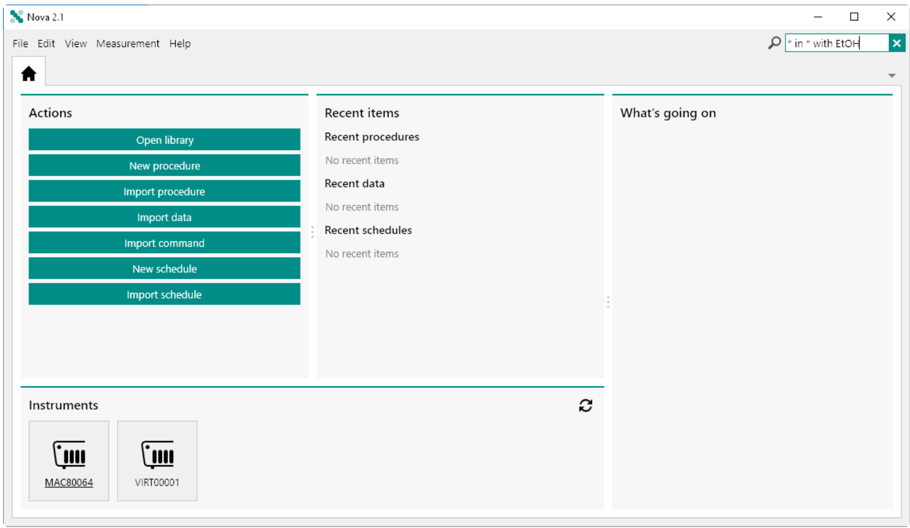


Figure 236 Wildcards can be used in the search field

The results of the search string used in *Figure 236* are shown in *Figure 237*.

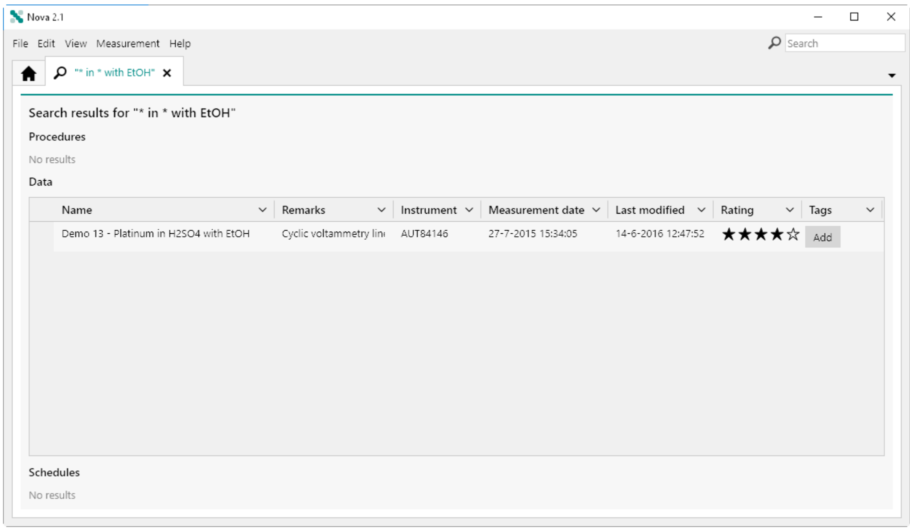


Figure 237 The results are shown in the dedicated tab

7 NOVA commands

NOVA is provided with an extensive set of commands which can be used to modify or create procedures. These commands can be arranged in sequence in order to match the experimental requirements. All the commands provided in NOVA are grouped into different sections:

- **Control:** this group contains commands for user interaction, flow control and external API interfacing. See *Chapter 7.1* for more details.
- **Measurement - general:** this group contains all the commands used to perform basic controls of the instrument. See *Chapter 7.2* for more details.
- **Measurement - cyclic and linear sweep voltammetry:** this group contains all the commands for cyclic and linear sweep voltammetry measurements. See *Chapter 7.3* for more details.
- **Measurement - voltammetric analysis:** this group contains all the commands for voltammetric analysis measurements. See *Chapter 7.4* for more details.
- **Measurement - chrono methods:** this group contains all the commands for time-resolved measurements. See *Chapter 7.5* for more details.
- **Measurement - impedance:** this group contains all the commands for impedance spectroscopy and electrochemical frequency modulation measurements. See *Chapter 7.6* for more details.
- **Data handling:** this group contains all commands designed to process the measured data. See *Chapter 7.7* for more details.
- **Analysis - general:** this group contains all general purpose data analysis commands. See *Chapter 7.8* for more details.
- **Analysis - impedance:** this group contains all the data analysis commands designed for impedance spectroscopy data. See *Chapter 7.9* for more details.
- **Metrohm devices:** this group contains commands that can be used to control supported Metrohm devices connected to the host computer. See *Chapter 7.10* for more details.
- **External devices:** this group contains commands that can be used to control supported external devices. See *Chapter 7.11* for more details.

7.1 Control commands

Control commands can be used to control the flow of events in a procedure, interact with the user or call external functionality. A full description of the commands provided in this group is provided in this chapter.

The available commands are represented by a dedicated symbol *(see Figure 238, page 194)*.

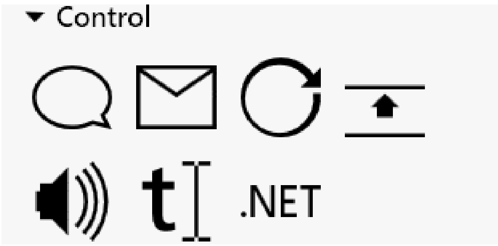
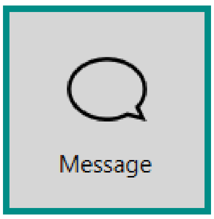


Figure 238 The Control commands

The following commands are available:

- **Message:** a command that can be used to display a message or request an input value *(see Chapter 7.1.1, page 194)*.
- **Send email:** a command that can be used to send an email during a measurement *(see Chapter 7.1.2, page 195)*.
- **Repeat:** a command that can be used to setup a repeat loop *(see Chapter 7.1.3, page 196)*.
- **Increment:** a command that can be used to increment a property during a measurement *(see Chapter 7.1.4, page 210)*.
- **Play sound:** a command that can be used to play a sound *(see Chapter 7.1.5, page 213)*.
- **Build text:** a command that can be used to format a string *(see Chapter 7.1.6, page 214)*.
- **.NET:** a command that can be used to interface to a .NET API *(see Chapter 7.1.7, page 215)*.

7.1.1 Message

	This command can be used to display a message to the user or request an input value from the user. A timeout option is included. The procedure is interrupted until the message is validated by the user.

The details of the command properties of the **Message** command are shown in *Figure 239*:

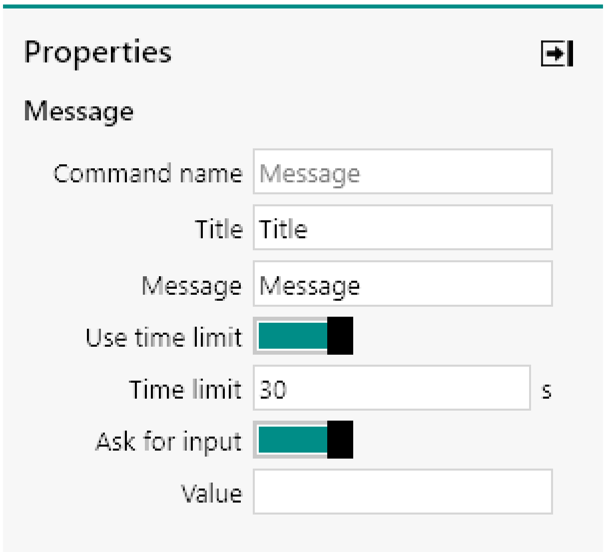
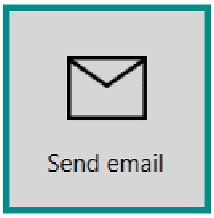


Figure 239 The properties of the Message command

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Title:** the title of the message.
- **Message:** the contents of the message.
- **Use time limit:** a  toggle provided to switch an automatic time-out of the message on or off.
- **Time limit (s):** the time limit, in s, after which the message is cleared if the *Use time limit* toggle is set to on (default 30 s).
- **Ask for input:** a  toggle provided to specify if an input field should be shown in the message.
- **Value:** the default value to show in the input field if the *Ask for input* toggle is set to on.

7.1.2 Send email

	This command can be used to send an email to the specified recipient during a procedure.

The details of the command properties of the **Send email** command are shown in *Figure 240*:

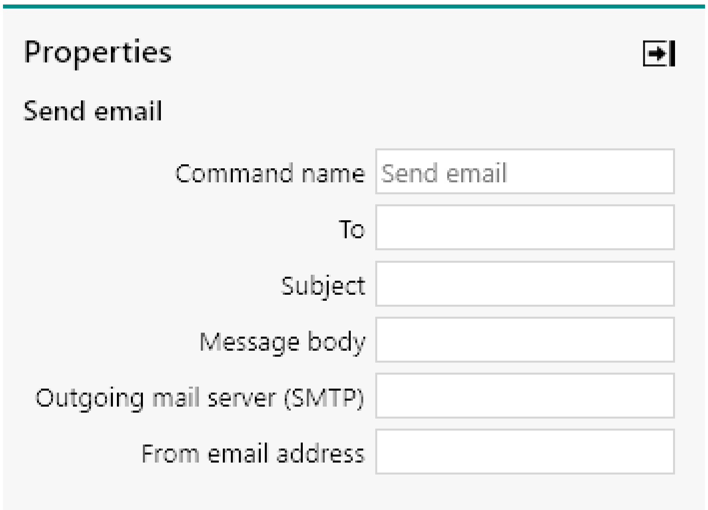
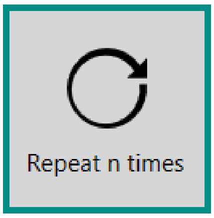


Figure 240 The properties of the Send email command

The following properties are available:

- **Command name:** a user-defined name for the command.
- **To:** the recipient of the email.
- **Subject:** the subject of the email.
- **Message body:** the content of the email.
- **Outgoing mail server (SMTP):** the SMTP server address.
- **From email address:** the email address of the sender.

7.1.3 Repeat

	This command can be used to create a repeat loop to which additional commands can be added.

The **Repeat** command can be used in three different modes, which can be selected using the provided drop-down list *(see Figure 241, page 197)*:

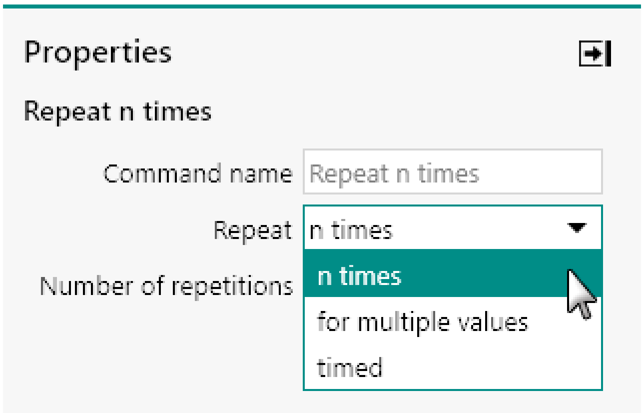


Figure 241 Three modes are provided by the Repeat command

1. Repeat n times (default mode)
2. Repeat for multiple values
3. Timed repeat

NOTE

The **Repeat** command description in the procedure editor is dynamically adjusted in function of the specified mode.

7.1.3.1 Repeat n times

The following properties are available when the command is used in the *Repeat n times* mode *(see Figure 242, page 197)*:

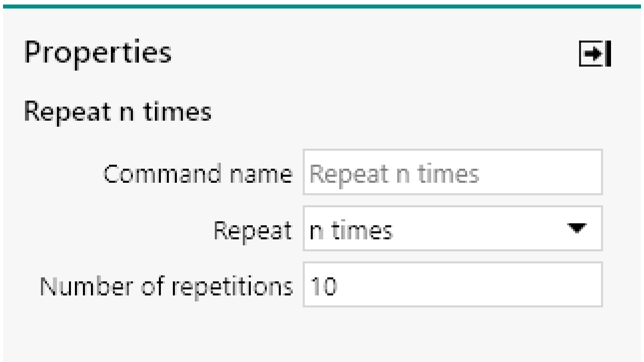


Figure 242 Repeat n times mode properties

- **Command name:** a user-defined name for the command.
- **Number of repetitions:** the number of repetitions in the repeat loop (default 10).

NOTE

The **Number of repetitions** property can be modified in real time.

7.1.3.2 Repeat for multiple values

The following property is available when the command is used in the *Repeat for multiple values* mode *(see Figure 243, page 198)*:

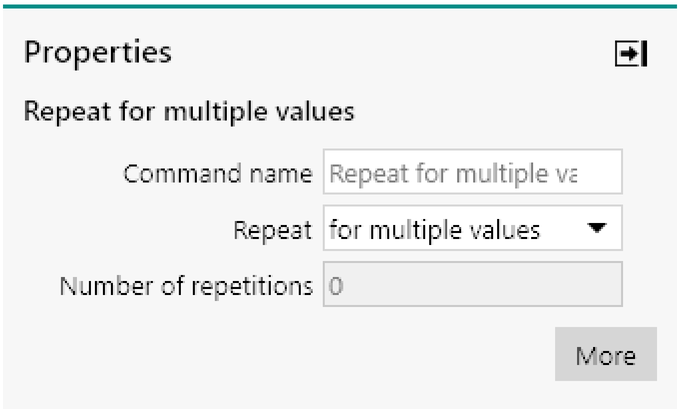


Figure 243 Repeat for multiple values mode properties

- **Command name:** a user-defined name for the command.

Additional properties are defined in a dedicated panel.

Clicking the More button opens a new panel where the properties of the *Repeat for multiple values* mode can be specified *(see Figure 244, page 198)*.

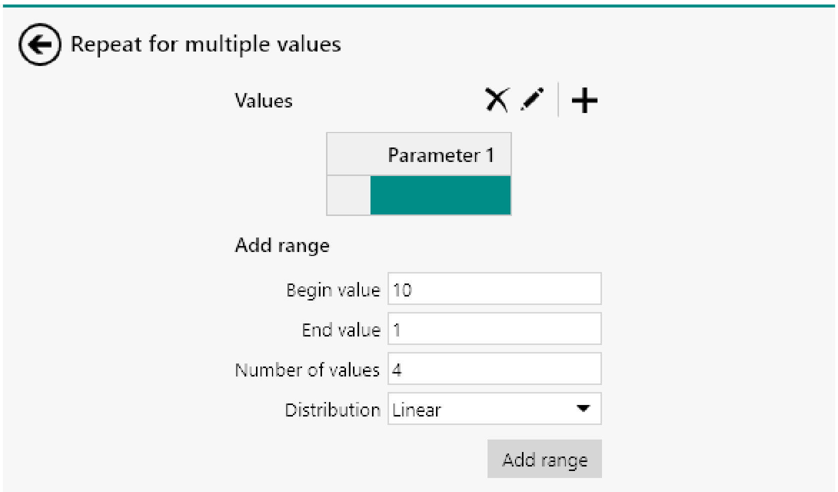


Figure 244 Overview of the Repeat for multiple values mode panel

The **Repeat for multiple values** panel can be used to build a table of values, in one or more columns. During the experiment, the repeat loop will cycle through each row of the table, and use the values of each column in the measurement.

NOTE

All the columns specified in the table must have the same number of elements.

The following actions can be performed in the **Repeat for multiple values** panel:

1. Edit the name of a column
2. Add values manually
3. Add a range of values using the range builder
4. Add additional columns
5. Move columns in the table
6. Delete values from the table
7. Sort the contents of the table
8. Clear the contents of the table
9. Remove columns

7.1.3.2.1 Edit the name of a column

To edit the name of a column header in the table click the ✎ button *(see Figure 245, page 199)*.

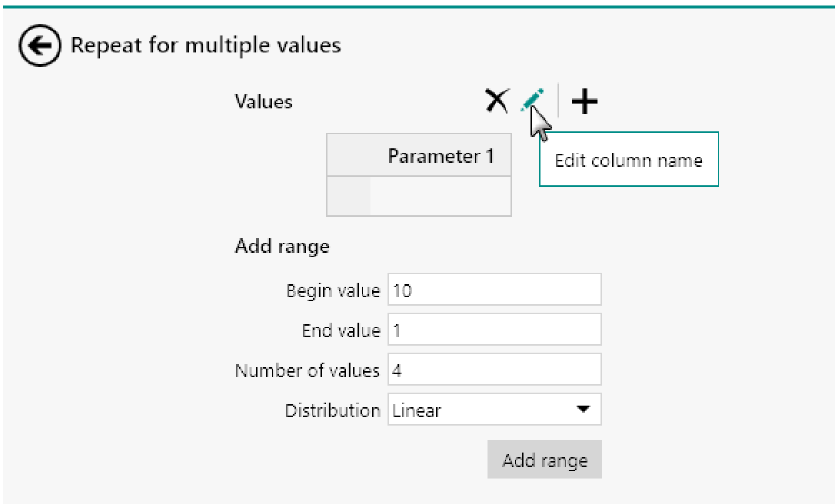


Figure 245 Column headers can be edited

The column header will be highlighted and the name can be edited. Press the **[Enter]** key, the **[Tab]** key or click away from the column header to validate the new name *(see Figure 246, page 200)*.

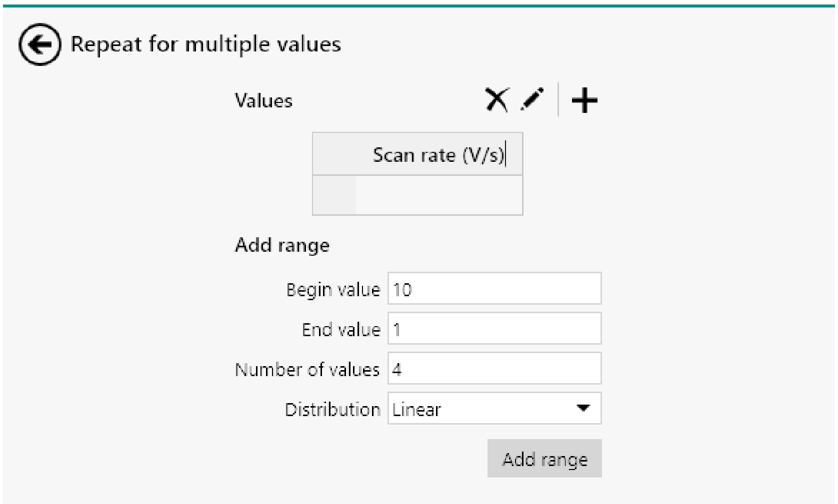


Figure 246 Editing the column header

NOTE

If the table contains more than one column, pressing the **[Tab]** key will validate the name of the selected column header and the column header of the next column will be selected for editing. Pressing the combination of **[Shift]** and **[Tab]** will do the same with the previous column header.

NOTE

If a cell is selected when the ✎ button is clicked, the column header of column containing the selected cell will be highlighted.

7.1.3.2.2 Manually add values to a table

To manually add values to a table, select the first available cell in a column and type the value *(see Figure 247, page 201)*.

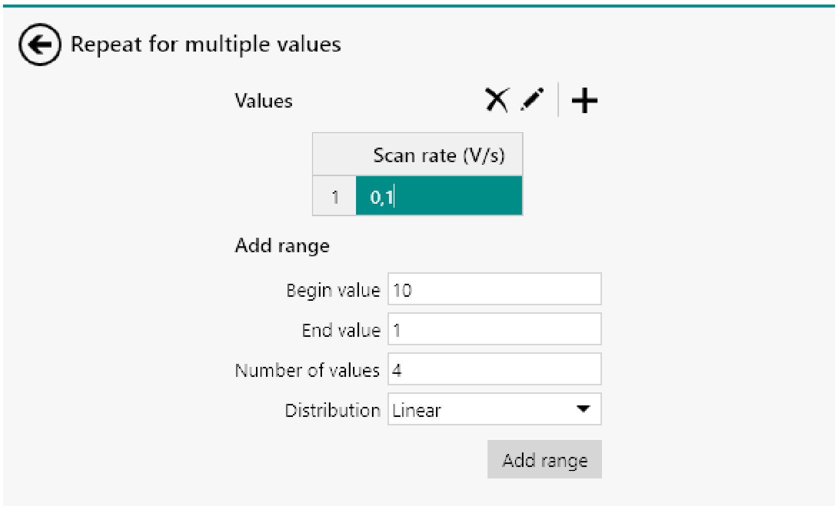


Figure 247 Manually adding values to the table

Press the **[Enter]** key, the **[Tab]** key or click away from the cell to validate the specified value.

NOTE

The **Index** column is automatically created and updated when the table is edited.

7.1.3.2.3 Add values to a table using the Add range option

To add values to a column using the **Add range** option, the properties located below the table must be specified *(see Figure 248, page 201)*.

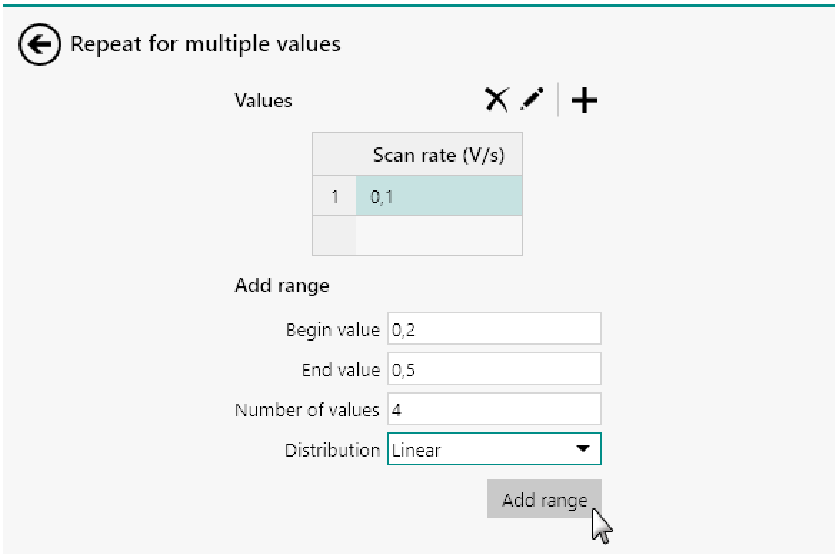


Figure 248 The Add range option can be used to add values to the table

The following properties are available:

- **Begin value:** the first value of the range.
- **End value:** the last value of the range.
- **Number of values/Number of values per decade:** the number of values in the range or the number of points per decade in the range.
- **Distribution:** the distribution used to calculate the range. Four distributions are available, selectable using the provided drop-down list:
  - **Linear:** the range is built using a linear distribution.
  - **Square root:** the range is built using a square root distribution.
  - **Logarithmic:** the range is built using a logarithmic distribution.
  - **Points per decade:** the range is built by calculating the number of decades in the range and by adding the specified number of points per calculated decade. This distribution is also logarithmic.

Table 6 provides an overview of the formulae used to calculate the distributions supported by the **Add range** option.

Table 6 The distributions used in the Add range option

Type	**Increment, Δ**	**Distribution**
Linear	$\Delta = \frac{End - Start}{N - 1}$	$V_i = \left(Start + (i-1)\Delta\right)$
Square root	$\Delta = \frac{\sqrt{End} - \sqrt{Start}}{N - 1}$	$V_i = \left(\sqrt{Start} + (i-1)\Delta\right)^2$
Logarithmic	$\frac{LOG(End) - LOG(Start)}{N - 1}$	$LOG(V_i) = \left(LOG(Start) + (i-1)\Delta\right)$
Points per decade	$\Delta = 10^{-\frac{FLOOR\left(\left\lvert LOG\frac{Start}{End}\right\rvert\right) N_{Decade}}{LOG\frac{Start}{End}}}$	$V_i = \left(Start \cdot \Delta^{(i-1)}\right)$

Click the Add range button to add a range of value to the table *(see Figure 249, page 203)*.

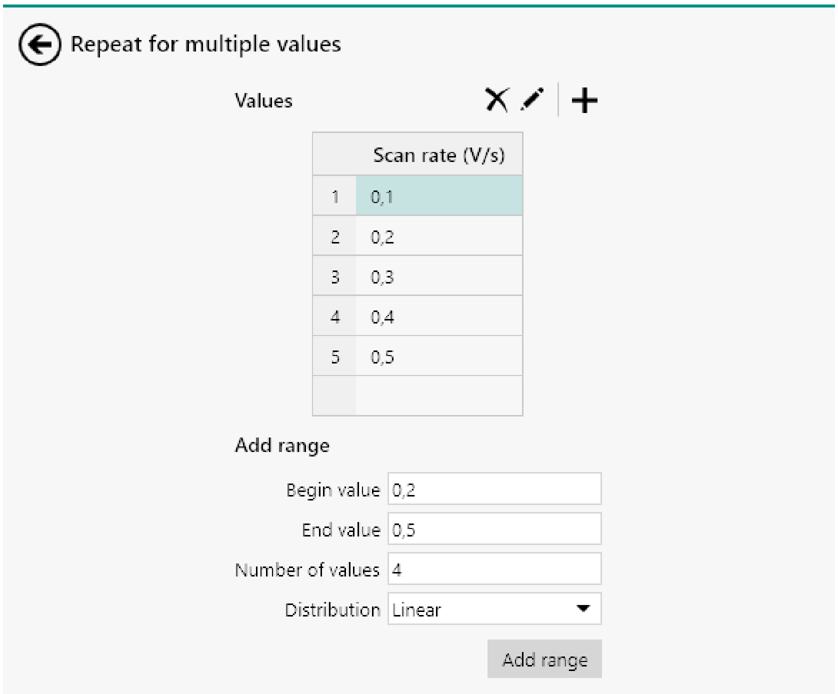


Figure 249 The specified range is added to the table

NOTE

The range is always added below the last value of the selected column of the table.

NOTE

The **Index** column is automatically created and updated when the table is edited.

7.1.3.2.4 Add additional columns

To add extra columns to the table, click the **+** button above the table. A drop-down list will be displayed offering a choice between two options *(see Figure 250, page 204)*:

- **A *value* column:** a column that contains only numbers.
- **A *text* column**: a column that contains anything. To identify this type of column, a small ^{T} is displayed in the column header *(see Figure 250, page 204)*.

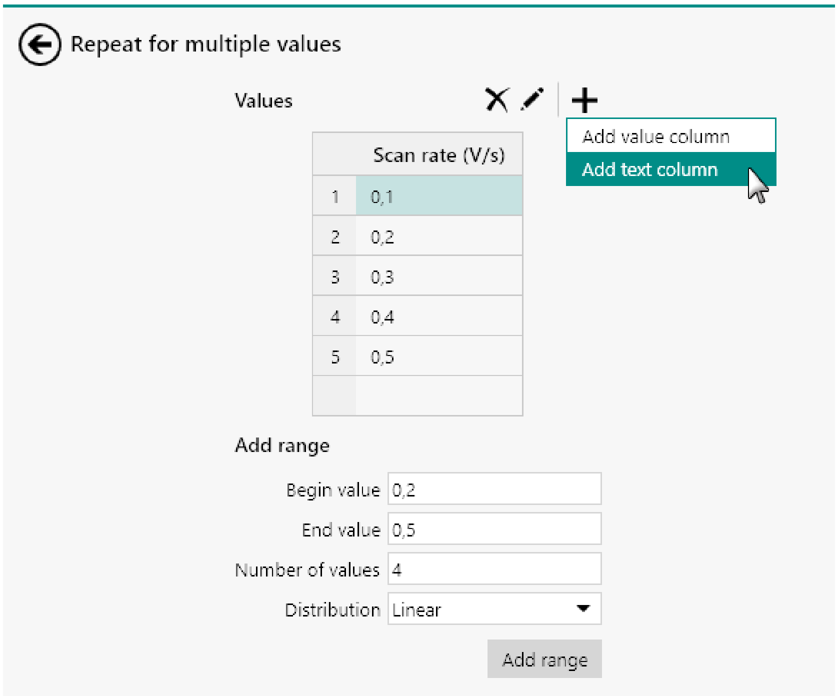


Figure 250 Adding an extra column to the table

An extra column will be added to the table *(see Figure 251, page 204)*.

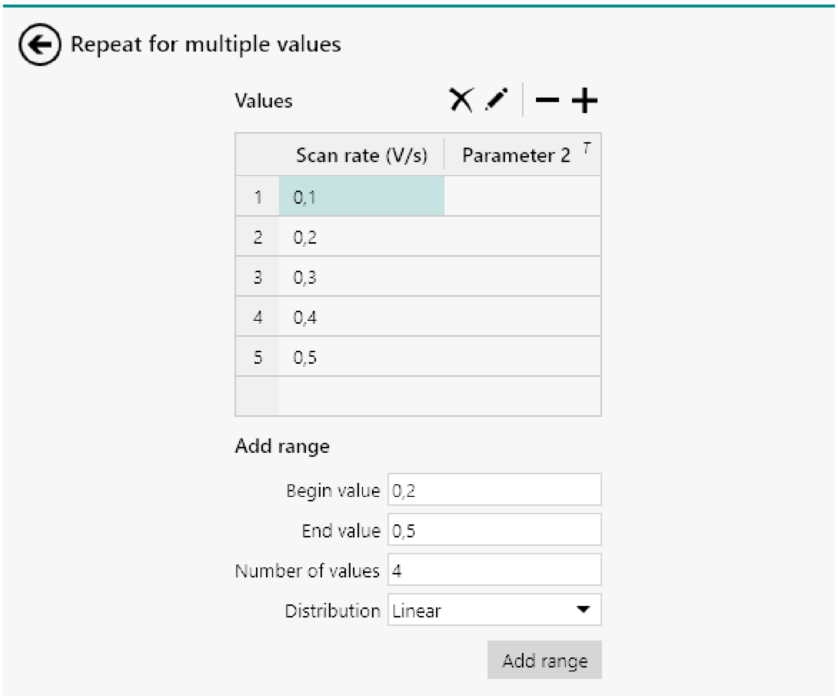


Figure 251 The extra column is added to the table

NOTE

A text column is identified by an uppercase *T* in the column header, as shown in *Figure 251*.

The new column can now be edited *(see Figure 252, page 205)*.

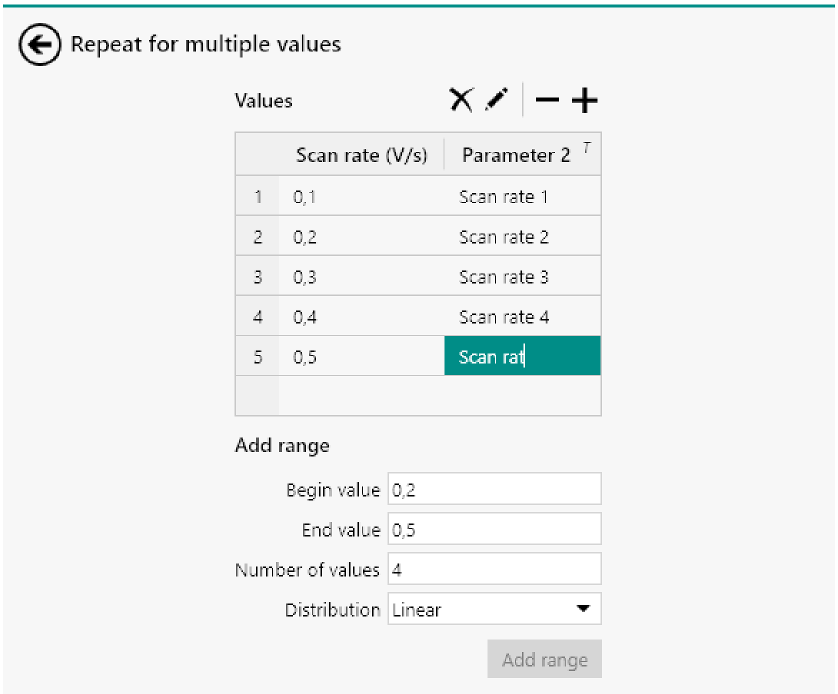


Figure 252 Editing the new column

NOTE

The Add range option cannot be used when editing a *text* column. The cells of this type of column must be edited manually.

7.1.3.2.5 Moving columns in the table

When the table contains two or more columns, it is possible to rearrange the order of the column by clicking a column header and dragging it to another location, while holding the mouse button *(see Figure 253, page 206)*.

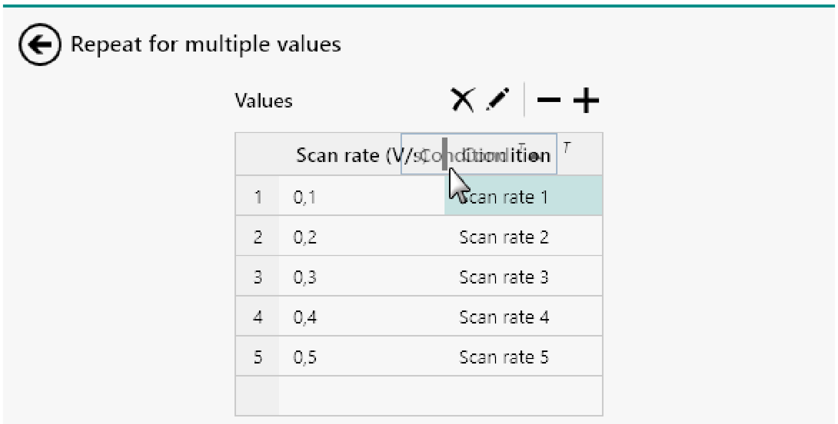


Figure 253 Moving the columns

When the mouse button is released, the column will be repositioned in the table *(see Figure 254, page 206)*.

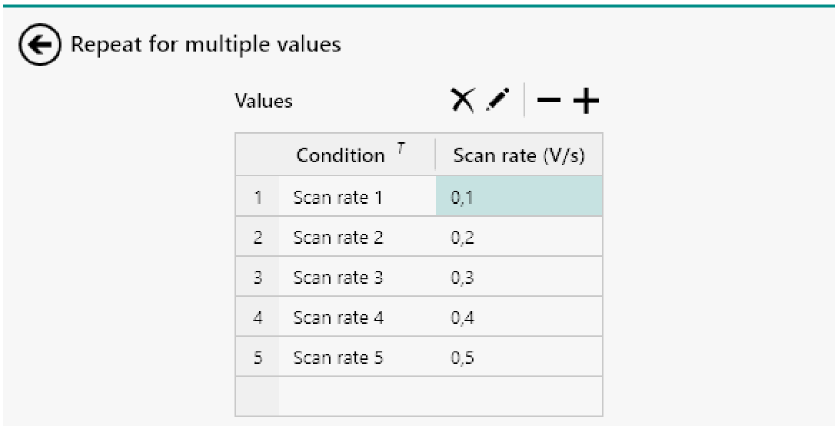


Figure 254 The repositioned column

7.1.3.2.6 Delete values from the table

Values from the table can be removed in two ways:

- A single value from the table can be deleted. This only clears the content of the table cell.
- A complete row from the table can be deleted. This removes the complete row from the table.

To delete a single value from the table, select the cell containing the value and press the **[Delete]** key *(see Figure 255, page 207)*.

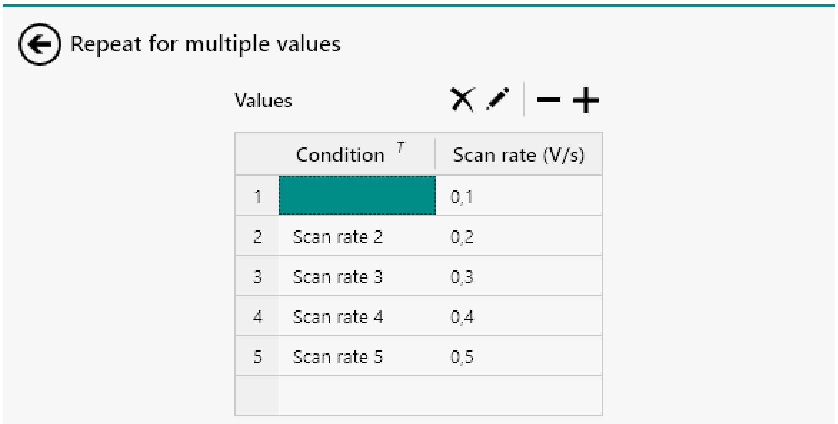


Figure 255 Deleting a value from the table

To delete a complete row of the table, click the index cell in front of the row to select the row and press the **[Delete]** key to delete the selected row of the table *(see Figure 256, page 207)*.

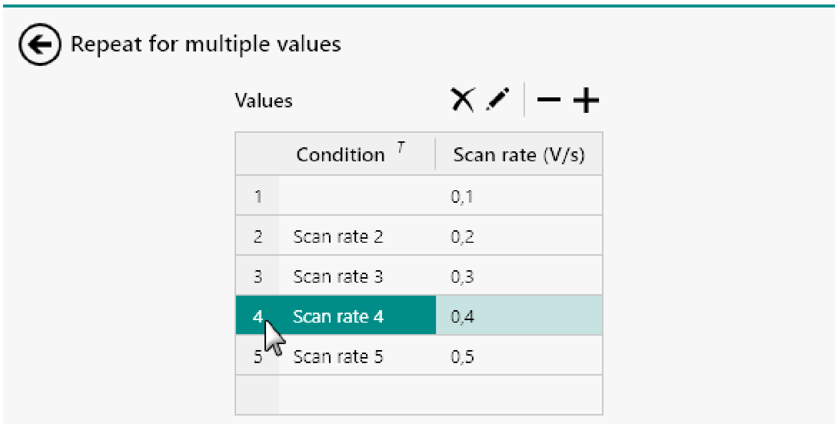


Figure 256 Deleting a complete row from the table

The selected row is completely removed from the table *(see Figure 257, page 207)*.

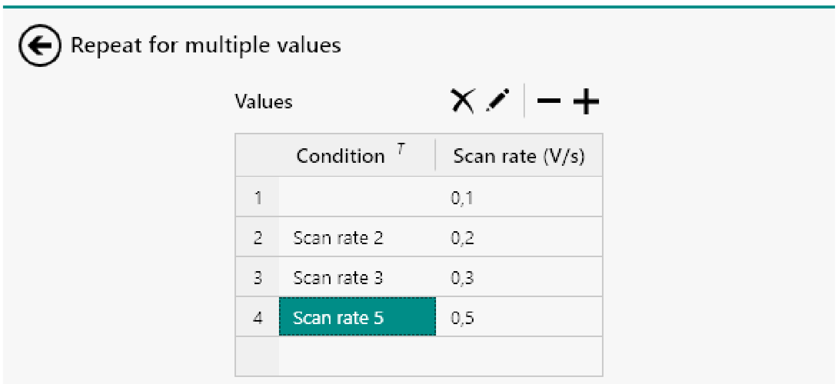


Figure 257 The selected row is removed from the table

7.1.3.2.7 Sorting the contents of the table

It is possible to sort the contents of the table by clicking one of the column header. This will sort the content of the column ascending or descending and the other columns of the table will be sorted based on the new order of the sorted column. Clicking the column header cycles from ascending sorting to descending sorting *(see Figure 258, page 208)*.

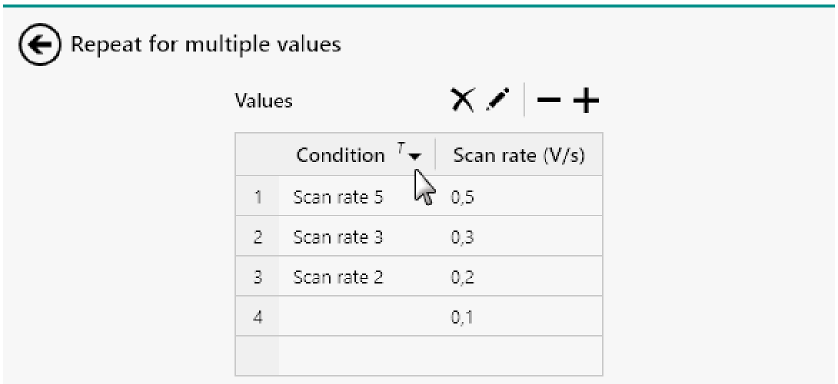


Figure 258 Sorting the columns of the table

NOTE

A column sorted in ascending mode is indicated by the ▼ symbol. A column sorted in descending mode is indicated by the ▲ symbol.

7.1.3.2.8 Clear the contents of a column

It is possible to clear the contents of a column in the table by selecting the column and clicking the X button located above the table *(see Figure 259, page 208)*.

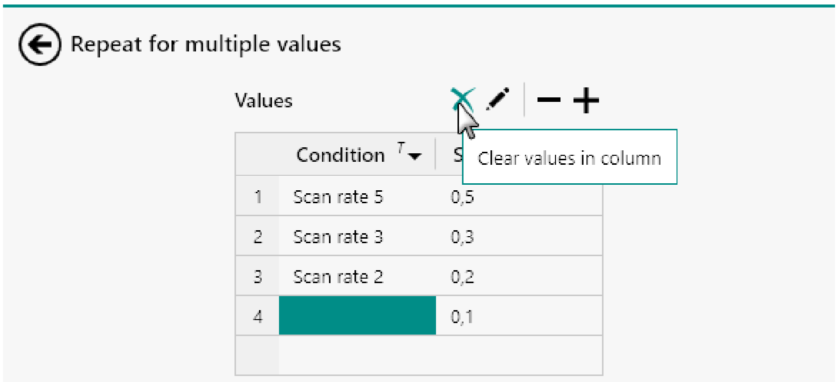


Figure 259 Clearing the values of a column

The column contents are cleared *(see Figure 260, page 209)*.

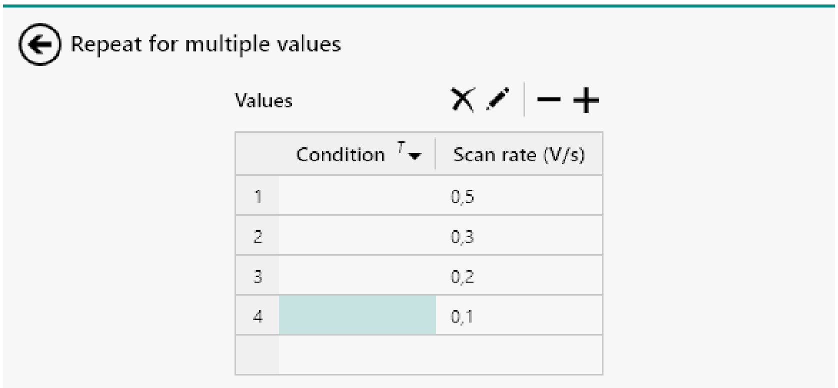


Figure 260 The column contents are cleared

7.1.3.2.9 Remove columns from the table

It is possible to remove a column from the table by selecting the column and clicking the ━ button located above the table *(see Figure 261, page 209)*.

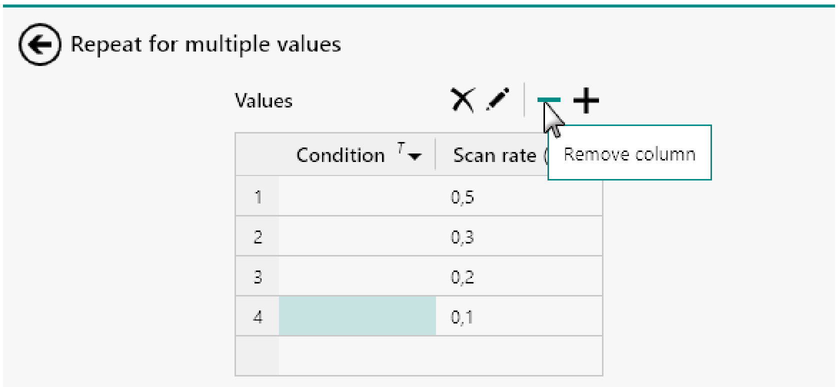


Figure 261 Removing a column from the table

The selected column is removed *(see Figure 262, page 209)*.

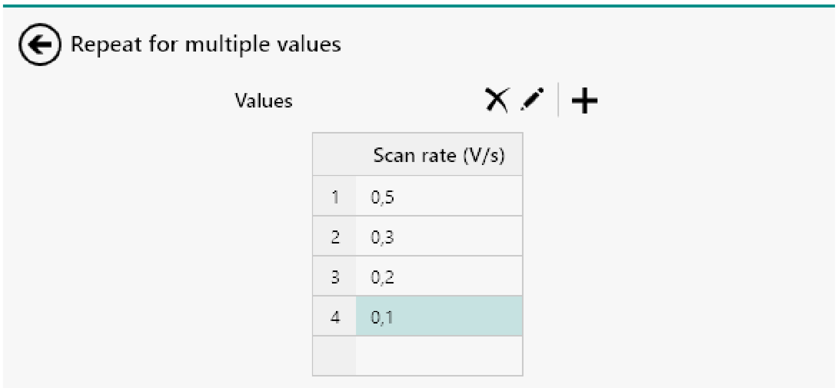


Figure 262 The selected column is removed

7.1.3.3 Timed repeat

The following properties are available when the command is used in the *Timed* mode *(see Figure 263, page 210)*:

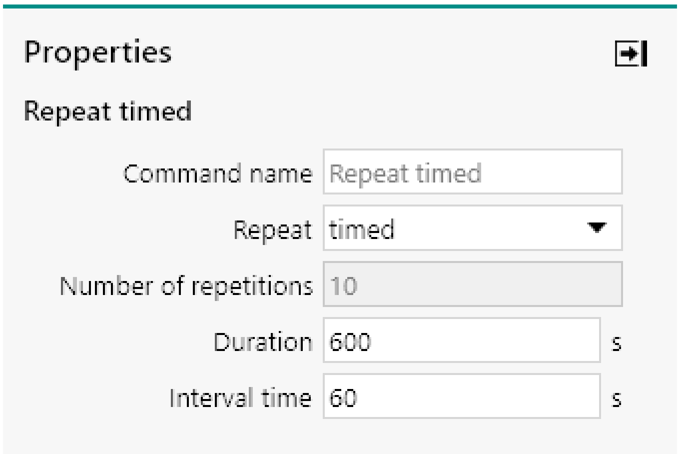


Figure 263 Timed repeat mode properties

- **Command name:** a user-defined name for the command.
- **Number of repetitions:** provides the calculated number of repetitions, provided as a read-only value. This value is determined based on the *Duration* and *Interval time* properties.
- **Duration:** specifies the duration of the repeat loop, in s.
- **Interval time:** specifies the interval time between two consecutive repetitions, in s.

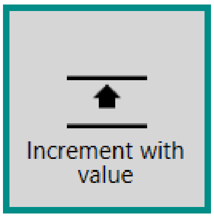
NOTE

The *Interval time* should be an integral fraction of the *Duration* in order to accurately determine the number of repetitions. A warning is provided when this condition is not met.

NOTE

The **Duration** property can be modified in real time.

7.1.4 Increment

	This command can be used to increment (or decrement) another parameter in the procedure.

The **Increment** command can be used in two different modes, which can be selected using the provided drop-down list *(see Figure 264, page 211)*:

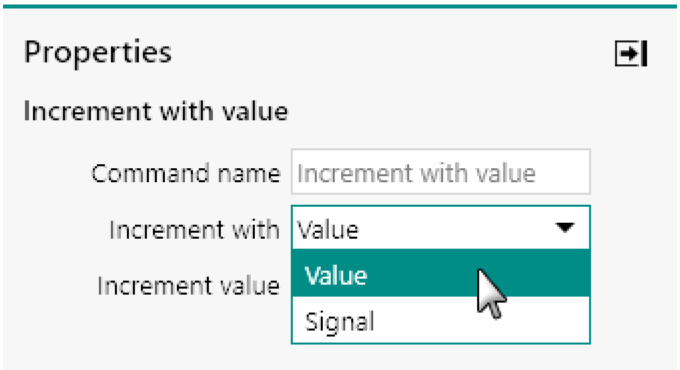


Figure 264 Two modes are provided by the Increment command

1. Increment with Value (default mode)
2. Increment with Signal

> **NOTE**
>
> The **Increment** command description in the procedure editor is dynamically adjusted in function of the specified mode.

7.1.4.1 Increment with Value

The following properties are available when the command is used in the *Increment with Value* mode *(see Figure 265, page 211)*:

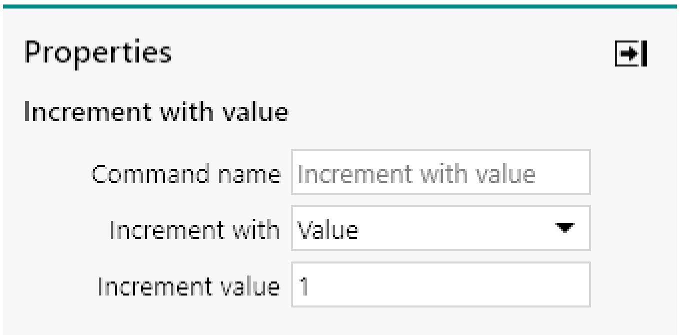


Figure 265 Increment with Value mode property

- **Command name:** a user-defined name for the command.
- **Increment value:** the value used to increment the target property (negative value for decrement).

NOTE

The target property of the **Increment** command is defined using a link *(see Chapter 10.13, page 657)*.

7.1.4.2 Increment with Signal

The following properties are available when the command is used in the *Increment with Signal* mode *(see Figure 266, page 212)*:

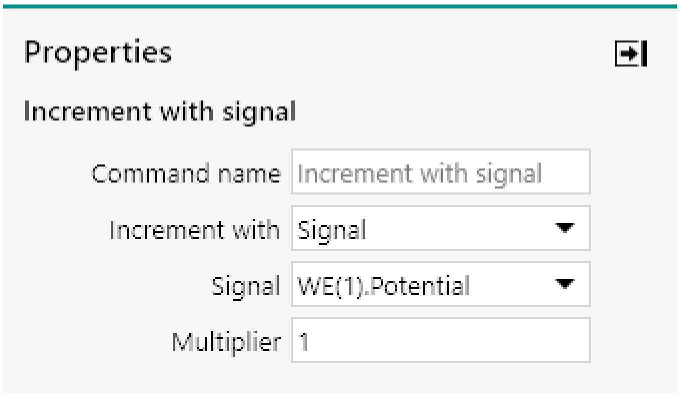


Figure 266 Increment with Signal mode properties

- **Command name:** a user-defined name for the command.
- **Signal:** the signal used to increment the target property (available from a drop-down list). All measurable signals are shown in this list.
- **Multiplier:** a multiplication factor that can be applied on the selected signal.

NOTE

The target property of the **Increment** command is defined using a link *(see Chapter 10.13, page 657)*.

NOTE

The list of signals provided in the Signal drop-down list depends on the hardware setup of the instrument *(see Figure 267, page 213)*.

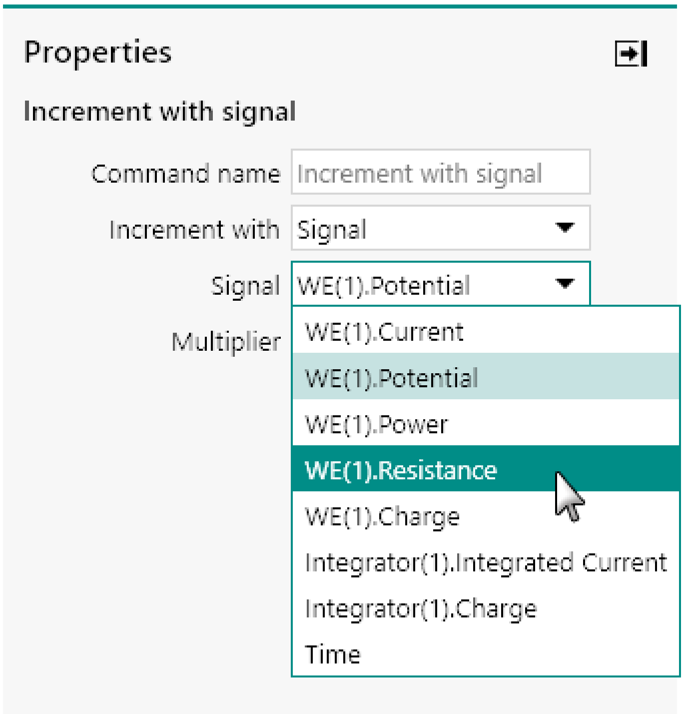


Figure 267 The signal used in the Increment with Signal mode

7.1.5 Play sound

	This command can be used to play a sound using a system or user defined source.

The details of the command properties of the **Play sound** command are shown in *Figure 268*:

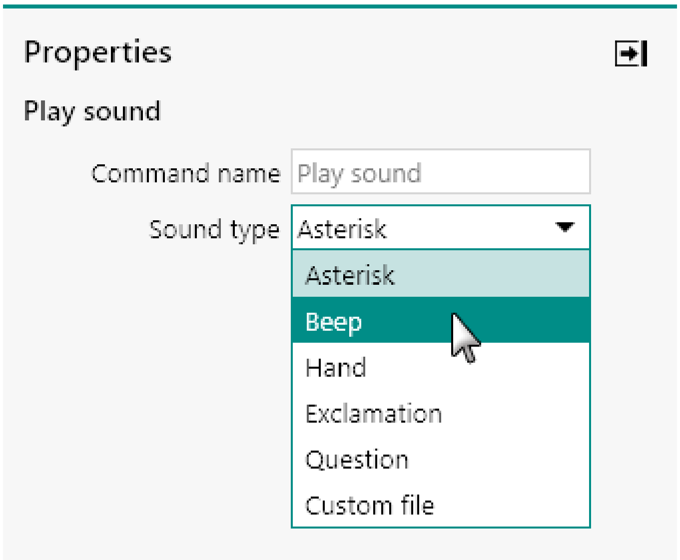
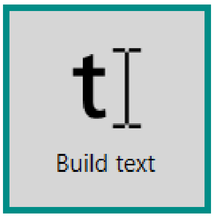


Figure 268 The properties of the Play sound command

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Sound type:** a drop-down list allows selection between System sounds or a custom file.
- **Filename:** the path to the sound file, only shown when a Custom file is selected using the provided drop-down list. A Browse button is provided to locate the file.

7.1.6 Build text

	This command can be used to build a custom string, which can be linked to another property in the procedure.

The details of the command properties of the **Build text** command are shown in *Figure 269*:

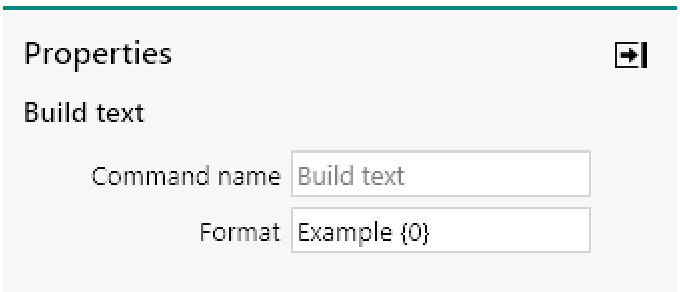


Figure 269 The property of the Build text command

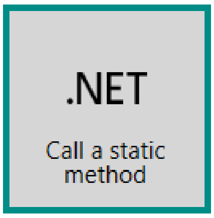
The following properties are available:

- **Command name:** a user-defined name for the command.
- **Format:** a string containing one or more *format items* (**{x}**, where **x** is an integer value, starting at 0). Each specified format item is linkable to another command property.

NOTE

The numbers used to specify the format items in the **Build text** command must be unique and must be sequential, starting at 0.

7.1.7 .NET



This command can be used to call functionality provided by a .NET API (Application Programming Interface).

CAUTION

This command is intended for advanced users. The use of this command falls outside of the scope of this manual.

The **.NET** command can be used in six different modes, which can be selected using the provided drop-down list *(see Figure 270, page 215)*:

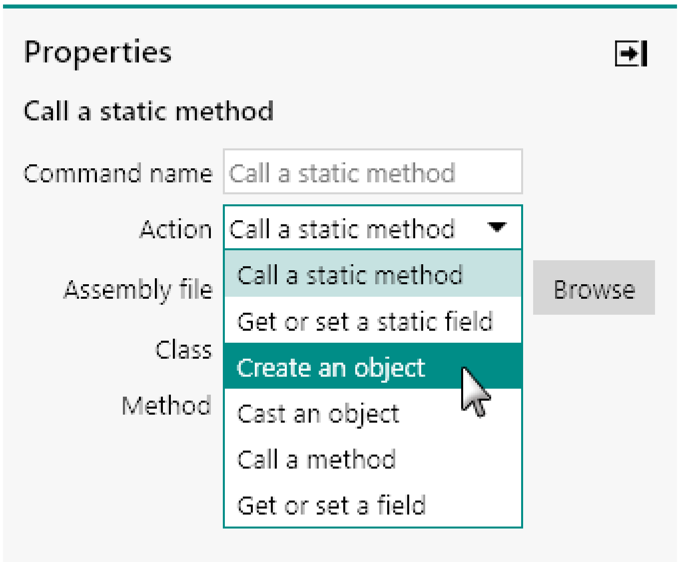


Figure 270 Six modes are provided by the .NET command

1. Call a static method (default mode)

2. Get or set a static field
3. Create an object
4. Cast an object
5. Call a method
6. Get or set a field

NOTE

The **.NET** command description in the procedure editor is dynamically adjusted in function of the specified mode.

7.1.7.1 Call a static method

The following properties are available when the command is used in the *Call a static method* mode *(see Figure 271, page 216)*:

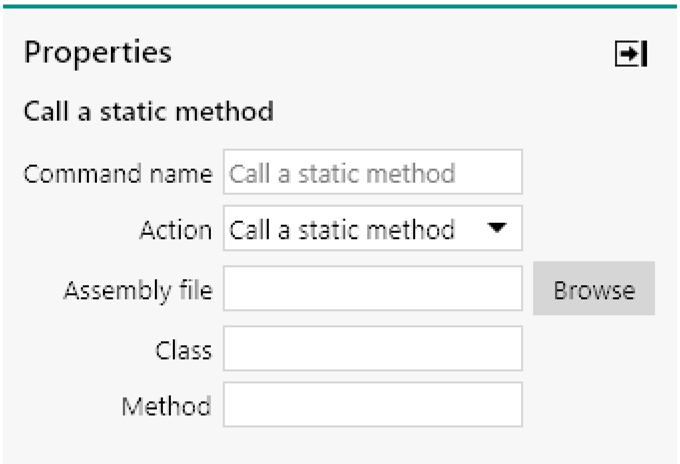


Figure 271 Call a static method mode properties

- **Command name:** a user-defined name for the command.
- **Assembly file:** specifies the path to the assembly file containing the functionality to call. A Browse button is provided to locate the file.
- **Class:** the class provided in the assembly file.
- **Method:** the method provided by the selected class.

NOTE

It is also possible to use the *Call a Static method* mode to access native .NET functionality. In that case, no Assembly file needs to be specified. Instead the .NET class that is required can be directly specified in the Class input field.

NOTE

The Class and Method fields are populated as soon as a valid Assembly file is specified.

7.1.7.2 Get or set a static field

The following properties are available when the command is used in the *Get or set a static field* mode *(see Figure 272, page 217)*:

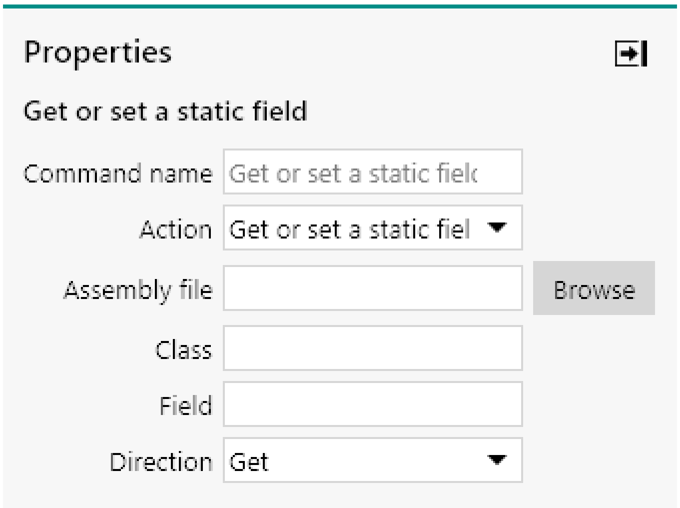


Figure 272 Get or set a static field mode properties

- **Command name:** a user-defined name for the command.
- **Assembly file:** specifies the path to the assembly file containing the functionality to call. A Browse button is provided to locate the file.
- **Class:** the class provided in the assembly file.
- **Field:** the field to get or set.
- **Direction (Get/Set):** a drop-down list that can be set to Get or Set.

7.1.7.3 Create object

The following properties are available when the command is used in the *Create an object* mode *(see Figure 273, page 218)*:

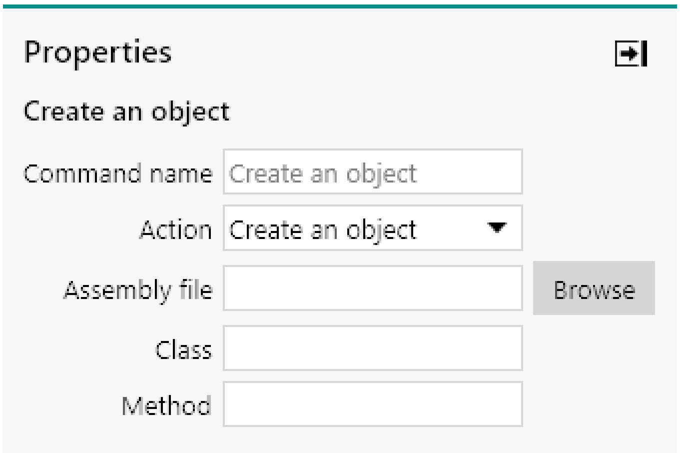


Figure 273 Create an object mode properties

- **Command name:** a user-defined name for the command.
- **Assembly file:** specifies the path to the assembly file containing the functionality to call. A Browse button is provided to locate the file.
- **Class:** the class provided in the assembly file.
- **Method:** the method provided by the selected class.

7.1.7.4 Cast object

The following properties are available when the command is used in the *Cast object* mode *(see Figure 274, page 218)*:

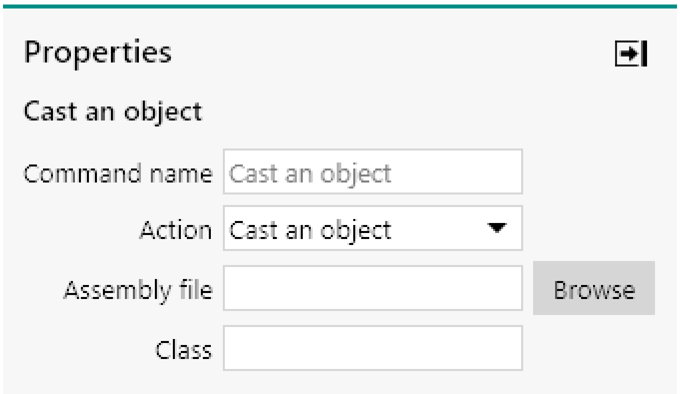


Figure 274 Cast an object mode properties

- **Command name:** a user-defined name for the command.
- **Assembly file:** specifies the path to the assembly file containing the functionality to call. A Browse button is provided to locate the file.
- **Class:** the class provided in the assembly file.

7.1.7.5 Call a method

The following properties are available when the command is used in the *Call a method* mode *(see Figure 275, page 219)*:

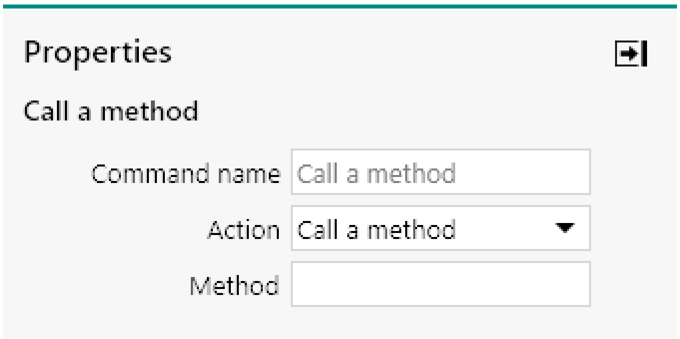


Figure 275 Call a method mode properties

- **Command name:** a user-defined name for the command.
- **Method:** the method to call.

7.1.7.6 Get or set a field

The following properties are available when the command is used in the *Get or set a field* mode *(see Figure 276, page 219)*:

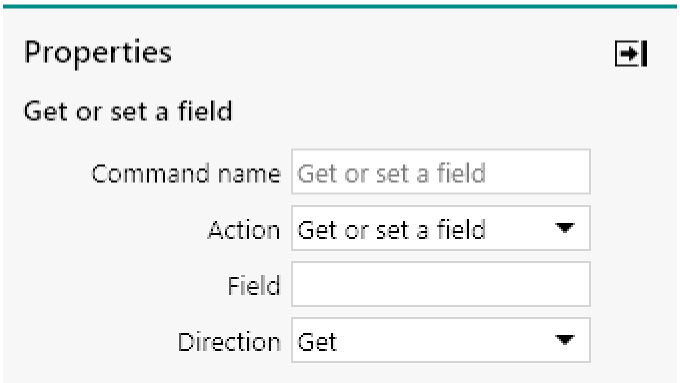


Figure 276 Get or set a field mode properties

- **Command name:** a user-defined name for the command.
- **Field:** the field to get or set.
- **Direction (Get/Set):** a drop-down list that can be set to Get or Set.

7.2 Measurement - general

Commands located in the **Measurement – general** group can be used to control the settings of the instrument during a procedure without performing an actual measurement.

The available commands are represented by a shortcut icon *(see Figure 277, page 220)*.

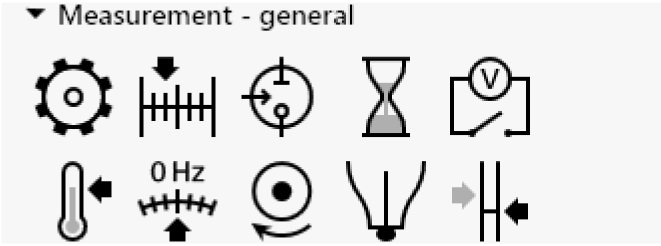
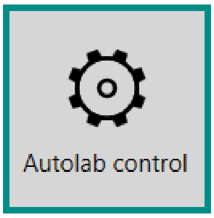


Figure 277 The Measurement - general commands

The following commands are available:

- **Autolab control:** a command which can be used to define the instrumental settings of the Autolab potentiostat/galvanostat and its optional modules *(see Chapter 7.2.1, page 221)*
- **Apply:** a command which can be used to define the applied potential or current *(see Chapter 7.2.2, page 224)*
- **Cell:** a command which can be used switch the cell on or off *(see Chapter 7.2.3, page 225)*
- **Wait:** a command which can be used to force the procedure to wait *(see Chapter 7.2.4, page 225)*
- **OCP measurement:** a command which can be used to measure the open circuit potential *(see Chapter 7.2.5, page 231)*
- **Set pH measurement temperature:** a command which can be used to set the pH measurement temperature *(see Chapter 7.2.6, page 233)*
- **Reset EQCM delta frequency:** a command which can be used to reset the ΔFrequency signal measured by the EQCM module *(see Chapter 7.2.7, page 234)*
- **Control Autolab R(R)DE:** a command used to control the rotation rate of the Autolab rotating disk electrode (RDE) or rotating ring disk electrode (RRDE)
- **MDE control:** a command used to control a mercury drop electrode stand connected to the Autolab using the IME663 or the IME303 module *(see Chapter 7.2.9, page 237)*
- **Multi Autolab synchronization:** a command which can be used to create a synchronization point in a procedure for multi Autolab measurements *(see Chapter 7.2.10, page 240)*

7.2.1 Autolab control

	This command can be used specify the hardware configuration of the instrument during a measurement. All the instrumental settings are configured using a dedicated dialog. The available settings depend on the hardware configuration.

The details of the command properties of the **Autolab control** command are shown in *Figure 278*.

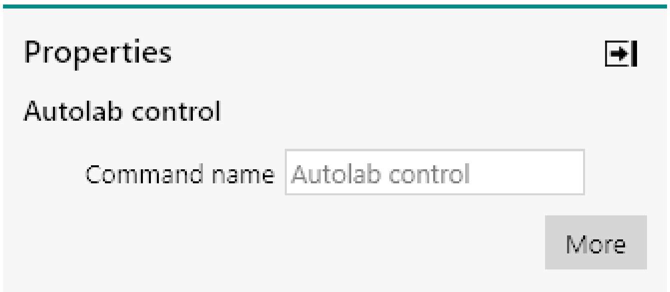


Figure 278 The properties of the Autolab control command

The following properties are available:

- **Command name:** a user-defined name for the command.

The   button can be used to edit the instrument settings for the **Autolab control** command. The Autolab control screen will be displayed *(see Figure 279, page 221)*.

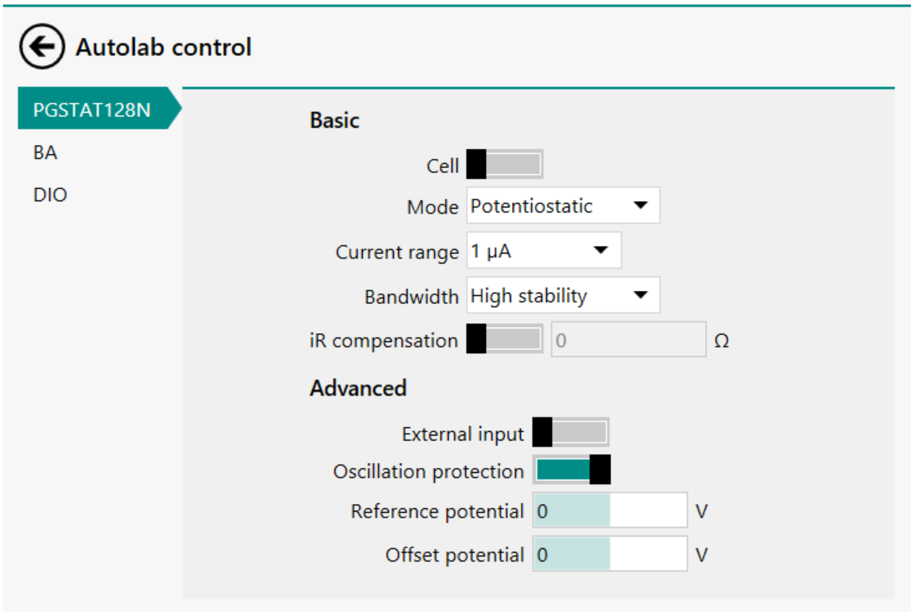


Figure 279 The Autolab control editor

NOTE

The settings provided in the **Autolab control** command dialog depend on the hardware setup. More details about the available settings are provided in the hardware description chapters located at the end of this document *(see Chapter 16, page 852)*.

Whenever a setting, available in the Autolab control screen, is adjusted, this setting will be made available in the **Properties** panel *(see Figure 280, page 222)*.

Figure 280 Modified settings are automatically made available in the Properties panel

These settings can be directly edited in the **Properties** panel, without the need of opening the Autolab control screen. A ↺ button will be added to any modified setting in order to undo this modification and remove this setting from the **Properties** panel *(see Figure 281, page 223)*.

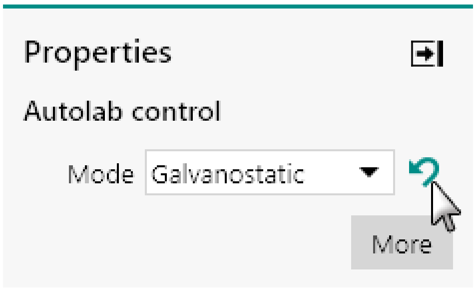


Figure 281 Modified settings can be directly edited or removed from the Properties panel

It is possible to add additional settings from the Autolab control screen to the **Properties** panel without modifying them, by clicking the button *(see Figure 282, page 223)*.

Figure 282 Additional settings can be added to the Properties panel

These additional settings will become visible in the **Properties** panel *(see Figure 283, page 224)*.

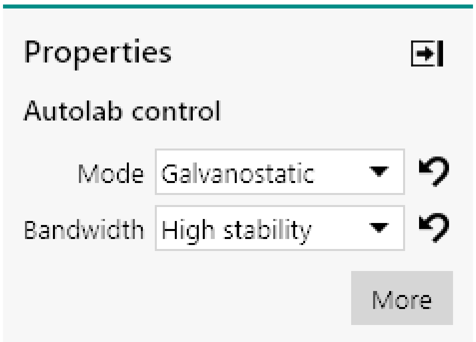
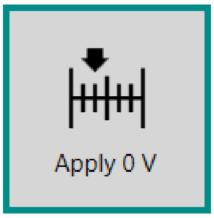


Figure 283 Additional settings will be visible in the Properties panel

7.2.2 Apply

	This command can be used to apply a fixed DC setpoint on the electrochemical cell. The applied value will be in volt (V) or ampere (A) depending on the settings of the instrument.

The details of the properties of the **Apply** command are shown in *Figure 284*:

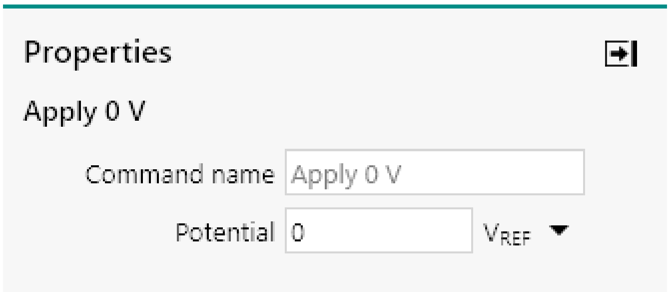


Figure 284 The properties of the Apply command

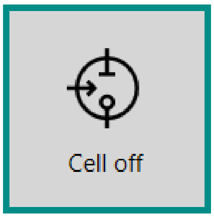
The following properties are available:

- **Command name:** a user-defined name for the command.
- **Potential/Current:** the applied potential or current value, in V or A respectively.
- **With respect to drop-down list:** a drop-down list that provides the choice of the reference used to apply a potential value (only shown in potentiostatic mode). The potential can be specified with respect to the reference electrode potential (V_{REF}) or the open-circuit potential (V_{OCP}).

NOTE

The **Apply** command description in the procedure editor is dynamically adjusted in function of the specified value.

7.2.3 Cell

	This command can be used to switch the cell off or on.

The details of the command properties of the **Cell** command are shown in *Figure 285*:

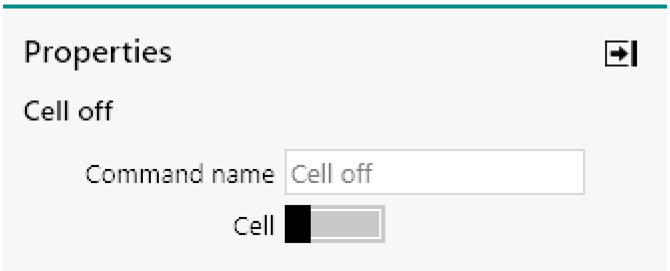


Figure 285 The properties of the Cell command

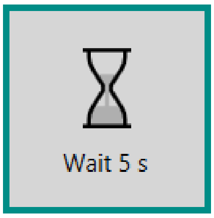
The following properties are available:

- **Command name:** a user-defined name for the command.
- **Switch cell:** a  toggle control provided to switch the cell off or on.

NOTE

The **Cell** command description in the procedure editor is dynamically adjusted in function of the toggle.

7.2.4 Wait

	This command can be used to force the procedure to wait for a predefined amount of time (in seconds) or until a certain trigger signal is recorded a DIO port of the instrument or the input lines of a Metrohm 6.2148.010 Remote Box.

The **Wait** command can be used in four different modes, which can be selected using the provided drop-down list *(see Figure 286, page 226)*:

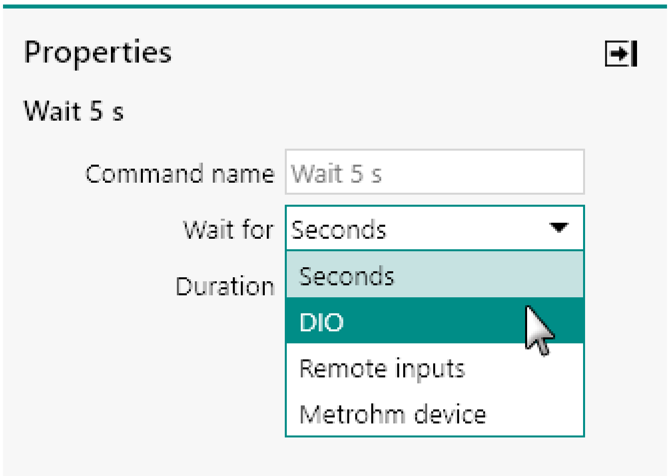


Figure 286 Four modes are provided by the Wait command

1. Wait for Seconds (default mode)
2. Wait for DIO
3. Wait for Remote inputs
4. Wait for Metrohm device

NOTE

The **Wait** command description in the procedure editor is dynamically adjusted in function of the specified mode.

7.2.4.1 Wait for Seconds

The following properties are available when the **Wait** command is used in the *Wait for Seconds* mode *(see Figure 287, page 227)*:

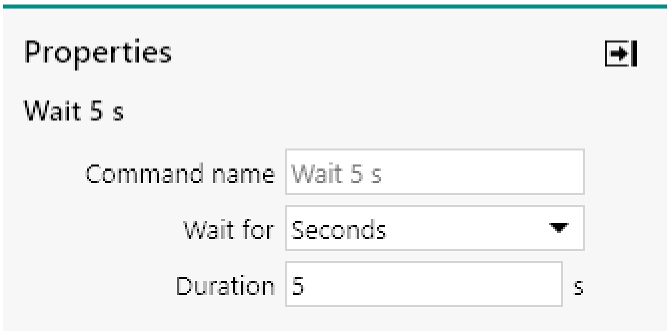


Figure 287 The properties of the Wait for Seconds mode

- **Command name:** a user-defined name for the command.
- **Duration:** specifies the duration of the wait time, in s.

NOTE

The **Duration** property can be modified in real time.

7.2.4.2 Wait for DIO

The following properties are available when the **Wait** command is used in the *Wait for DIO* mode *(see Figure 288, page 227)*:

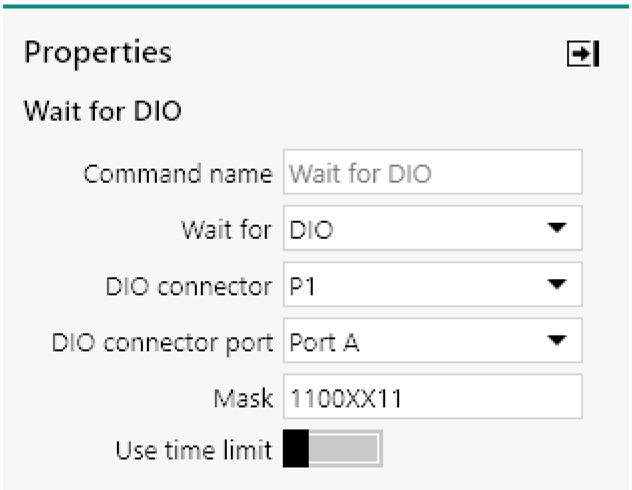


Figure 288 The properties of the Wait for DIO mode

- **Command name:** a user-defined name for the command.
- **DIO connector:** the connector used to receive the trigger (P1 or P2, available from a drop-down list).
- **DIO connector port:** the port of the DIO connector used to receive the trigger (A, B or C, available from a drop-down list).

- **Mask:** the trigger mask, specified as 8 or 4 bits. The expected bit sequence must be formatted using 1, 0 and X (1 indicates that the pin status must be 'high', 0 indicates that the pin status must be 'low', X indicates that the pin status may be both). In *Figure 288*, the command will force the procedure to wait until pins 8, 7, 2 and 1 are 'high', pins 6 and 5 are 'low'. The status of pins 4 and 3 is irrelevant (X status).
- **Use time limit:** a toggle is provided to enable or disable the time limit. When this option is enabled, the *Wait for DIO* command will stop waiting after the specified amount of time.
- **Time limit:** the time limit after which the command stops waiting (if the *Use time limit* toggle is on), in s.

7.2.4.3 Wait for Remote inputs

CAUTION

This mode requires a **Metrohm 6.2148.010 Remote Box** to be connected to the computer.

The following properties are available when the **Wait** command is used in the *Wait for Remote inputs* mode *(see Figure 289, page 228)*:

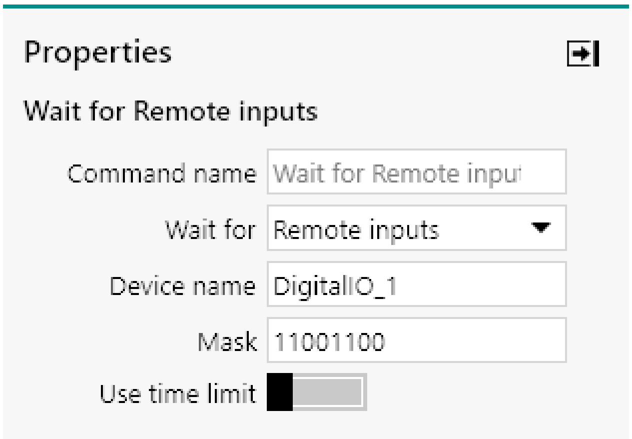


Figure 289 The properties of the Wait for Remote inputs mode

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying name of the Remote Box.
- **Mask:** the trigger mask, specified as byte. The expected byte must be formatted using 1 and 0 (1 indicates that the pin status must be 'high' and 0 indicates that the pin status must be 'low'). In *Figure 289*, the command will force the procedure to wait until IN7, IN6, IN3 and IN2 are set to 'high' state and IN5, IN4, IN1 and IN0 are set to 'low' state.
- **Use time limit:** a toggle is provided to enable or disable the time limit. When this option is enabled, the *Wait for Remote inputs* command will stop waiting after the specified amount of time.

- **Time limit:** the time limit after which the command stops waiting (if the *Use time limit* toggle is on), in s.

7.2.4.4 Wait for Metrohm device

The following properties are available when the **Wait** command is used in the *Wait for Metrohm device* mode *(see Figure 290, page 229)*:

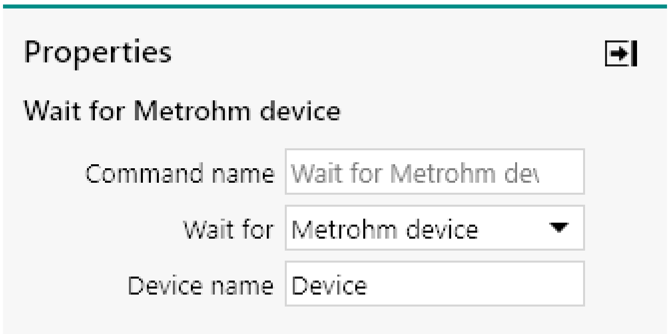


Figure 290 The properties of the Wait for Metrohm device mode

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying name of the Metrohm device.

This command can be used to force the procedure to wait until the specified **Metrohm** device returns to idle state. This is only relevant for devices for which the *parallel execution* setting is been set to on in the hardware setup *(see Chapter 5.5.1.1, page 140)*.

When *parallel execution* is enabled, the **Metrohm** device will not block the procedure while it is executing an action, allowing the next command to run and the procedure to continue. If this setting is disabled, the device will hold the procedure until the action being carried out by the device is finished.

Figure 291 illustrates the use of the parallel execution, schematically.

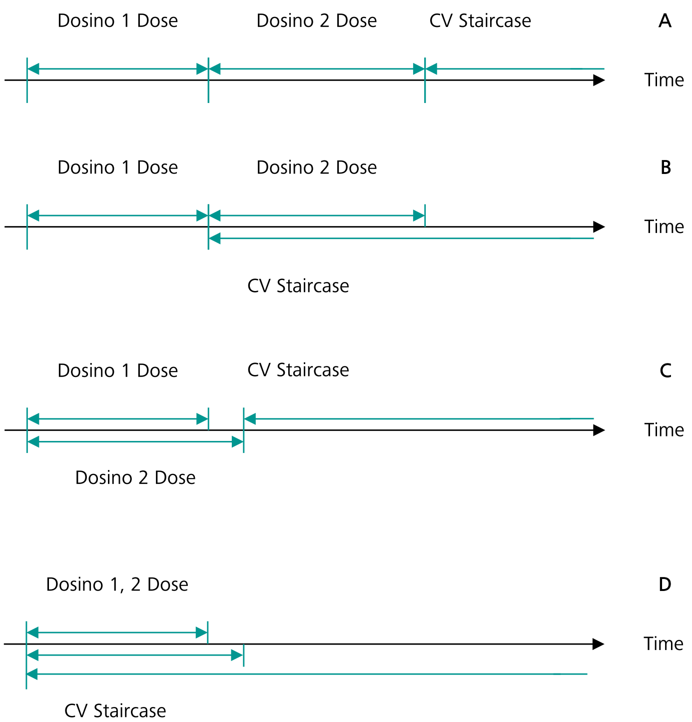


Figure 291 *Illustration of the parallel execution option for the Metrohm devices*

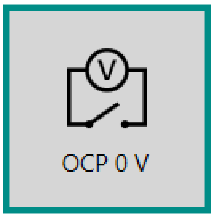
- In *Figure 291*, A: Dosino 1 and Dosino 2 have parallel execution disabled. Both Dosino need to finish the Dose command before the CV staircase command can start.
- In *Figure 291*, B: parallel execution is enabled on Dosino 2 and disabled on Dosino 1. Dosino 2 starts dosing immediately after Dosino 1 is finished. The CV staircase command starts as soon as Dosino 2 starts dosing.
- In *Figure 291*, C: parallel execution is enabled on Dosino 1 and disabled on Dosino 2. Dosino 2 starts dosing at the same time as Dosino 1. Only when Dosino 2 is finished can the CV staircase command start.
- In *Figure 291*, D: parallel execution is enabled for both Dosino 1 and Dosino 2. All three commands start at the same time.

The *Wait for Metrohm device* mode can be used in a procedure to force the procedure to wait until the specified device finishes the command it is executing. This mode can thus be used to overrule the parallel execution of the device.

NOTE

The *Wait for Metrohm device* mode has no effect on devices for which parallel execution is disabled.

7.2.5 OCP

	This command can be used to measure the open circuit potential (OCP). This command will switch the cell off before starting the measurement.

The details of the command properties of the **OCP** command are shown in *Figure 292*:

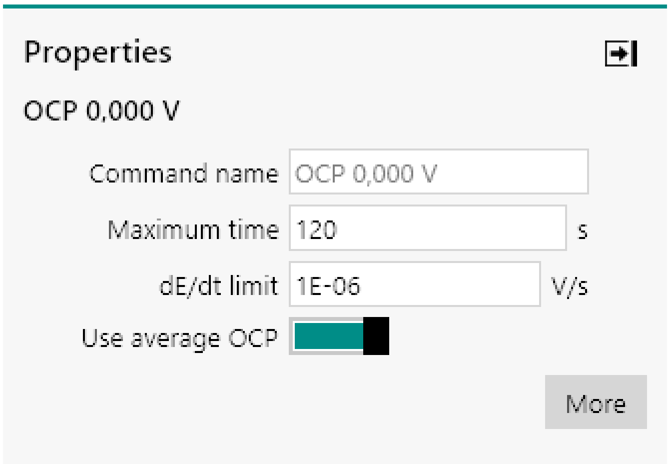


Figure 292 The properties of the OCP command

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Maximum time:** the maximum duration for the OCP measurement, in s.
- **dE/dt limit:** the time derivative limit, in V/s. When this value is not 0, the recording of the OCP will stop when the time derivative of the potential is smaller or equal to the specified limit.
- **Use average OCP:** a toggle control provided to specify if the averaged value of the OCP should be stored or the final value of the OCP. When the average value is used, the OCP is determined using a 5 seconds moving average.

NOTE

The **OCP** command provides access to additional options, through the More button *(see Chapter 9, page 594)*.

When the **OCP** command is executed, a dedicated window will be shown, providing additional controls during the measurement *(see Figure 293, page 232)*.

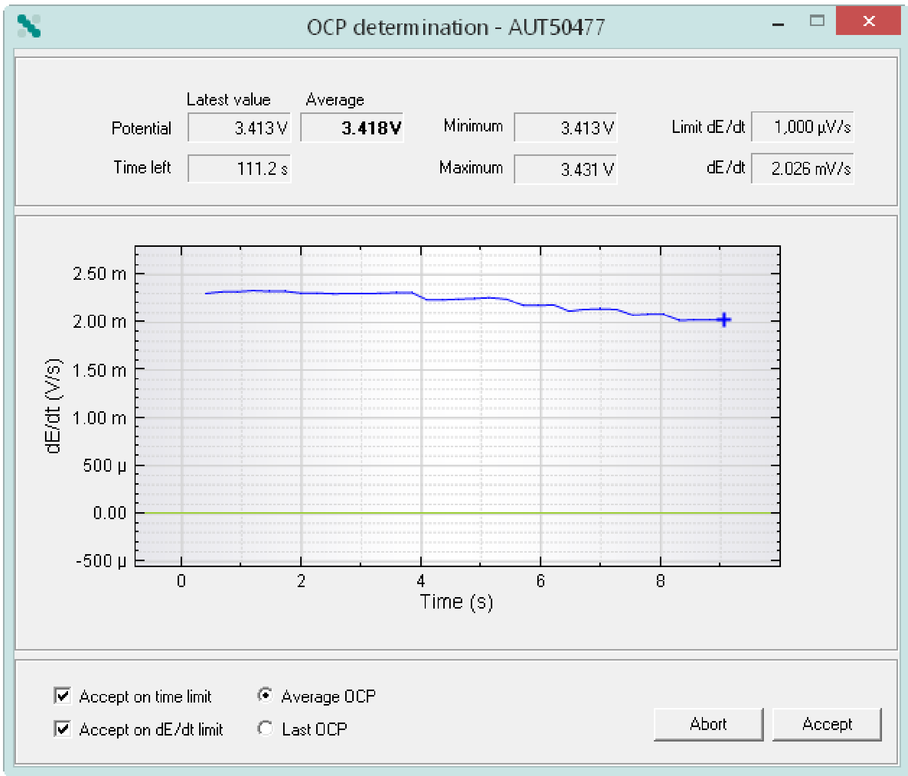


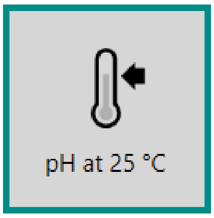
Figure 293 The OCP determination window displayed when the OCP command is executed

The following information is shown in the window:

- **Top section:**
  - **Potential:** the latest and average value of the measured open circuit potential are displayed, in V.
  - **Time left:** displays the remaining measurement time, in s.
  - **Minimum and Maximum:** the minimum and maximum value of the open circuit potential that have been measured, in V.
  - **Limit dE/dt and dE/dt:** the target open circuit potential time derivative value and the actual time derivative value of the open circuit potential, in V/s.
- **Middle section:** this section displays a real time plot of the time derivative of the open circuit potential. The blue line corresponds to the measured dE/dt and the green line corresponds to the limit dE/dt value.

- **Bottom section:**
  - **Accept on time limit:** this check box specifies if the OCP command should stop when the Time left reaches 0 s. If the checkbox is not checked, the measurement will not stop when the time runs out.
  - **Accept on dE/dt limit:** this check box specifies if the OCP command should stop when the measured dE/dt value becomes equal or lower than the limit dE/dt value. When this check box is not checked, the measurement will not stop when the measured dE/dt becomes smaller or equal to the limit dE/dt.
  - **Average OCP/Last OCP:** this radio button specifies if the averaged OCP or the last measured OCP value should be returned by the command when the measurement is finished.
  - **Abort button:** this button can be used to force the procedure to stop.
  - **Accept button:** this button can be used to force the OCP command to stop measuring the open circuit potential and to proceed with the rest of the procedure.

7.2.6 Set pH measurement temperature

	This command can be used to specify the measurement temperature, for automatic pH correction (if the temperature is not measured through the T input of the **pX1000** module or if the pH is measured using a **pX** module).

CAUTION

This command requires a **pX1000** or **pX** module *(see Chapter 16.3.2.18, page 1141)* installed in the Autolab.

This command performs the following mathematical adjustment:

$$pH = 7 - \frac{a}{b} + \frac{E}{b} \cdot \frac{T_{cal}}{T}$$

Where a and b are the intercept and the slope of the calibration curve, E is the measured potential, T is the specified temperature and T_{cal} is the calibration temperature.

The details of the properties of the **Set pH measurement temperature** command is shown in *Figure 294*:

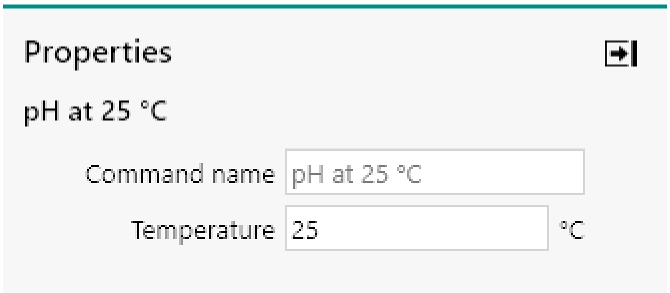


Figure 294 The properties of the Set pH measurement temperature command

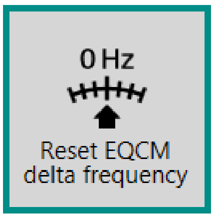
The following properties are available:

- **Command name:** a user-defined name for the command.
- **Temperature:** the measurement temperature, in °C.

NOTE

The **Set pH temperature measurement** command description in the procedure editor is dynamically adjusted in function of the specified value.

7.2.7 Reset EQCM delta frequency

	This command can be used to create a breakpoint in the procedure during which the signals from the **EQCM** module can be adjusted and zeroed, if necessary.

CAUTION

This command requires an **EQCM** module *(see Chapter 16.3.2.10, page 1054)* installed in the Autolab.

The details of the command properties of the **Reset EQCM delta frequency** command are shown in *Figure 295*.

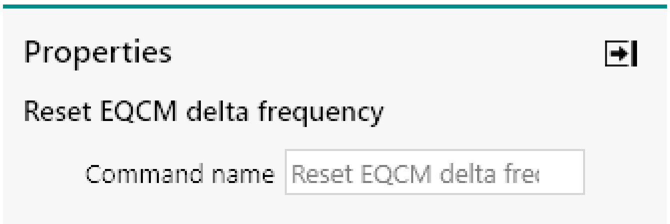


Figure 295 The properties of the Autolab control command

The following properties are available:

- **Command name:** a user-defined name for the command.

When the **Reset EQCM delta frequency** command is executed, a dedicated window will be shown, providing additional control during the measurement *(see Figure 296, page 235)*.

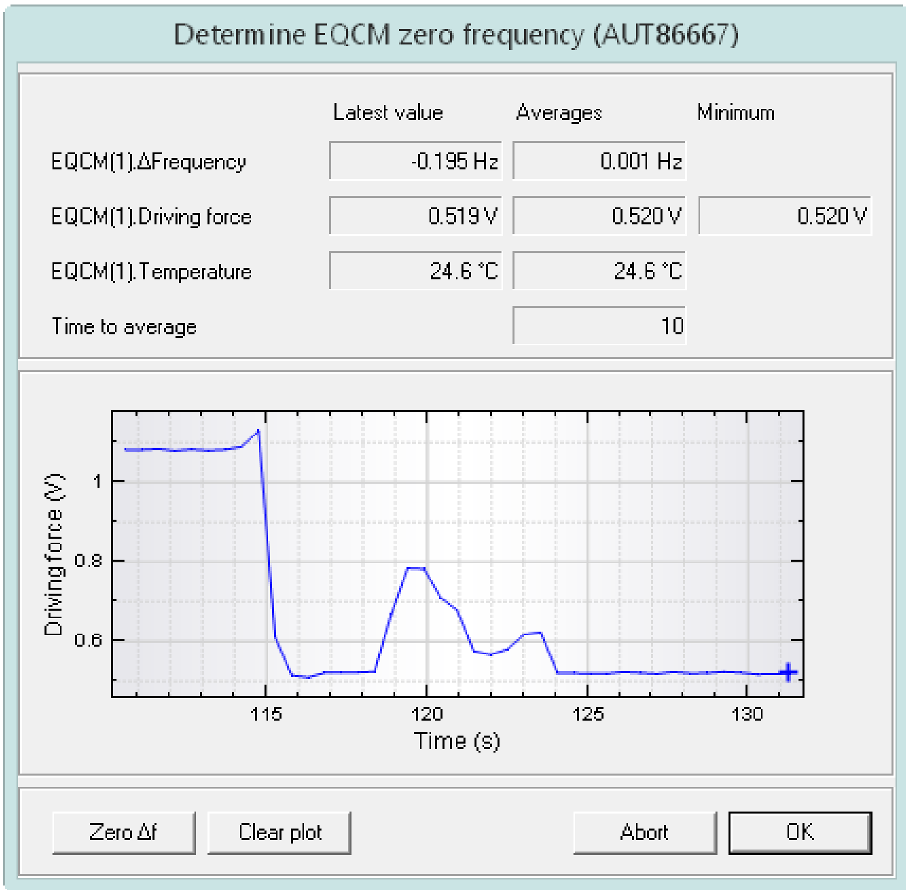
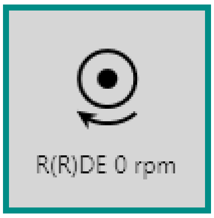


Figure 296 The Determine EQCM zero frequency window displayed when the Reset EQCM delta frequency command is executed

The following information is shown in the window:

- **Top section:**
  – **EQCM(1).ΔFrequency:** displays the latest and average values of the measured EQCM(1).Frequency signal are displayed, in V.
  – **EQCM(1).Driving force:** displays the latest, average and minimum values of the measured EQCM(1).Driving force signal are displayed, in V.
  – **EQCM(1).Temperature:** displays the latest and average values of the measured EQCM(1).Temperature signal are displayed, in °C.
  – **Time to average:** a read-only field that indicates the duration, in seconds, used to determine the average values of the EQCM signals.
- **Middle section:** this section displays a real time plot of the EQCM(1).Driving force signal. Using the provided trimmer, it is possible to adjust the driving force as indicated in the EQCM User Manual.
- **Bottom section:**
  – **Zero Δf button:** this button can be used to reset the measured value of the EQCM(1).ΔFrequency signal. When this button is pressed, the value of the signal is recorded and then subtracted from the actual value, thus zeroing the signal. While the EQCM.(1).ΔFrequency signal is set to zero, the button is disabled.
  – **Clear plot button:** this button clears the plot of the EQCM(1).Driving force signal.
  – **Abort button:** this button can be used to force the procedure to stop.
  – **Accept button:** this button can be used to close the Determine EQCM zero frequency window. The procedure will continue using the last value of the EQCM(1).ΔFrequency signal.

7.2.8 Autolab R(R)DE control

	This command can be used to control the Autolab rotating disk electrode (RDE) or rotating ring disk electrode (RRDE), connected to the Autolab and operated in remote control mode.

The details of the command properties of the **Autolab R(R)DE control** command are shown in *Figure 297*:

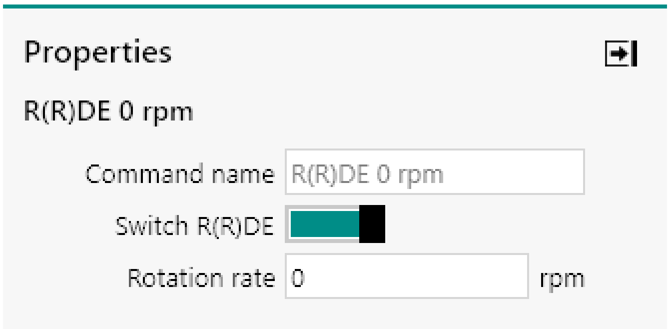


Figure 297 The properties of the Autolab R(R)DE control command

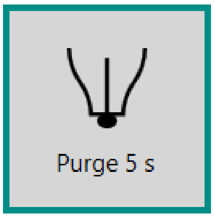
The following properties are available:

- **Command name:** a user-defined name for the command.
- **Switch R(R)DE:** a toggle control provided to switch the Autolab RDE or RRDE off or on.
- **Rotation rate:** specifies the rotation rate of the Autolab RDE or RRDE, in RPM.

NOTE

The **Autolab R(R)DE control** command description in the procedure editor is dynamically adjusted in function of the toggle.

7.2.9 MDE control

	This command can be used to control the Mercury Drop Electrode (MDE) using the **IME663** or the **IME303** interface and the **Metrohm 663 VA Stand**, **the Princeton Applied Research PAR303(A) Stand** or a compatible mercury drop electrode stand.

CAUTION

This command requires a **IME663** *(see Chapter 16.3.2.15, page 1109)* or **IME303** *(see Chapter 16.3.2.14, page 1103)* connected to the Autolab. When this command is used without a **IME663** or **IME303**, an **error** will be displayed for the command.

The **MDE control** command can be used in three different modes, which can be selected using the provided drop-down list *(see Figure 298, page 238)*:

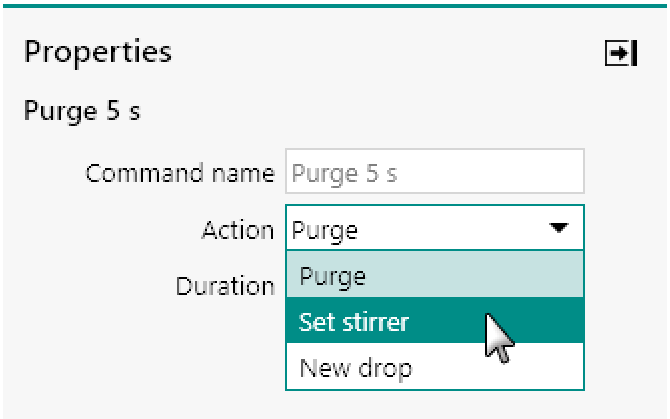


Figure 298 Three modes are provided by the MDE control command

1. Purge (default mode)
2. Set Stirrer
3. New Drop

NOTE

The **MDE control** command description in the procedure editor is dynamically adjusted in function of the specified mode.

CAUTION

Take all necessary precautions when working with mercury. It is highly recommended to consult the Material Safety Data Sheet (MSDS) before operating the **Metrohm 663 VA Stand**, the **Princeton Applied Research PAR303(A) Stand** or any other compatible stand. It is also recommended to dispose of the mercury waste properly.

7.2.9.1 Purge

The following properties are available when the **MDE control** command is used in the *Purge* mode *(see Figure 299, page 239)*:

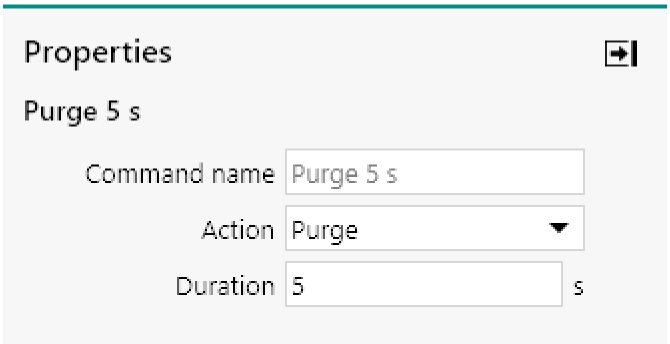


Figure 299 Purge mode properties

- **Command name:** a user-defined name for the command.
- **Duration:** specifies the duration during which the N_2 purging functionality provided by the MDE is active, in s.

7.2.9.2 Set Stirrer

The following properties are available when the **MDE control** command is used in the *Set Stirrer* mode *(see Figure 300, page 239)*:

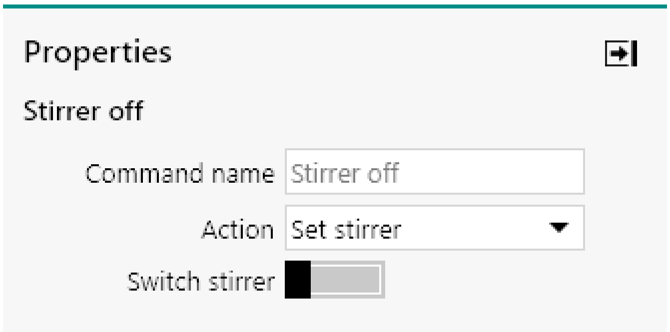


Figure 300 Set stirrer mode properties

- **Command name:** a user-defined name for the command.
- **Switch stirrer:** specifies the status of the built-in stirrer of the MDE through the provided  toggle. The specified status remains active until changed.

7.2.9.3 New drop

The following properties are available when the **MDE control** command is used in the *New drop* mode *(see Figure 301, page 240)*:

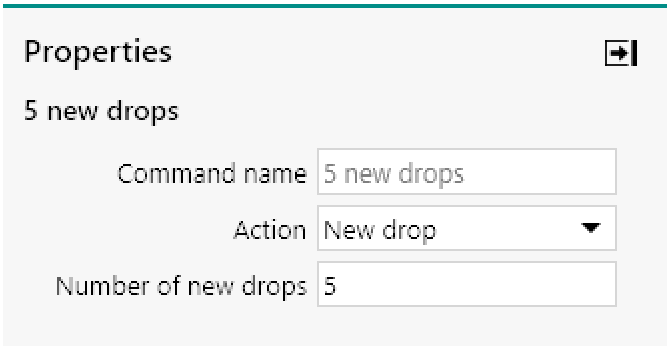
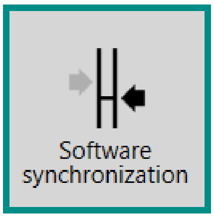


Figure 301 New drop mode properties

- **Command name:** a user-defined name for the command.
- **Number of new drops:** specifies the number of drops to knock off the capillary of the MDE by activating the built-in tapper.

> **NOTE**
>
> A 500 ms settling time is used each time the tapper is activated. This settling time can be adjusted in the hardware setup.

7.2.10 Synchronization

	This command can be used to create a synchronization point in the procedure.

The **Synchronization** command can be used in two different modes, which can be selected using the provided drop-down list *(see Figure 302, page 241)*:

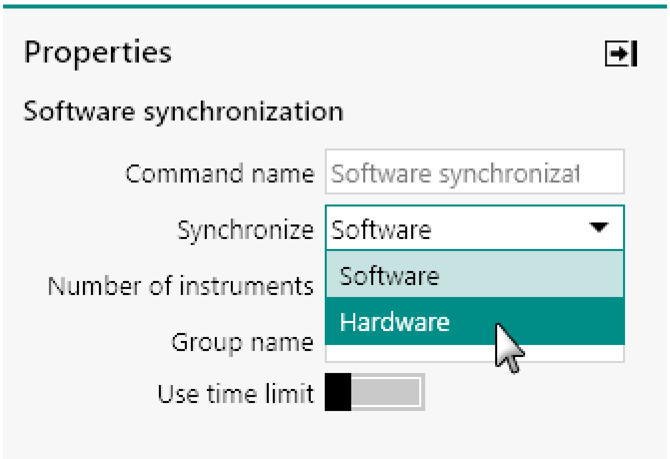


Figure 302 Two modes are provided by the Synchronization

Two modes are provided by the Synchronization command

1. Software synchronization (default mode)
2. Hardware synchronization

NOTE

The **Synchronization** command description in the procedure editor is dynamically adjusted in function of the specified mode.

7.2.10.1 Software synchronization

The following properties are available when the **Synchronization** command is used in the *Software synchronization* mode *(see Figure 303, page 241)*:

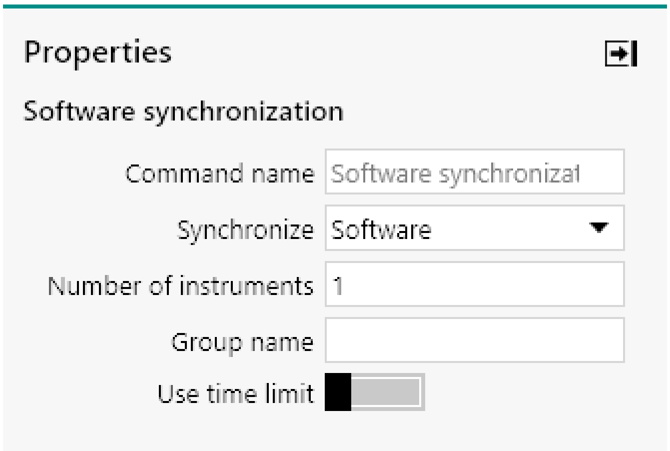


Figure 303 Software synchronization mode properties

- **Command name:** a user-defined name for the command.

- **Number of instruments:** the number Autolab instruments to synchronize. The synchronization is triggered as soon as this number is reached or when the optional time limit has been reached.
- **Group name:** defines a unique name for the synchronization command.
- **Use time limit:** a toggle that can be used to specify if a time limit should be used for the synchronization command.
- **Time limit:** specifies the time limit, in s.
- **Abort after time limit:** a toggle that can be used to specify if the measurement should be aborted if the time limit is reached.

NOTE

The normal behavior of the software synchronization command is to wait indefinitely until the number of specified instruments reach the synchronization command in their respective procedures. The time limit toggle can be used to overrule this waiting stage. When the waiting stage is overruled, the command can be further adjusted to allow the procedure to continue on each instrument or to abort the measurement.

7.2.10.2 Hardware synchronization

The following properties are available when the **Synchronization** command is used in the *Hardware synchronization* mode *(see Figure 304, page 242)*:

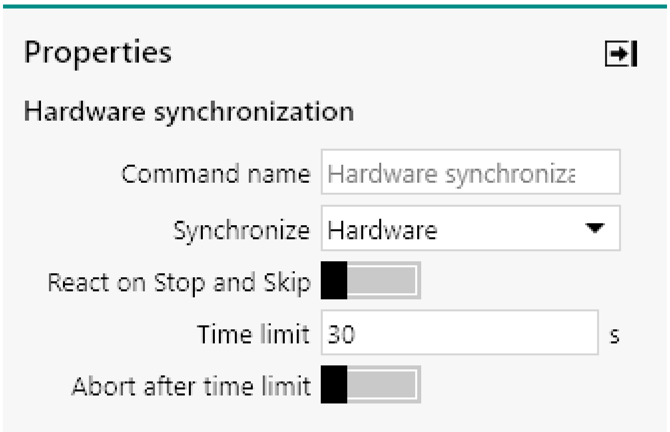


Figure 304 Hardware synchronization mode properties

- **Command name:** a user-defined name for the command.

- **React on Stop and Skip:** a toggle provided to allow the use of the Stop and the Skip option during measurements. When this option is disabled, the synchronization speed is as fast as possible but the Stop and Skip option cannot be used. When this option is used, this restriction no longer applies but the synchronization speed is slower.
- **Time limit:** specifies a time limit for the hardware synchronization, in s.
- **Abort after time limit:** a toggle that can be used to specify if the measurement should be aborted if the time limit is reached.

7.3 Measurement - cyclic and linear sweep voltammetry commands

Commands located in the **Measurement – cyclic and linear sweep voltammetry** group can be used to perform programmed potential or current sweep measurements.

The available commands are represented by a shortcut icon *(see Figure 305, page 243)*.

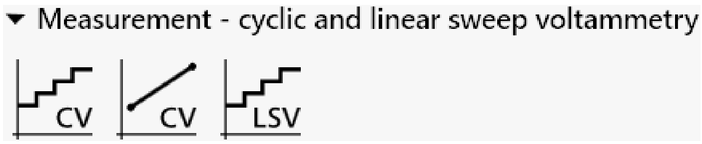
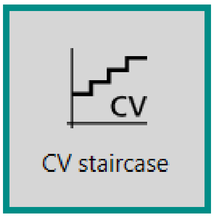


Figure 305 The Measurement - cyclic and linear sweep voltammetry commands

The following commands are available:

- **CV staircase:** a command which can be used to perform staircase cyclic voltammetry measurements *(see Chapter 7.3.1, page 243)*.
- **CV linear scan:** a command which can be used to perform linear scan cyclic voltammetry measurements *(see Chapter 7.3.2, page 246)*. This command requires the **SCAN250** or **SCANGEN** module *(see Chapter 16.3.2.19, page 1148)*.
- **LSV staircase:** a command which can be used to perform staircase linear sweep voltammetry measurements *(see Chapter 7.3.3, page 248)*.

7.3.1 CV staircase

	This command can be used to perform a staircase cyclic voltammetry measurement, in potentiostatic or galvanostatic conditions.

The details of the properties of the **CV staircase** command are shown in *Figure 306*:

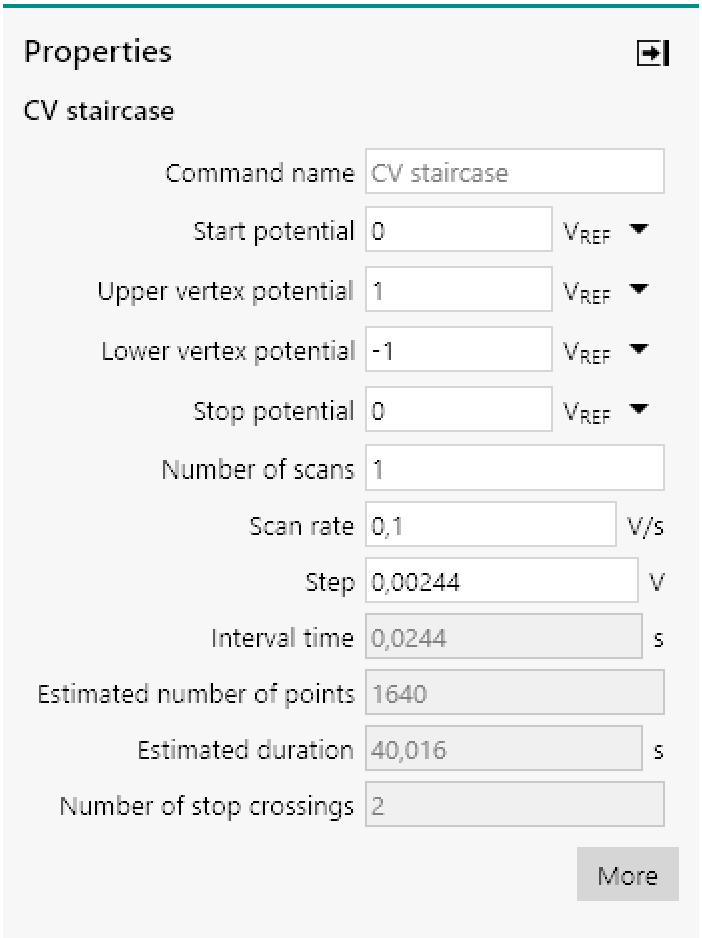


Figure 306 CV staircase properties

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Start potential/current:** the start potential or current value, in V or A respectively. The start value can be located outside of the scan range defined by the lower and upper vertices.
- **Upper vertex potential/current:** the upper vertex potential or current value, in V or A respectively. The upper vertex must be higher than the lower vertex.
- **Lower vertex potential/current:** the lower vertex potential or current value, in V or A respectively. The lower vertex must be lower than the upper vertex.
- **Stop potential/current:** the stop potential or current value, in V or A respectively. The stop value must be located within the scan range defined by the upper and lower vertices.
- **Number of scans:** the number of potential or current scans.

- **Step:** the potential or current step, in V or A respectively. The step can be positive or negative. With a positive step, the scan starts from the start potential or current towards the upper vertex potential or current. With a negative step, the scan direction is reversed.
- **Scan rate:** the scan rate of the potential or current sweep, in V/s or A/s.

Four additional properties are shown as *read-only*:

- **Interval time:** the time interval between two consecutive points in the scan. This property is defined by the potential or current step and the scan rate.
- **Estimated number of points:** the estimated number of points in the scan. This property is defined by the start and stop potential or current and the potential or current step.
- **Estimated duration:** this property shows the estimated duration, in s. This property is defined by the estimated number of points and the interval time, as well as the duration of underlying commands, if applicable.
- **Number of stop crossings:** the number of times the potential or current scan will cross the stop potential or current value. This property is defined by the number of scans.

CAUTION

in order to properly identify the scans in the data, it is important to make sure that the following conditions are respected when defining the parameters of the **CV staircase** command:

- The stop value must be smaller than the upper vertex minus the step value.
- The stop value must be larger than the lower vertex plus the step value.

NOTE

When this command is used in potentiostatic mode, a drop-down list provides the choice of the reference used to apply a potential value. The potential can be specified with respect to the reference electrode potential (V_{REF}) or the open-circuit potential (V_{OCP}).

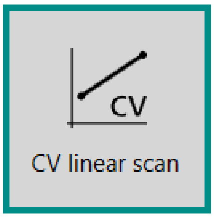
NOTE

The **CV staircase** command provides access to additional options, through the More button *(see Chapter 9, page 594)*.

NOTE

The **Upper vertex potential (or current)**, **Lower vertex potential (or current)**, **Stop potential (or current)**, **Number of scans** and **Scan rate** properties can be modified in real time.

7.3.2 CV linear scan

	This command can be used to perform a linear scan cyclic voltammetry measurement, in potentiostatic. This method can only be used with instrument fitted with the optional **SCAN250** or **SCANGEN** module.

CAUTION

This command requires a **SCAN250** or **SCANGEN** module *(see Chapter 16.3.2.19, page 1148)* installed in the Autolab. The high speed mode also needs an **ADC10M** or **ADC750** module *(see Chapter 16.3.2.1, page 977)* installed in the Autolab.

CAUTION

This command can only be used in Potentiostatic mode.

The details of the properties of the **CV linear scan** command are shown in *Figure 307*:

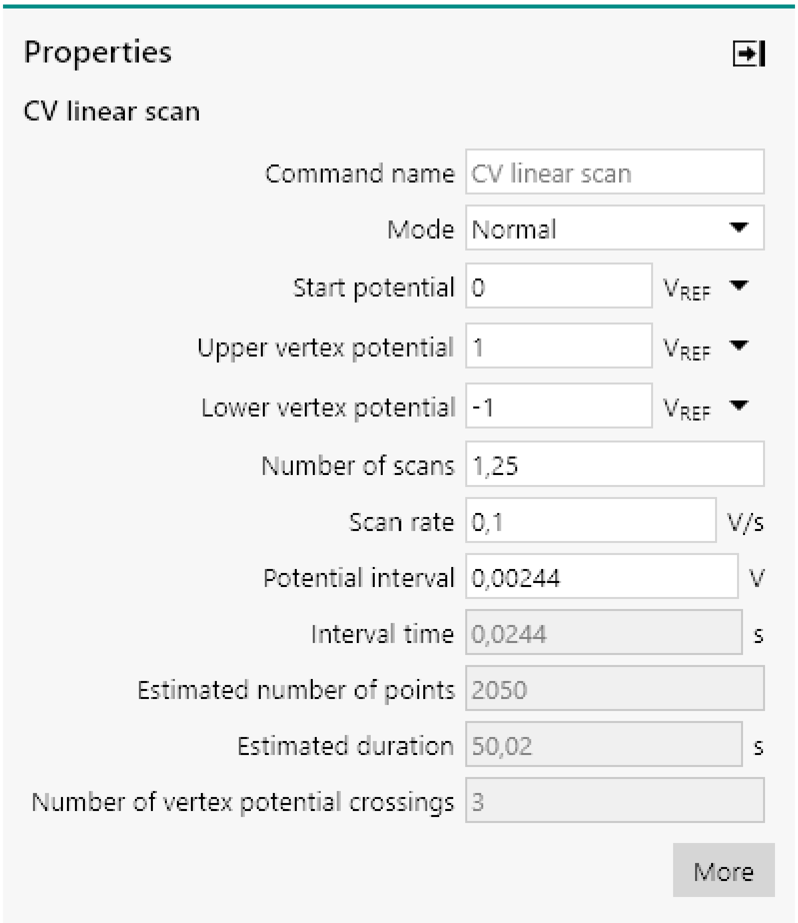


Figure 307 CV linear scan properties

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Mode:** specifies if the scan is performed in normal mode or high speed mode. This parameter is only shown when the optional **ADC10M** or **ADC750** module is present in the instrument. For high speed potential scans (more than 10 V/s), the high speed mode is recommended.
- **Start potential:** the start potential, in V.
- **Upper vertex potential:** the upper vertex potential, in V.
- **Lower vertex potential:** the lower vertex potential, in V.
- **Stop on:** specifies if the scan should stop on one of the vertices or on the start potential value, using the provided drop-down list. This parameter is only shown when the optional **SCAN250** module is present.
- **Number of scans:** the number of potential scans.
- **Potential interval:** the potential interval between two consecutive data points. The interval can be positive or negative. With a positive interval, the scan starts from the start potential towards the upper vertex potential. With a negative interval, the scan direction is reversed.
- **Scan rate:** the scan rate of the potential sweep, in V/s.

Four additional properties are shown as *read-only*:

- **Interval time:** the time interval between two consecutive points in the scan. This property is defined by the potential interval and the scan rate.
- **Estimated number of points:** the estimated number of points in the scan. This property is defined by the start and stop potential and the potential interval.
- **Estimated duration:** the estimated duration of the command, in s. This property is defined by the estimated number of points and the interval time as well as the duration of the underlying commands, if applicable.
- **Number of vertex/start potential crossings:** the number of times the potential scan will cross one of the potential vertices or the start potential. This property is defined by the number of scans.

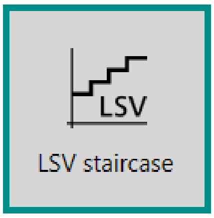
NOTE

For each potential value, a drop-down list provides the choice of the reference used to apply a potential value. The potential can be specified with respect to the reference electrode potential (V_{REF}) or the open-circuit potential (V_{OCP}).

NOTE

The **CV linear scan** command provides access to additional options, through the More button *(see Chapter 9, page 594)*.

7.3.3 LSV staircase

	This command can be used to perform a staircase linear sweep voltammetry measurement, in potentiostatic or galvanostatic conditions.

The details of the properties of the **LSV staircase** command are shown in *Figure 308*:

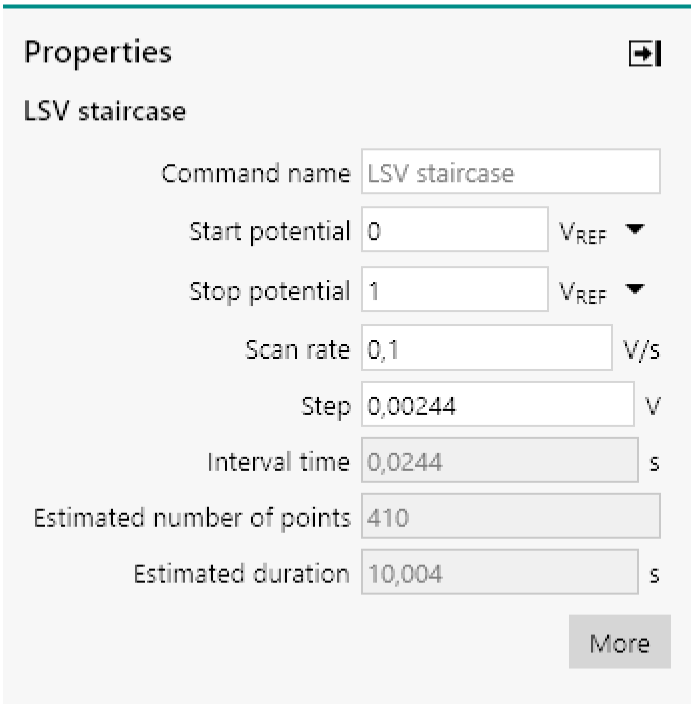


Figure 308 LSV staircase properties

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Start potential/current:** the start potential or current value, in V or A respectively.
- **Stop potential/current:** the stop potential or current value, in V or A respectively.
- **Scan rate:** the scan rate of the potential or current sweep, in V/s or A/s.
- **Step:** the potential or current step, in V or A respectively.

Three additional properties are shown as *read-only*:

- **Interval time:** the time interval between two consecutive points in the scan. This property is defined by the potential or current step and the scan rate.
- **Estimated number of points:** the estimated number of points in the scan. This property is defined by the start and stop potential or current and the potential or current step.
- **Estimated duration:** this property shows the estimated duration, in s. This property is defined by the estimated number of points and the interval time, as well as the duration of underlying commands, if applicable.

NOTE

When this command is used in potentiostatic mode, a drop-down list provides the choice of the reference used to apply a potential value. The potential can be specified with respect to the reference electrode potential (V_{REF}) or the open-circuit potential (V_{OCP}).

NOTE

The **LSV staircase** command provides access to additional options, through the More button *(see Chapter 9, page 594)*.

NOTE

The **Stop potential (or current)** and **Scan rate** properties can be modified in real time.

7.4 Measurement - voltammetric analysis commands

Commands located in the **Measurement – voltammetric analysis** group can be used to perform programmed potential measurements suitable for electroanalytical purposes.

The available commands are represented by a shortcut icon *(see Figure 309, page 250)*.

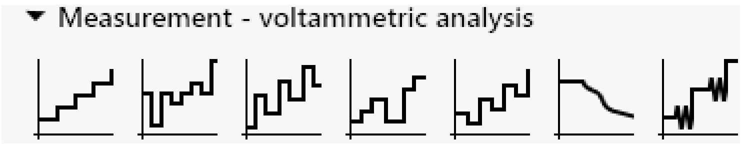


Figure 309 The Measurement - voltammetric analysis commands

The following commands are available:

- **Sampled DC voltammetry:** a command which can be used to perform a sampled DC measurement *(see Chapter 7.4.1, page 251)*.
- **Normal pulse voltammetry:** a command which can be used to perform a normal pulse voltammetry measurement *(see Chapter 7.4.2, page 253)*.

- **Differential pulse voltammetry:** a command which can be used to perform a differential pulse voltammetry measurement. The differential current is calculated during the measurement *(see Chapter 7.4.3, page 256)*.
- **Differential normal pulse voltammetry:** a command which can be used to perform a differential normal pulse voltammetry measurement. The differential current is calculated during the measurement *(see Chapter 7.4.4, page 259)*.
- **Square wave voltammetry:** a command which can be used to perform a square wave voltammetry measurement. The differential current is calculated during the measurement *(see Chapter 7.4.5, page 262)*.
- **Potentiometric stripping analysis:** a command which can be used to perform chemical and constant current potentiometric stripping analysis *(see Chapter 7.4.6, page 265)*.
- **AC voltammetry:** a command which can be used to perform an AC voltammetry measurement *(see Chapter 7.4.7, page 269)*.

7.4.1 Sampled DC voltammetry

	This command can be used to perform a sampled DC voltammetry measurement, in potentiostatic conditions.

CAUTION

This command can only be used in Potentiostatic mode.

The details of the properties of the **Sampled DC voltammetry** command are shown in *Figure 310*.

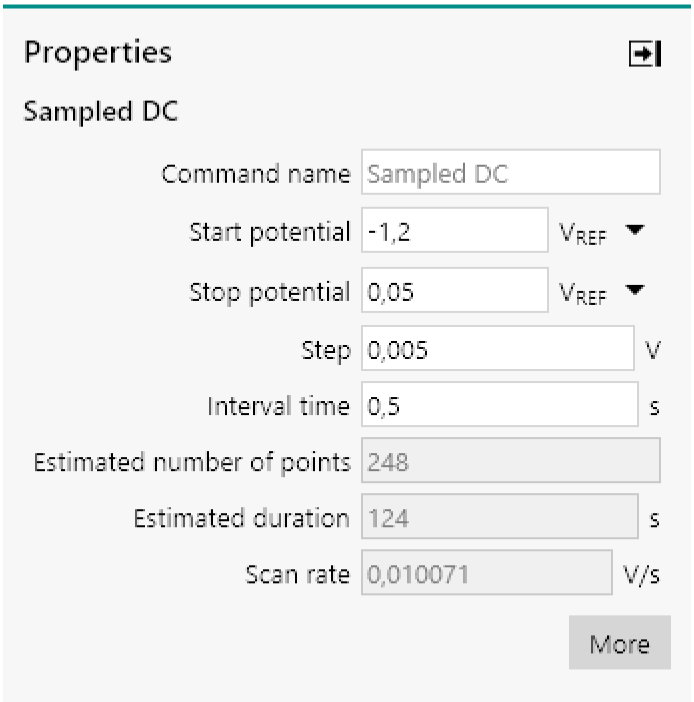


Figure 310 Sampled DC voltammetry properties

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Start potential:** the start potential value, in V.
- **Stop potential:** the stop potential value, in V.
- **Step:** the potential step, in V.
- **Interval time:** the duration of the interval time, in s.

Three additional properties are shown as *read-only*:

- **Estimated number of points:** the estimated number of points. This property is defined by the start and stop potential and the potential step.
- **Estimated duration:** this property shows the estimated duration, in s. This property is defined by the estimated number of points and the interval time, as well as the duration of underlying commands, if applicable.
- **Scan rate:** the calculated scan rate, in V/s, determined based on the step potential and the interval time.

Figure 317 represents the measurement properties of the **Sampled DC voltammetry** command, schematically.

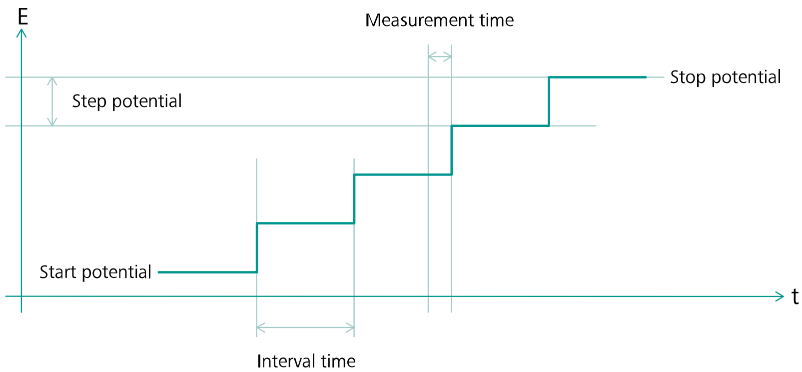


Figure 311 *Overview of the measurement properties of the Sampled DC voltammetry command*

NOTE

When this command is used, a drop-down list provides the choice of the reference used to apply a potential value. The potential can be specified with respect to the reference electrode potential (V_{REF}) or the open-circuit potential (V_{OCP}).

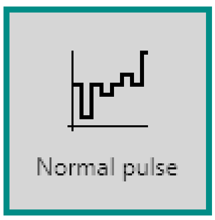
NOTE

The **Sampled DC voltammetry** command provides access to additional options, through the More button *(see Chapter 9, page 594)*.

NOTE

The **Stop potential**, **Step** and **Interval time** properties can be modified in real time.

7.4.2 Normal pulse voltammetry

	This command can be used to perform a normal pulse voltammetry measurement, in potentiostatic conditions.

CAUTION

This command can only be used in Potentiostatic mode.

The details of the properties of the **Normal pulse voltammetry** command are shown in *Figure 312*.

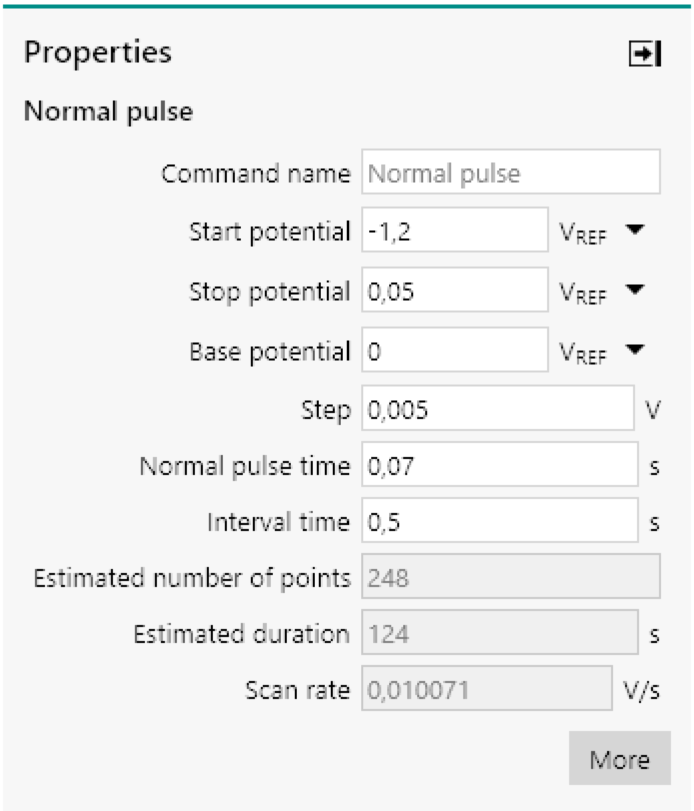


Figure 312 Normal pulse voltammetry properties

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Start potential:** the start potential value, in V.
- **Stop potential:** the stop potential value, in V.
- **Step:** the potential step, in V.
- **Base potential:** the base potential, in V.
- **Normal pulse time:** the duration of the normal pulse, in s.
- **Interval time:** the duration of the interval time, in s.

Three additional properties are shown as *read-only*:

- **Estimated number of points:** the estimated number of points. This property is defined by the start and stop potential and the potential step.

- **Estimated duration:** this property shows the estimated duration, in s. This property is defined by the estimated number of points and the interval time, as well as the duration of underlying commands, if applicable.
- **Scan rate:** the calculated scan rate, in V/s, determined based on the step potential and the interval time.

Figure 313 represents the measurement properties of the **Normal pulse voltammetry** command, schematically.

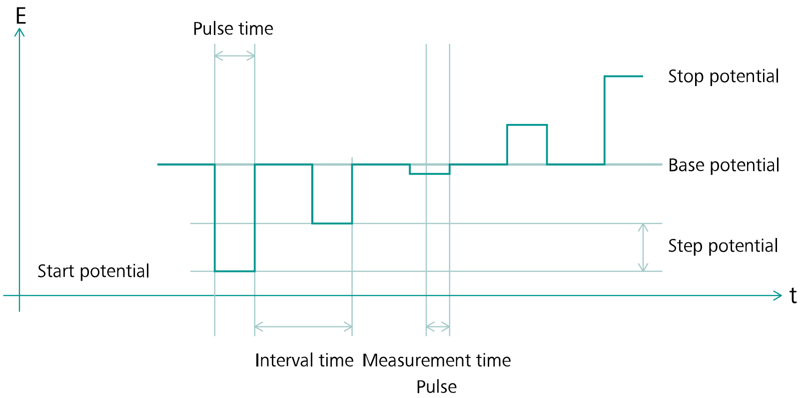


Figure 313 Overview of the measurement properties of the Normal pulse voltammetry command

NOTE

When this command is used, a drop-down list provides the choice of the reference used to apply a potential value. The potential can be specified with respect to the reference electrode potential (V_{REF}) or the open-circuit potential (V_{OCP}).

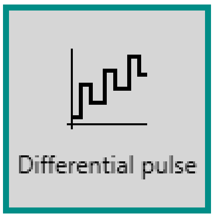
NOTE

The **Normal pulse voltammetry** command provides access to additional options, through the More button *(see Chapter 9, page 594)*.

NOTE

The **Stop potential**, **Base potential**, **Step**, **Normal pulse time** and **Interval time** properties can be modified in real time.

7.4.3 Differential pulse voltammetry



This command can be used to perform a differential pulse voltammetry measurement, in potentiostatic conditions.

CAUTION

This command can only be used in Potentiostatic mode.

The details of the properties of the **Differential pulse voltammetry** command are shown in *Figure 314*.

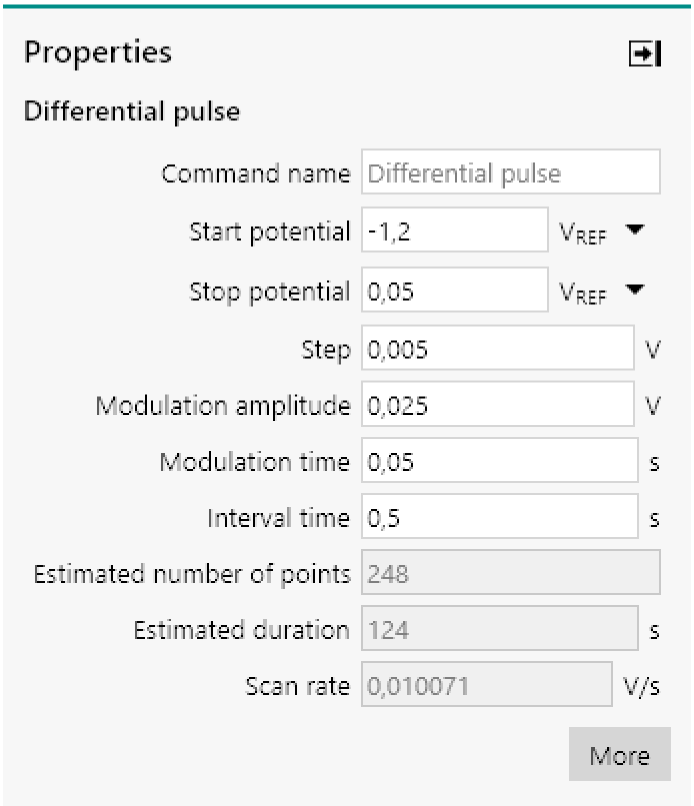


Figure 314 Differential pulse voltammetry properties

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Start potential:** the start potential value, in V.
- **Stop potential:** the stop potential value, in V.
- **Step:** the potential step, in V.

- **Modulation amplitude:** the amplitude of the potential modulation, in V.
- **Modulation time:** the duration of the potential modulation, in s.
- **Interval time:** the duration of the interval time, in s.

Three additional properties are shown as *read-only*:

- **Estimated number of points:** the estimated number of points. This property is defined by the start and stop potential and the potential step.
- **Estimated duration:** this property shows the estimated duration, in s. This property is defined by the estimated number of points and the interval time, as well as the duration of underlying commands, if applicable.
- **Scan rate:** the calculated scan rate, in V/s, determined based on the step potential and the interval time.

CAUTION

The definition of the **modulation amplitude** property is consistent with the definition used the Metrohm Computrace and VIVA software packages. When this value is positive, the pulse will be applied in the same direction as the potential scan (positive pulse in the positive going direction and negative pulse in the negative going direction). When this value is negative, the pulse will be applied in the **reverse** direction as the potential scan (negative pulse in the positive going direction and positive pulse in the negative going direction). This definition differs from the definition used in NOVA 1.X. Procedures imported from NOVA 1.X are automatically converted to the new definition.

Figure 315 represents the measurement properties of the **Differential pulse voltammetry** command, schematically.

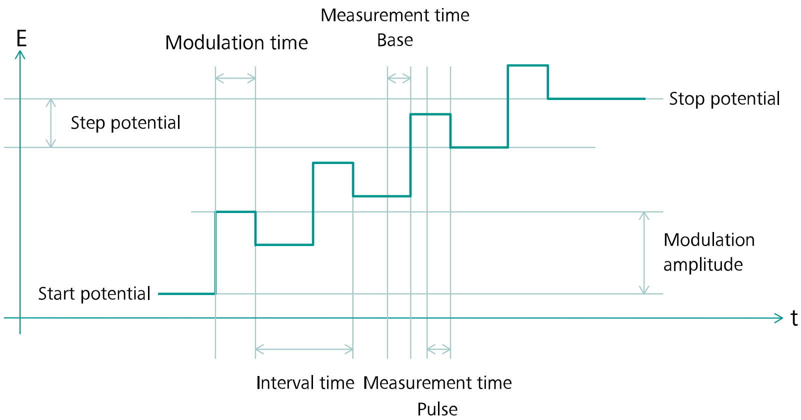


Figure 315 Overview of the measurement properties of the Differential pulse voltammetry command

In a **Differential pulse voltammetry** measurement, two consecutive current samples are collected for each step. The current value measured in the first part of the step corresponds to the WE(1).Base.Current signal while the current value measured at the end of the pulse corresponds to the WE(1).Pulse.Current signal. The differential value, corresponding to the WE(1).δ.Current signal is given by the difference of the pulse and the base current values

NOTE

When this command is used, a drop-down list provides the choice of the reference used to apply a potential value. The potential can be specified with respect to the reference electrode potential (V_{REF}) or the open-circuit potential (V_{OCP}).

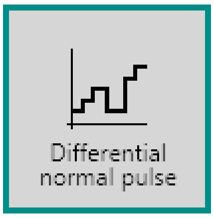
NOTE

The **Differential pulse voltammetry** command provides access to additional options, through the More button *(see Chapter 9, page 594)*.

NOTE

The **Stop potential**, **Step**, **Modulation amplitude**, **Modulation time** and **Interval time** properties can be modified in real time.

7.4.4 Differential normal pulse voltammetry



This command can be used to perform a differential normal pulse voltammetry measurement, in potentiostatic conditions.

CAUTION

This command can only be used in Potentiostatic mode.

The details of the properties of the **Differential normal pulse voltammetry** command are shown in *Figure 316*.

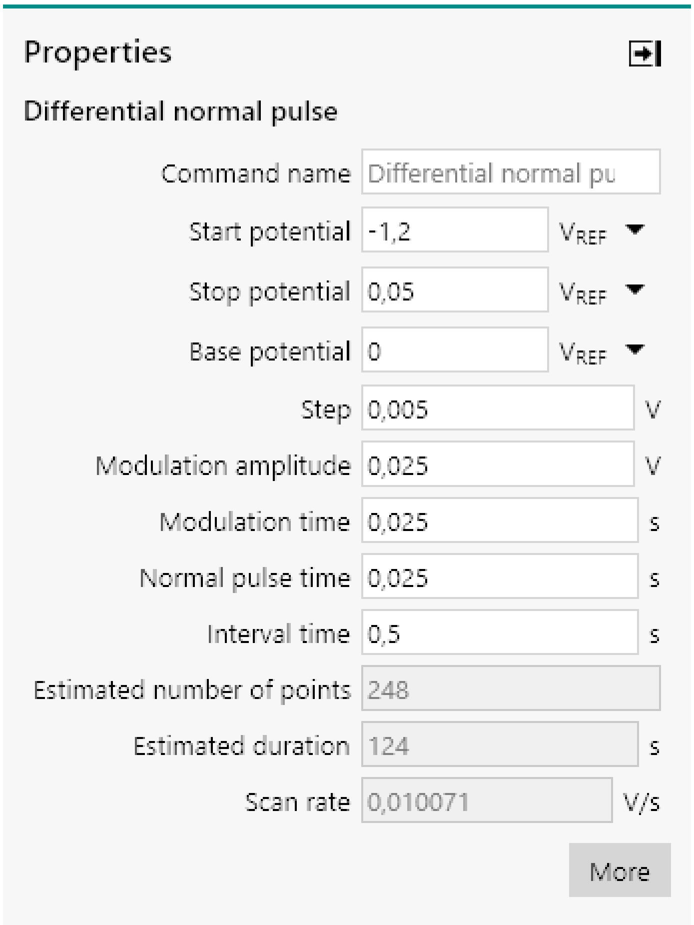


Figure 316 Differential normal pulse voltammetry properties

The following properties are available:

- **Command name:** a user-defined name for the command.

- **Start potential:** the start potential value, in V.
- **Stop potential:** the stop potential value, in V.
- **Base potential:** the base potential, in V.
- **Step:** the potential step, in V.
- **Modulation amplitude:** the amplitude of the potential modulation, in V.
- **Modulation time:** the duration of the potential modulation, in s.
- **Normal pulse time:** the duration of the normal pulse, in s.
- **Interval time:** the duration of the interval time, in s.

CAUTION

The definition of the **modulation amplitude** property is consistent with the definition used the Metrohm Computrace and VIVA software packages. When this value is positive, the pulse will be applied in the same direction as the potential scan (positive pulse in the positive going direction and negative pulse in the negative going direction). When this value is negative, the pulse will be applied in the **reverse** direction as the potential scan (negative pulse in the positive going direction and positive pulse in the negative going direction). This definition differs from the definition used in NOVA 1.X. Procedures imported from NOVA 1.X are automatically converted to the new definition.

Three additional properties are shown as *read-only*:

- **Estimated number of points:** the estimated number of points. This property is defined by the start and stop potential and the potential step.
- **Estimated duration:** this property shows the estimated duration, in s. This property is defined by the estimated number of points and the interval time, as well as the duration of underlying commands, if applicable.
- **Scan rate (V/s):** the calculated scan rate, determined based on the step potential and the interval time.

Figure 317 represents the measurement properties of the **Differential normal pulse voltammetry** command, schematically.

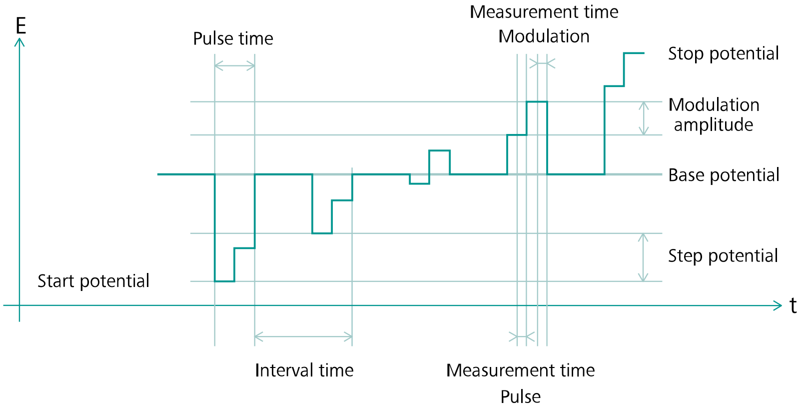


Figure 317 Overview of the measurement properties of the Differential normal pulse voltammetry command

CAUTION

The implementation of Differential normal pulse voltammetry is different from the description provided in *Electrochemistry* by C. M. A. Brett and A. M. Oliveira Brett, Oxford University Press, 1993.

In a **Differential normal pulse voltammetry** measurement, two consecutive current samples are collected for each step. The current value measured in pulse of the step corresponds to the WE(1).Pulse.Current signal while the current value measured at the end of the modulation corresponds to the WE(1).Modulation.Current signal. The difference between the modulation and the pulse current corresponds to the WE(1).δ.Current signal.

NOTE

When this command is used, a drop-down list provides the choice of the reference used to apply a potential value. The potential can be specified with respect to the reference electrode potential (V_{REF}) or the open-circuit potential (V_{OCP}).

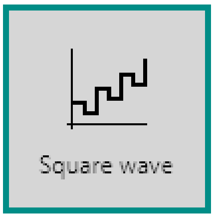
NOTE

The **Differential normal pulse voltammetry** command provides access to additional options, through the More button *(see Chapter 9, page 594)*.

NOTE

The **Stop potential**, **Base potential**, **Step**, **Modulation amplitude**, **Modulation time**, **Normal pulse time** and **Interval time** properties can be modified in real time.

7.4.5 Square wave voltammetry

	This command can be used to perform a square wave voltammetry measurement, in potentiostatic conditions.

CAUTION

This command can only be used in Potentiostatic mode.

The details of the properties of the **Square wave voltammetry** command are shown in *Figure 318*.

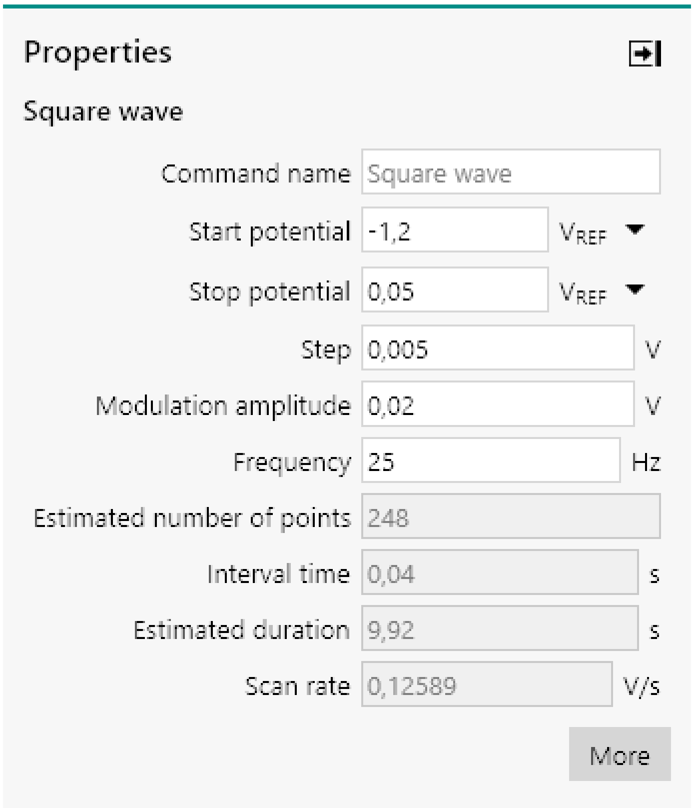


Figure 318 Square wave voltammetry properties

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Start potential:** the start potential value, in V.
- **Stop potential:** the stop potential value, in V.
- **Step:** the potential step, in V.
- **Modulation amplitude:** the amplitude of the square wave, in V.
- **Frequency:** the frequency of the square wave, in Hz.

Four additional properties are shown as *read-only*:

- **Estimated number of points:** the estimated number of points. This property is defined by the start and stop potential and the potential step.
- **Interval time:** the calculated interval time, based on the value of the frequency.
- **Estimated duration:** this property shows the estimated duration, in s. This property is defined by the estimated number of points and the interval time, as well as the duration of underlying commands, if applicable.
- **Scan rate:** the calculated scan rate, in V/s, determined based on the step potential and the interval time.

Figure 319 represents the measurement properties of the **Square wave voltammetry** command, schematically.

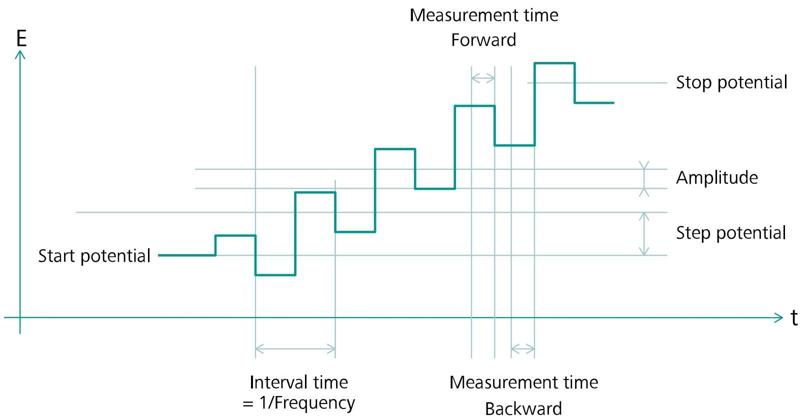


Figure 319 Overview of the measurement properties of the Square wave voltammetry command

In a **Square wave voltammetry** measurement, two consecutive current samples are collected for each step. The current value measured in first half of the step corresponds to the WE(1).Forward.Current signal while the current value measured in the second half of the step corresponds to the WE(1).Backward.Current signal. The difference between the backward and forward currents corresponds to the WE(1).δ.Current signal

NOTE

When this command is used, a drop-down list provides the choice of the reference used to apply a potential value. The potential can be specified with respect to the reference electrode potential (V_{REF}) or the open-circuit potential (V_{OCP}).

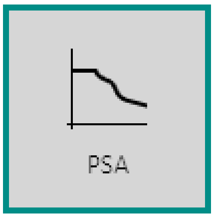
NOTE

The **Square wave voltammetry** command provides access to additional options, through the More button *(see Chapter 9, page 594)*.

NOTE

The **Stop potential**, **Step**, **Amplitude** and **Frequency** properties can be modified in real time.

7.4.6 PSA (Potentiometric stripping analysis)

	This command can be used to perform potentiometric stripping analysis (PSA) measurements.

The **PSA** command can be used in two different modes, which can be selected using the provided drop-down list *(see Figure 320, page 265)*:

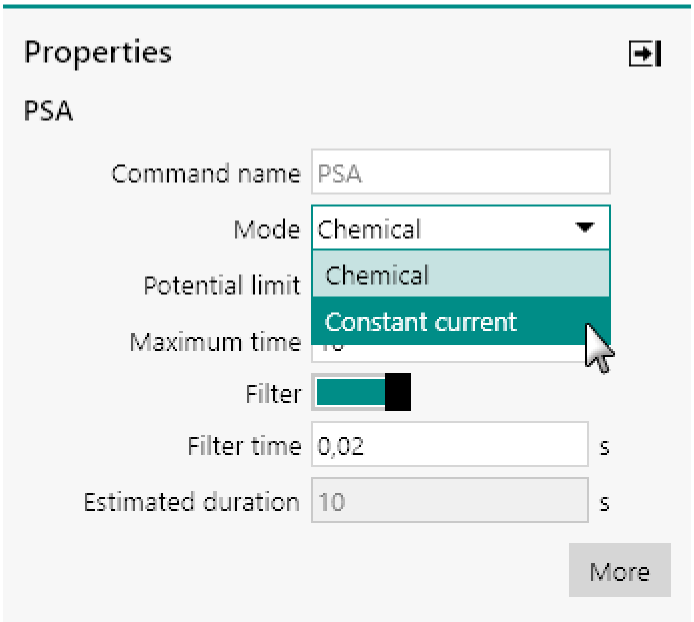


Figure 320 Two modes are provided by the PSA command

1. Chemical (default mode)
2. Constant current

CAUTION

No additional signals can be measured by the **PSA** command. The sampling rate is set to the highest possible value during this type of measurement and the measured data cannot be displayed in real time. Options like cutoffs and counters cannot be used.

During a potentiometric stripping analysis measurement, the potential of the working electrode is recorded as a function of time while a chemical or electrochemical oxidation is taking place. As such, this method is the equivalent of a potentiometric titration, in which the titrant is added *in situ* at a constant rate. The measurement stops when the maximum time is reached or when the measured potential exceeds a user defined limit. A typical potentiometric stripping analysis potential profile is shown in *Figure 321*.

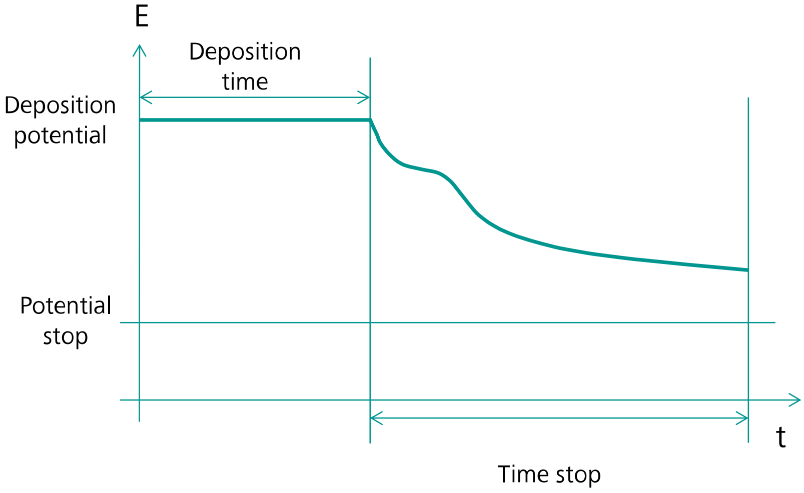


Figure 321 A typical potentiometric stripping analysis measurement

The voltage measurement E versus time is used to calculate the retention times dt/dE vs E. *Figure 322* shows an example of the E vs t measurement and the resulting peak-shaped plot.

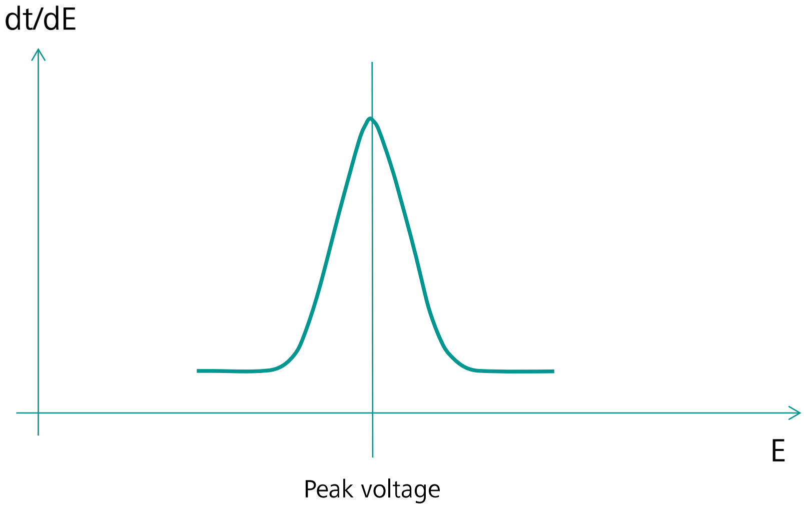


Figure 322 dt/dE versus potential curve

The peak voltage position is characteristic of the substance, the peak area is proportional to its concentration.

7.4.6.1 Chemical PSA

The following properties are available when the command is used in the *Chemical* mode *(see Figure 323, page 267)*:

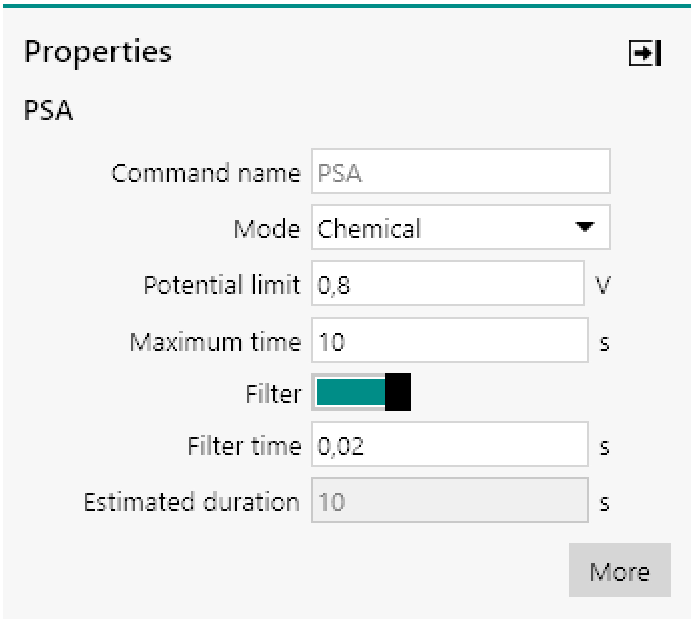


Figure 323 Chemical mode properties

- **Command name:** a user-defined name for the command.
- **Potential limit:** the maximum potential of the working electrode, in V. The measurement stops when the potential of the working electrode exceeds the specified value.
- **Maximum time:** the maximum duration of the stripping stage, in s. The measurement stops when this limit is reached.
- **Filter:** specifies if a filter must be applied on the measured potential signal using the provided toggle. The implemented filter is based on a moving average over the specified *Filter time* property.
- **Filter time:** the filter time, in s, used if the filter is On. This property is automatically set to 20 ms or 16.66 ms depending on the line frequency specified in the hardware (50 Hz or 60 Hz, respectively).
- **Estimated duration:** this property shows the estimated duration, in s. This property is defined by the maximum time and the duration of underlying commands, if applicable.

7.4.6.2 Constant current PSA

The following properties are available when the command is used in the *Constant current* mode *(see Figure 324, page 268)*:

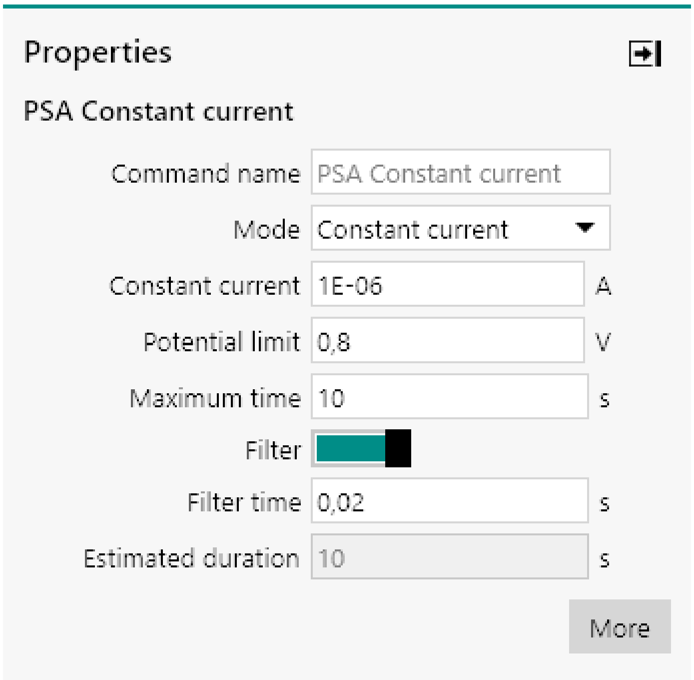


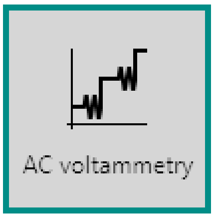
Figure 324   Constant current mode properties

- **Command name:** a user-defined name for the command.
- **Stripping current:** the current applied during the stripping stage, in A.
- **Potential limit:** the maximum potential of the working electrode, in V. The measurement stops when the potential of the working electrode exceeds the specified value.
- **Maximum time:** the maximum duration of the stripping stage, in s. The measurement stops when this limit is reached.
- **Filter:** specifies if a filter must be applied on the measured potential signal using the provided toggle. The implemented filter is based on a moving average over the specified *Filter time* property.
- **Filter time:** the filter time, in s, used if the filter is On. This property is automatically set to 20 ms or 16.66 ms depending on the line frequency specified in the hardware (50 Hz or 60 Hz, respectively).
- **Estimated duration:** this property shows the estimated duration, in s. This property is defined by the maximum time and the duration of underlying commands, if applicable.

NOTE

The current range needs to be adjusted in order to apply the stripping current properly. This can be done using the **Autolab control** command.

7.4.7 AC voltammetry

	This command can be used to perform an AC voltammetry measurement, in potentiostatic conditions.

CAUTION

This command can only be used in Potentiostatic mode.

The details of the properties of the **AC voltammetry** command are shown in *Figure 325*.

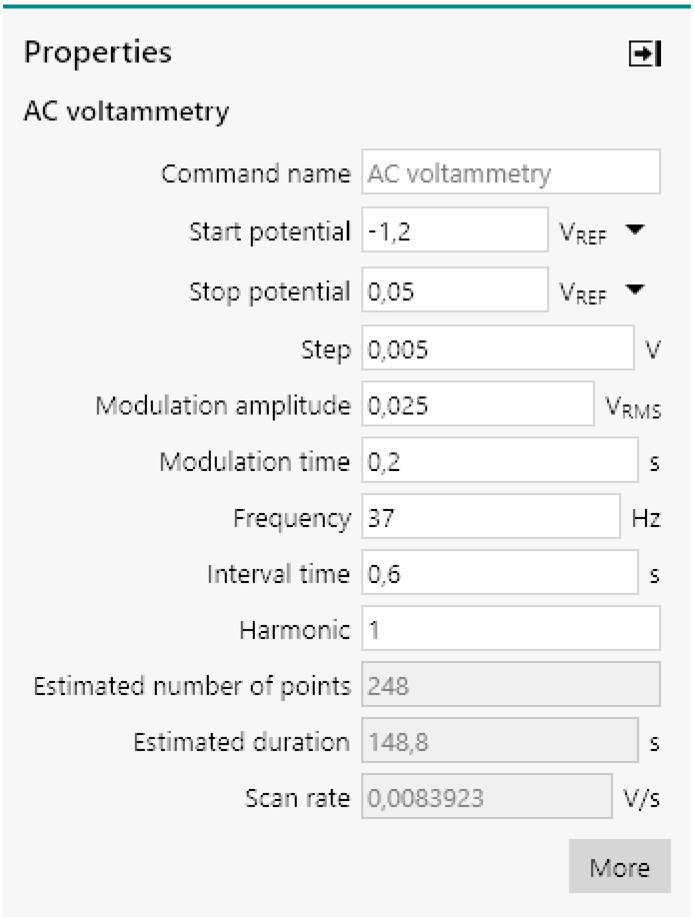


Figure 325 AC voltammetry properties

The following properties are available:

- **Command name:** a user-defined name for the command.

- **Start potential:** the start potential value, in V.
- **Stop potential:** the stop potential value, in V.
- **Step:** the potential step, in V.
- **Modulation amplitude:** the amplitude of the AC modulation, in V. This amplitude is specified as a RMS value.
- **Modulation time:** the duration of the AC modulation, in s.
- **Frequency:** the frequency of the AC modulation, in Hz.
- **Interval time:** the interval time, in s.
- **Harmonic:** specifies if the measurement should be carried out using the first or second harmonic (1 or 2 respectively).

Three additional properties are shown as *read-only*:

- **Estimated number of points:** the estimated number of points. This property is defined by the start and stop potential and the potential step.
- **Estimated duration:** this property shows the estimated duration, in s. This property is defined by the estimated number of points and the interval time, as well as the duration of underlying commands, if applicable.
- **Scan rate:** the calculated scan rate, in V/s, determined based on the step potential and the interval time.

Figure 326 represents the measurement properties of the **AC voltammetry** command, schematically.

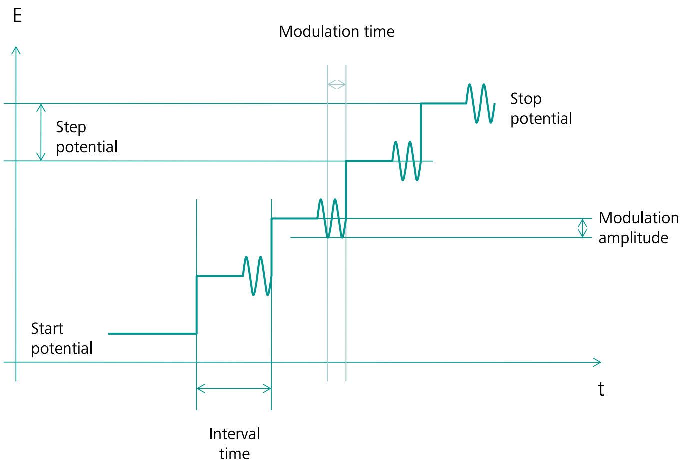


Figure 326 Overview of the measurement properties of the AC voltammetry command

In a **AC voltammetry** measurement, the AC current is measured while the modulation is applied. This value is normally plotted against the applied potential. Alongside the AC current signal, the calculated impedance, admittance and phase shift are also available. Additional signals, provided by the Autolab instrument, can be sampled as well.

NOTE

When this command is used, a drop-down list provides the choice of the reference used to apply a potential value. The potential can be specified with respect to the reference electrode potential (V_{REF}) or the open-circuit potential (V_{OCP}).

NOTE

The **AC voltammetry** command provides access to additional options, through the More button *(see Chapter 9, page 594)*.

NOTE

The **Stop potential**, **Step** and **Interval time** properties can be modified in real time.

7.5 Measurement - chrono methods commands

Commands located in the **Measurement – chrono methods** group can be used to perform time-resolved measurements.

The available commands are represented by a shortcut icon *(see Figure 327, page 271)*.

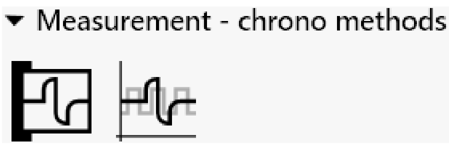
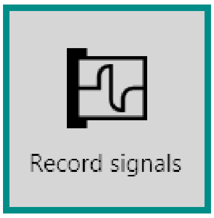


Figure 327 The Measurement - chrono methods commands

The following commands are available:

- **Record signals:** a command which can be used to records the signals provided by the instrument in time *(see Chapter 7.5.1, page 272)*.
- **Chrono methods:** a command which can be used to apply a sequence of potential or current steps and record the response from the electrochemical cell *(see Chapter 7.5.2, page 275)*.

7.5.1 Record signals



This command can be used to record one or more signals with interval time of 1.33 ms or larger.

NOTE

The **Record signals** command does not apply any potential or current.

The details of the properties of the **Record signals** command are shown in *Figure 328*:

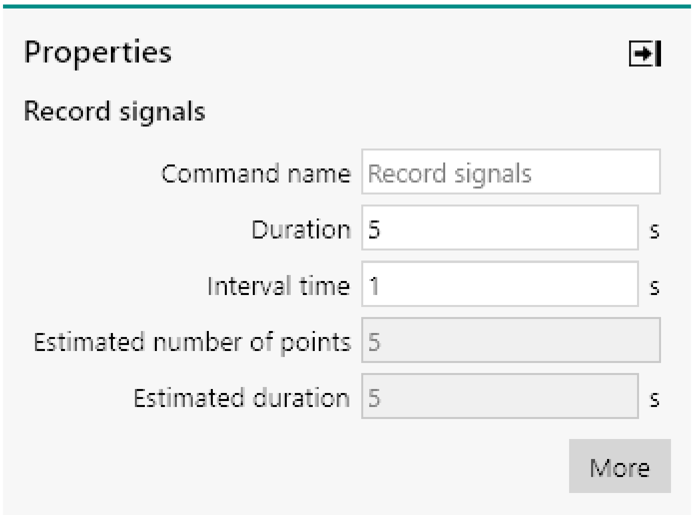


Figure 328 The properties of the Record signals command

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Duration:** specifies the duration of the measurement, in s.
- **Interval time:** specifies the interval time used in the measurement, in s. The smallest interval time is 1.33 ms.
- **Estimated number of points:** this property shows the estimated number of points, determined from the specified Duration and Interval time.
- **Estimated duration:** this property shows the estimated duration, in s. This property is defined by the estimated number of points and the interval time, as well as the duration of underlying commands, if applicable.

The **Record signals** command provides access to additional measurement options, through the More button *(see Chapter 9, page 594)*.

The following additional properties are available *(see Figure 329, page 273)*:

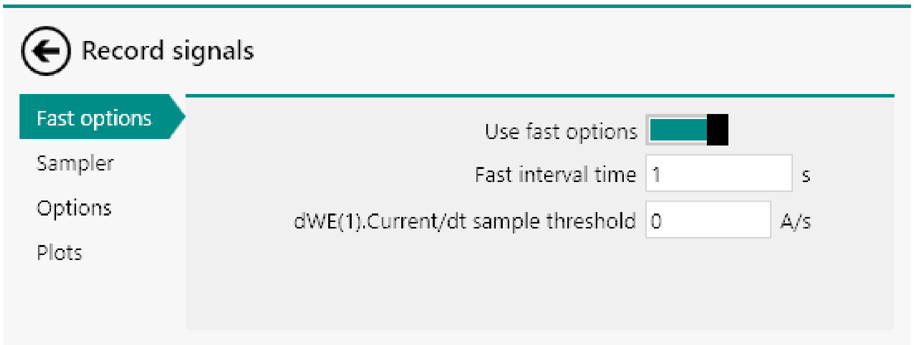


Figure 329 The fast options of the Record signals command

- **Use fast options:** a toggle control that can be used to specify if the fast options are used.
- **Fast interval time:** specifies the fast interval time, in s, used in the measurement. This property is only available if the fast options are used. The fast interval time must be smaller than the interval time and must be an integral fraction of the interval time.
- **dX/dt sample threshold:** specifies a threshold value for the time derivative of a sampled signal, in signal units per second. This property is only available when the a time derivative of a signal is sampled and if the fast options are used.

NOTE

The **Record signals** command provides access to additional options, through the More button *(see Chapter 9, page 594)*.

Using the provided properties, the **Record signal** command can be used in three different ways:

1. Using the default parameters
2. Using the fast options
3. Using the fast options and the time derivative threshold value

7.5.1.1 Using the default properties

When the default properties are used, the signals defined in the signal sampler are measured for the specified duration. Each data point is recorded after the user-defined interval time. The measurement options are verified after each interval time.

Using these properties, measurement options like **Counters** *(see Chapter 9.4, page 603)* and **Cutoffs** *(see Chapter 9.3, page 599)* are verified after each interval time.

7.5.1.2 Using the fast options

When the fast options are used, the same strategy as in the default mode used is for measuring the data points but the options are now verified after each user-defined fast interval time. This means that the options can be verified at a faster rate that the sampling rate.

NOTE

The fast interval time must be smaller than the interval time and must be an integral fraction of the interval time.

When the fast options are used, the measurement options are decoupled from the sampling of the data. This is particularly useful for long measurements on a cell that requires the options to be tested with a short interval time.

Figure 330 shows an example of such a set of properties. The duration and interval time are set to 10 s and 0.2 seconds, respectively. This leads to an estimated number of points of 50. Setting the fast interval time to 20 ms, the options will be verified at a much faster rate that the sampling rate.

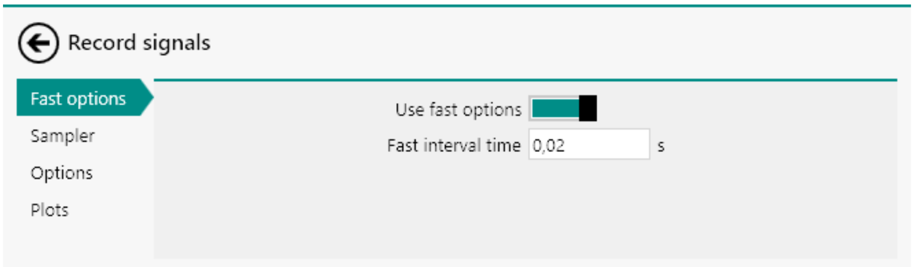


Figure 330 Example of fast options

7.5.1.3 Use the fast options and the time derivative threshold

When both the fast options and the time derivative threshold are used, the same strategy as in the previous mode is used. The time derivative value of one or more signals is determined using the fast interval time. For each time derivative signal, a threshold can be defined by the user. When the absolute value of a time derivative signal exceeds the specified threshold, the data points are measured using the fast interval time instead of the interval time. This means that the sampling rate can be modified depending on the derivative of one or more signals.

NOTE

In order to use the time derivative threshold, at least one time derivative must be sampled.

NOTE

The fast interval time must be smaller than the interval time and must be an integral fraction of the interval time.

Figure 331 shows an example of such a set of properties. The duration and interval time are set to 10 s and 0.2 seconds, respectively. This leads to an estimated number of points of 50. Setting the fast interval time to 20 ms, the options will be verified at a much faster rate that the sampling rate. Furthermore, if the dWE(1).Potential/dt signal is measured, a threshold value can be specified for this signal. In this example, the threshold is set to 1 V/s.

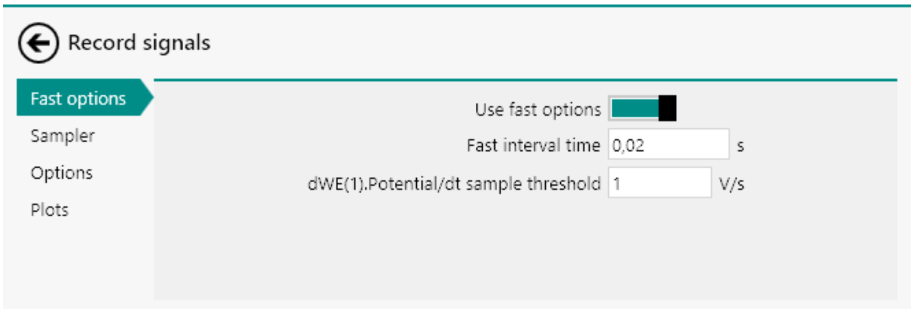
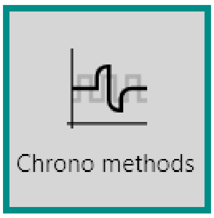


Figure 331 Example of time derivative threshold

Using these properties, the dWE(1).Potential/dt signal is calculated every 20 ms. If the value of this signal is larger (in absolute value) than the specified threshold, a data point is collected using the fast interval time instead of the interval time.

7.5.2 Chrono methods

	This command can be used to record the signals during a user-defined sequence of potential or current steps.

The **Chrono methods** command can be used in two different modes, which can be selected using the provided drop-down list *(see Figure 332, page 276)*:

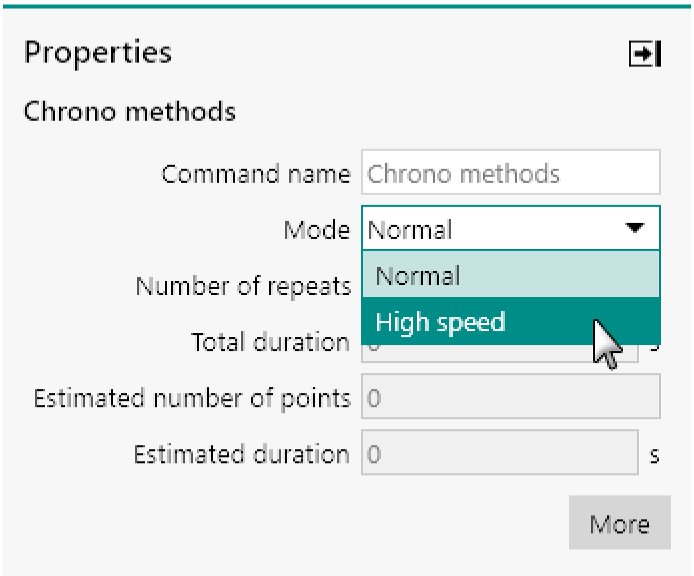


Figure 332 Two modes are provided by the Chrono methods command

- Normal
- High speed

NOTE

The **Mode** selection drop-down is only shown for instruments that are equipped with a fast sampling ADC module (**ADC10M** or **ADC750**). Please refer to *Chapter 16.3.2.1* and *Chapter 16.3.2.2* for more information. Instruments that are not fitted with a fast sampling ADC module can only use the Normal mode. For those instruments, the **Mode** drop-down control is not shown.

7.5.2.1 Chrono methods - Normal

The following properties are available when the **Chrono methods** command is used in **Normal** mode *(see Figure 333, page 277)*:

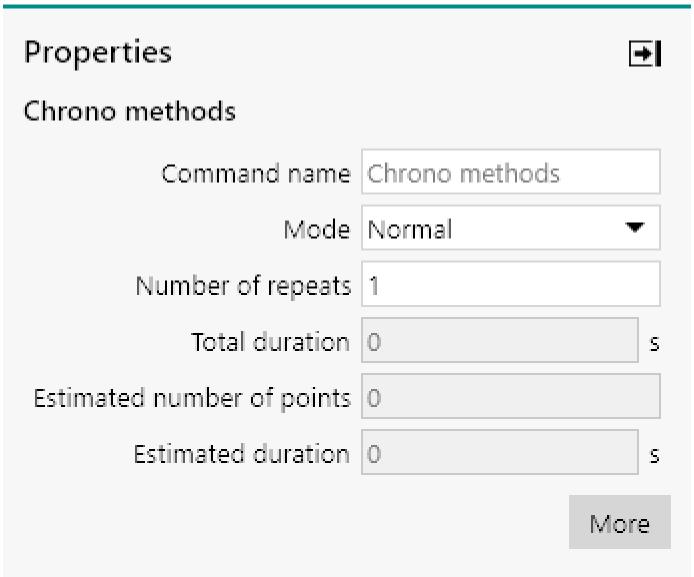


Figure 333 The properties of the Chrono methods command in Normal mode

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Mode:** this property defines the measurement mode of the Chrono methods command. A drop-down control provides the choice between Normal (default) and High speed.
- **Number of repeats:** specifies the number of times the chrono methods sequence should be repeated.
- **Total duration:** indicates the expected duration of the chrono methods measurement, as read-only property, in s.
- **Estimated number of points:** this read-only property shows the estimated number of points.
- **Estimated duration:** this read-only property shows the estimated duration, in s. This property is defined by the estimated number of points and the interval time, as well as the duration of underlying commands, if applicable.

The sequence of steps used by the **Chrono methods** command can be edited by clicking the button.

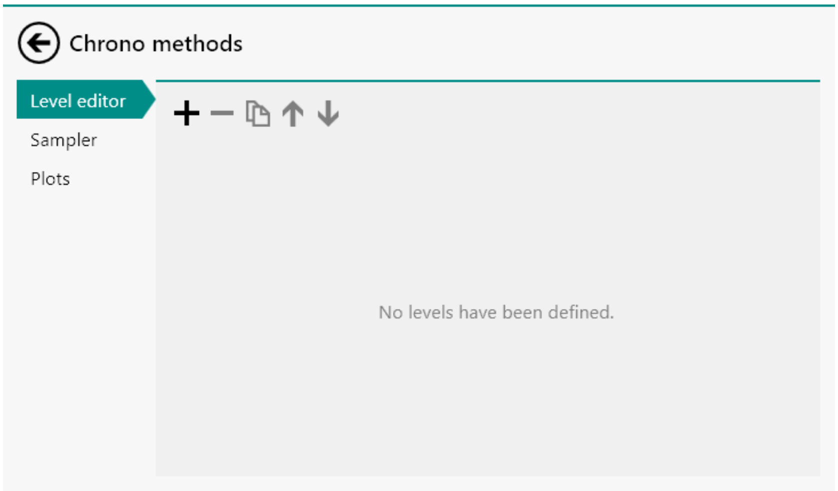


Figure 334 The Chrono methods Sequence editor in Normal mode

NOTE

Additional settings can be adjusted in the Sequence editor. Please refer to *Chapter 9* for more information.

7.5.2.2 Chrono methods - High speed

The following properties are available when the **Chrono methods** command is used in **High speed** mode *(see Figure 335, page 278)*:

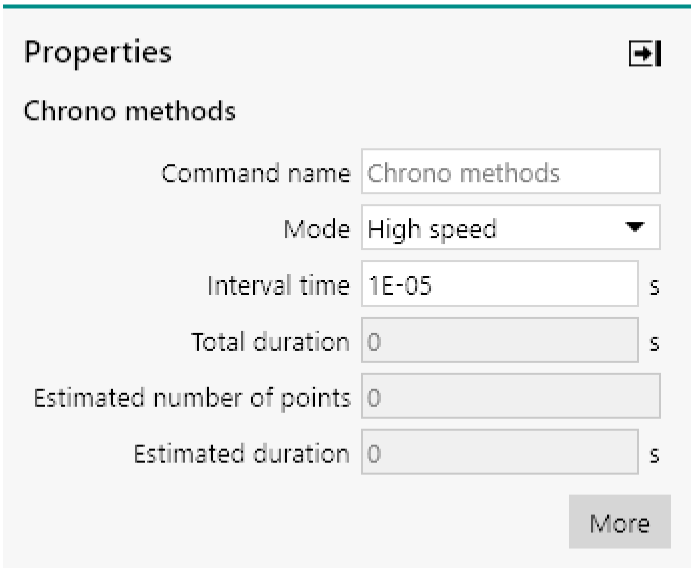


Figure 335 The properties of the Chrono methods command in High speed mode

The following properties are available:

- **Command name:** a user-defined name for the command.

- **Mode:** this property defines the measurement mode of the Chrono methods command. A drop-down control provides the choice between Normal (default) and High speed.
- **Interval time:** specifies the interval time, in s, used for the measurement.
- **Total duration:** indicates the expected duration of the chrono methods measurement, as read-only property, in s.
- **Estimated number of points:** this read-only property shows the estimated number of points.
- **Estimated duration:** this read-only property shows the estimated duration, in s. This property is defined by the estimated number of points and the interval time, as well as the duration of underlying commands, if applicable.

NOTE

In **High speed** mode, the interval time used in the **Chrono methods** command is constant.

The sequence of steps used by the **Chrono methods** command can be edited by clicking the More button.

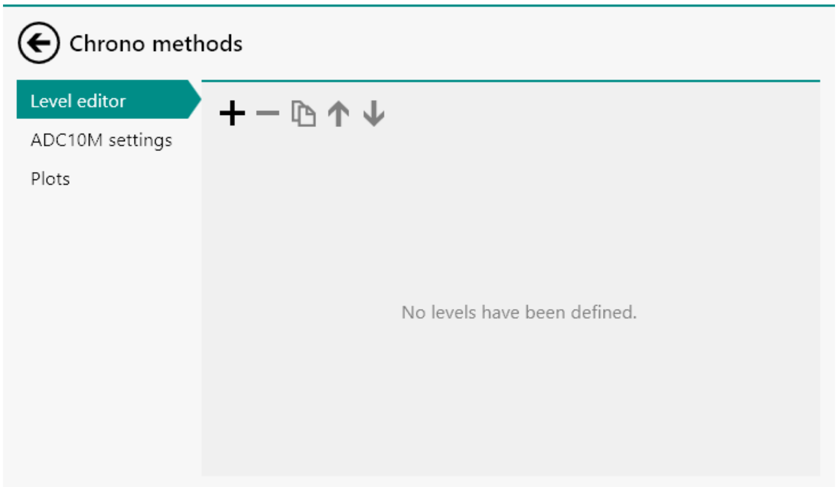


Figure 336 The Chrono methods Sequence editor in High speed mode

NOTE

Additional settings can be adjusted in the Sequence editor. Please refer to *Chapter 9* for more information.

7.5.2.3 Using the Sequence editor

Using the buttons located in the top left corner of the Sequence editor panel, the sequence of steps used in the **Chrono methods** command can be constructed.

The following tasks can be carried out using the Sequence editor:

1. Add an item to the sequence (using the + button)
2. Remove an item from the sequence (using the − button)
3. Duplicate an item in the sequence (using the ⧉ button)
4. Move a sequence item up or down (using the ↑ button and the ↓ button)

Clicking the + button in the Chrono methods Sequence editor reveals a drop-down list that can be used to add one of the following items:

- **Step:** this item creates a step in the sequence. A step applies a potential or current value on the cell for the specified duration during which the data is recorded.
- **Level:** this item creates a level in the sequence. A level does not apply a potential or current value on the cell. The data is recorded during the specified duration.
- **Repeat:** this item creates a new sub-sequence in the main sequence, in which new items can be added. This sub-sequence can be repeated any number of times and the electrochemical response of the cell is sampled during the whole sub-sequence.
- **Repeat (unsampled):** this item creates a sub-sequence in the main sequence, in which new items can be added. This sub sequence can be repeated any number of times, but does not generate any data points, as the electrochemical response of the cell is unsampled for the whole sub-sequence. This can be useful for conditioning the electrode with a pulse sequence, or for reducing the number of data points recorded during long measurements.

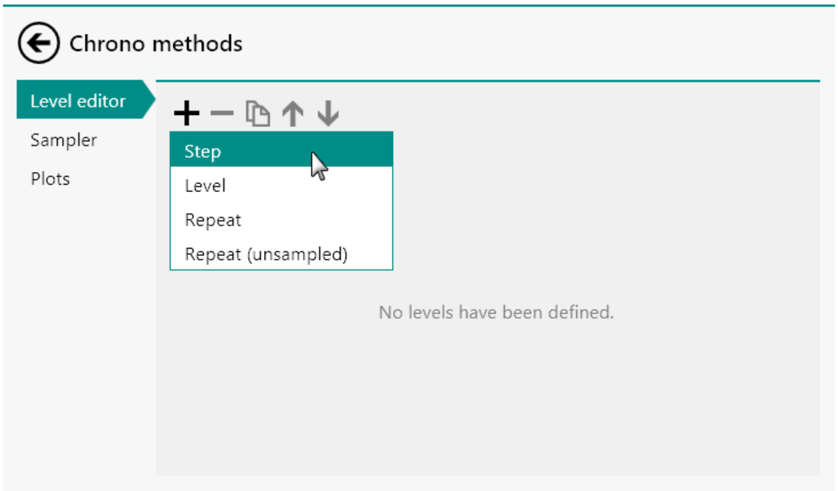


Figure 337 Add an item to the Sequence editor

NOTE

The **Level** and **Repeat (unsampled)** are not available when the **Chrono methods** command is used in **High speed** mode.

When an item is added to the sequence, it will be displayed on the left-hand side of the dedicated panel. Its properties will be displayed in the same panel.

7.5.2.3.1 Using the Step item

The Step item can be added to the Sequence editor by clicking the + button.

This item creates a step in the sequence. The Step applies a constant voltage or current, for the specified duration, during which the response of the cell can be measured.

The properties of the added Step are shown in the dedicated sub-panel *(see Figure 338, page 282)*.

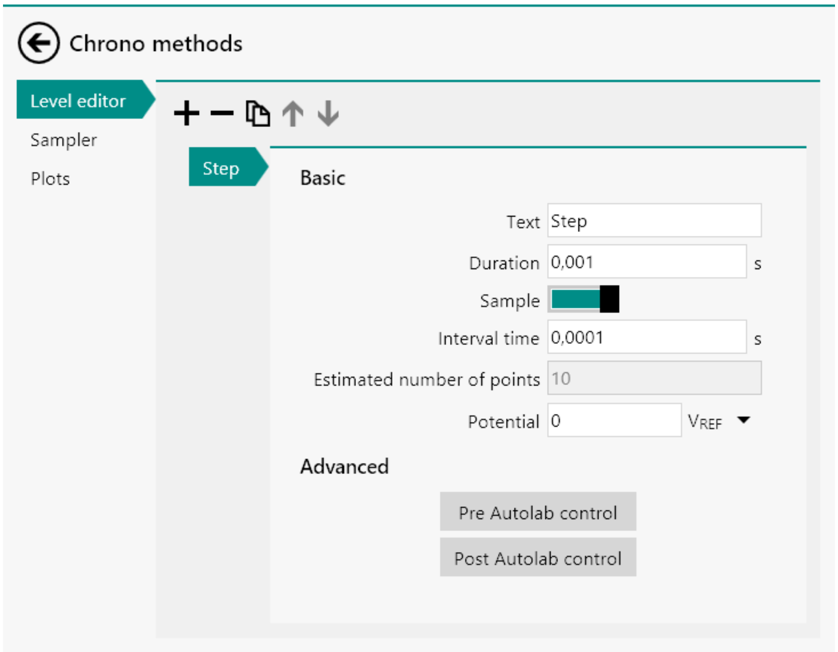


Figure 338 The properties of the Step item

Depending on the mode in which the **Chrono methods** command is used, the Step item has the following properties:

- **Text:** a label that can be used to provide a name to the Step, for bookkeeping purposes.
- **Duration:** the duration of the Step, in s.
- **Sample*:** a  toggle that can be used to switch the sampling of data on or off. When the sampling is switched off, the specified potential or current value will be applied but no data points will be measured.
- **Interval time*:** the interval time, in s, used for sampling the data during the Step.
- **Estimated number of points*:** a read-only property that indicates the expected number of data points, based on the interval time and the duration.
- **Potential/Current:** the potential or current, in V or A, applied during the Step, respectively.

NOTE

When the Chrono methods command is used in potentiostatic mode, a drop-down list provides the choice of reference used to apply a potential value. The potential can be specified with respect to the reference electrode potential (V_{REF}) or the open-circuit potential (V_{OCP}).

NOTE

The properties indicated by a * are not available when the **Chrono methods** command is used in **High speed** mode.

Two additional advanced properties are available, through the Pre Autolab control and the Post Autolab control buttons. These buttons can be used to define a specific instrument setting before the Step is applied or after the step is applied, respectively. Both buttons provide access to the **Autolab control** properties.

NOTE

For more information on the **Autolab control** command, please refer to *Chapter 7.2.1*.

7.5.2.3.2 Using the Level item

The Level item can be added to the Sequence editor by clicking the + button.

This item creates a Level in the sequence. The Level does not change the applied potential or current. The response of the cell is measured for the specified duration.

The properties of the added Level are shown in the dedicated sub-panel *(see Figure 339, page 284)*.

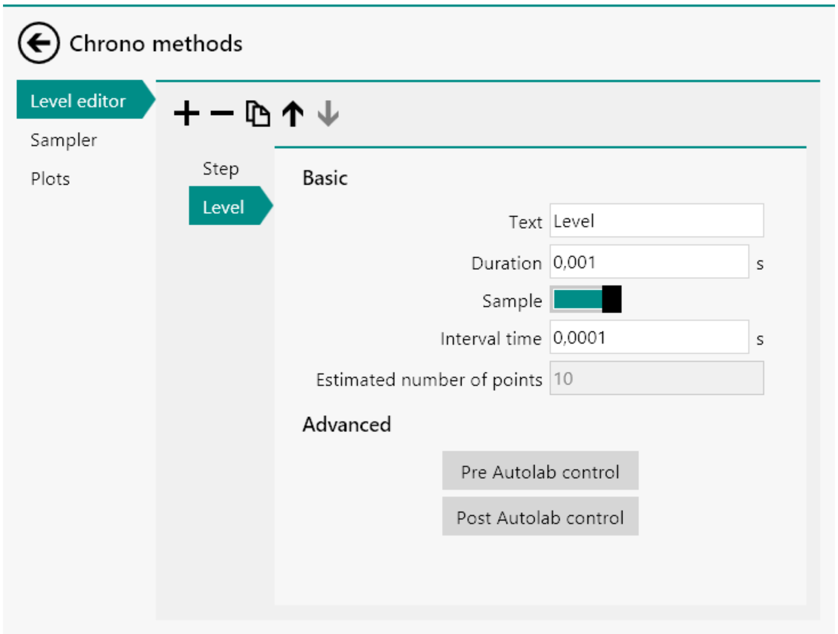


Figure 339 The properties of the Level item

NOTE

The Level item is not available when the **Chrono methods** command is used in **High speed** mode.

The Level item has the following properties:

- **Text:** a label that can be used to provide a name to the Level, for bookkeeping purposes.
- **Duration:** the duration of the Level, in s.
- **Sample:** a toggle that can be used to switch the sampling of data on or off. When the sampling is switched off, the specified potential or current value will be applied but no data points will be measured.
- **Interval time:** the interval time, in s, used for sampling the data during the Level.
- **Estimated number of points:** a read-only property that indicates the expected number of data points, based on the interval time and the duration.

NOTE

When the Chrono methods command is used in potentiostatic mode, a drop-down list provides the choice of reference used to apply a potential value. The potential can be specified with respect to the reference electrode potential (V_{REF}) or the open-circuit potential (V_{OCP}).

Two additional advanced properties are available, through the Pre Autolab control and the Post Autolab control buttons. These buttons can be used to define a specific instrument setting before the Level is applied or after the Level is applied, respectively. Both buttons provide access to the **Autolab control** properties.

NOTE

For more information on the **Autolab control** command, please refer to *Chapter 7.2.1*.

7.5.2.3.3 Using the Repeat item

The Repeat item can be added to the Sequence editor by clicking the + button.

This item creates a Repeat in the sequence. The Repeat item creates a subsequence to which new Steps or Levels can be added. The whole subsequence can be repeated.

The properties of the added Repeat are shown in the dedicated sub-panel *(see Figure 340, page 285)*.

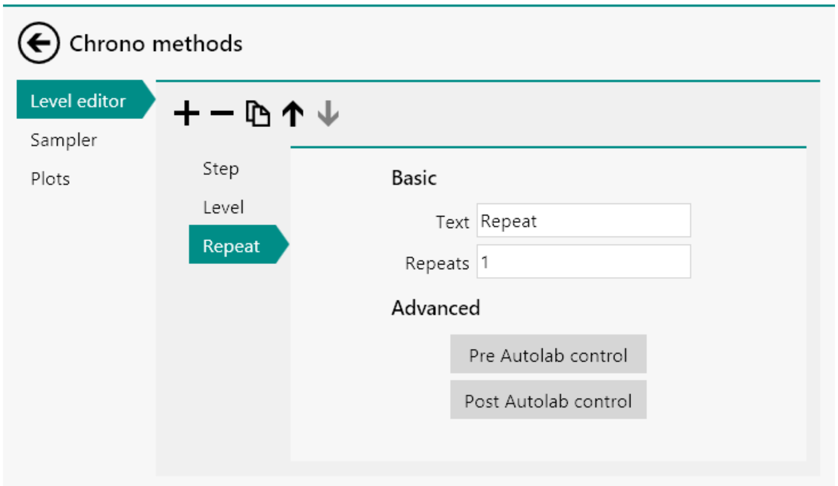


Figure 340 The properties of the Repeat item

The Repeat item has the following properties:

- **Text:** a label that can be used to provide a name to the Repeat, for bookkeeping purposes.
- **Repeats:** the number of repetitions of the sub-sequence.

Two additional advanced properties are available, through the Pre Autolab control and the Post Autolab control buttons. These buttons can be used to define a specific instrument setting before the Repeat is started or after the Repeat is finished, respectively. Both buttons provide access to the **Autolab control** properties.

NOTE

For more information on the **Autolab control** command, please refer to *Chapter 7.2.1*.

7.5.2.3.4 Using the Repeat (unsampled) item

The Repeat (unsampled) item can be added to the Sequence editor by clicking the + button.

This item creates a Repeat (unsampled) in the sequence. The Repeat (unsampled) item creates a sub-sequence to which new Steps or Levels can be added. The whole sub-sequence can be repeated, however no data will be sampled during this sub-sequence.

The properties of the added Repeat (unsampled) are shown in the dedicated sub-panel *(see Figure 340, page 285)*.

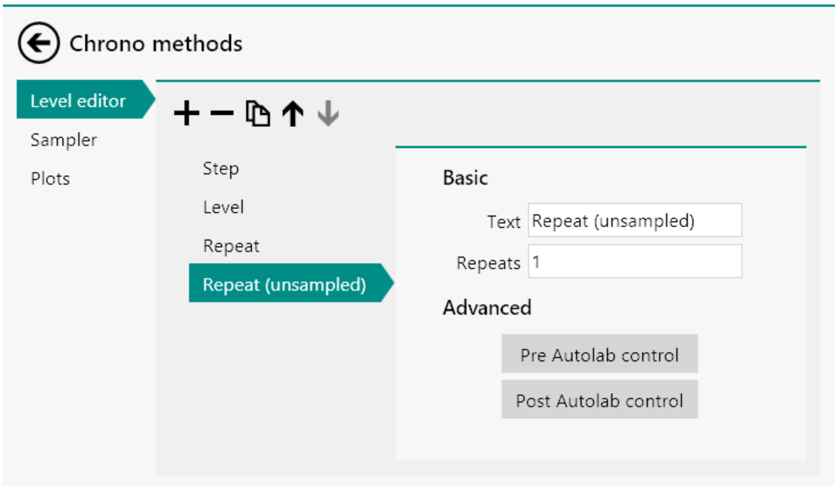


Figure 341 The properties of the Repeat (unsampled) item

NOTE

The Repeat (unsampled) item is not available when the **Chrono methods** command is used in High speed mode.

The Repeat (unsampled) item has the following properties:

- **Text:** a label that can be used to provide a name to the Repeat (unsampled), for bookkeeping purposes.
- **Repeat (unsampled):** the number of repetitions of the sub-sequence.

Two additional advanced properties are available, through the Pre Autolab control and the Post Autolab control buttons. These buttons can be used to define a specific instrument setting before the Repeat (unsampled) is started or after the Repeat (unsampled) is finished, respectively. Both buttons provide access to the **Autolab control** properties.

NOTE

For more information on the **Autolab control** command, please refer to *Chapter 7.2.1*.

7.6 Measurement - impedance commands

Commands located in the **Measurement – impedance** group can be used to perform impedance and impedance spectroscopy measurements or measurements involving sinewave modulations.

CAUTION

Impedance measurements require the optional **FRA32M** or **FRA2** module *(see Chapter 16.3.2.13, page 1091)*.

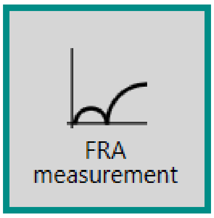
The available commands are represented by a shortcut icon *(see Figure 342, page 287)*

Figure 342 The Measurement - impedance commands

The following commands are available:

- **FRA measurement:** a command which can be used to perform a frequency scan *(see Chapter 7.6.1, page 288)*.
- **FRA single frequency:** a command which can be used to measure the impedance at a single frequency *(see Chapter 7.6.2, page 290)*.
- **Electrochemical frequency modulation:** a command which can be used to perform Electrochemical Frequency Modulation measurements *(see Chapter 7.6.4, page 312)*.

7.6.1 FRA measurement

	This command can be used to perform an impedance spectroscopy measurement through a frequency scan. This command requires the optional **FRA32M** or **FRA2** module.

CAUTION

The **FRA measurement** command requires the optional **FRA32M** or **FRA2** module *(see Chapter 16.3.2.13, page 1091)*.

The details of the properties of the **FRA measurement** command are shown in *Figure 343*:

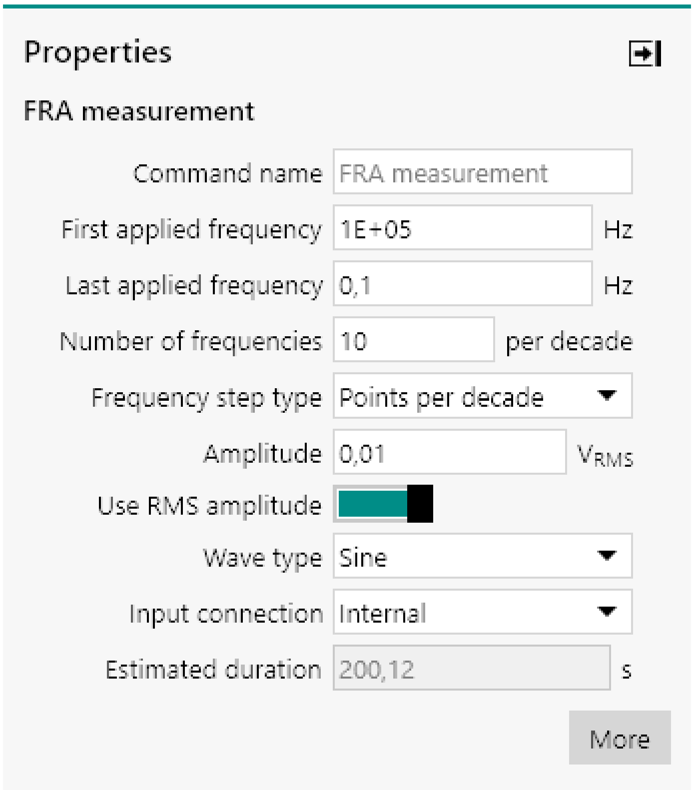


Figure 343 The properties of the FRA measurement command

The following properties are available:

- **Command name:** a user-defined name for the command.
- **First applied frequency:** specifies the first frequency used in the frequency scan, in Hz.
- **Last applied frequency:** specifies the last applied frequency used in the frequency scan, in Hz.
- **Number of frequencies per decade (or number of frequencies):** the number of frequencies per decade used in the frequency scan or the number of frequencies used in the frequency scan. This property depends on the *Frequency step type* property.
- **Frequency step type:** the distribution used to calculate the frequency range. Four distributions are available, selectable using the provided drop-down list:
  - **Linear:** the frequency range is built using a linear distribution.
  - **Square root:** the frequency range is built using a square root distribution.
  - **Logarithmic:** the frequency range is built using a logarithmic distribution.
  - **Points per decade:** the frequency range is built by calculating the number of decades in the range and by adding the specified number of points per calculated decade. This distribution is also logarithmic.

- **Amplitude:** specifies the amplitude to apply during the frequency scan. The units depend on the specified mode and on the *Input connection* property (V, A, or external units).
- **Use RMS amplitude:** a toggle control provided to specify if the amplitude value is the root mean squared (RMS) or top value.
- **Wave type:** specifies the type of signal used during the frequency scan. The choice is provided between the default single sine or the multi sine wave types (Single sine, 5 sines or 15 sines).
- **Input connection:** specifies if the measurement should be carried out internally (through the PGSTAT) or externally, using the external inputs provided on the front panel of the **FRA32M** or **FRA2** module.
- **Estimated duration:** the estimated duration of the command, in s. This property is defined by the frequencies and the acquisition settings and the duration of underlying commands, if applicable.

Table 7 provides an overview of the formulae used to calculate the distributions supported by the FRA measurement command.

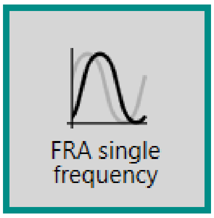
Table 7 The distributions used in the FRA measurement command

Type	Increment, Δ	Distribution
Linear	$\Delta = \frac{\text{End} - \text{Start}}{\text{N} - 1}$	$V_i = \left(\text{Start} + (i-1)\Delta\right)$
Square root	$\Delta = \frac{\sqrt{\text{End}} - \sqrt{\text{Start}}}{\text{N} - 1}$	$V_i = \left(\sqrt{\text{Start}} + (i-1)\Delta\right)^2$
Logarithmic	$\frac{\text{LOG(End)} - \text{LOG(Start)}}{\text{N} - 1}$	$\text{LOG}(V_i) = \left(\text{LOG}(\text{Start}) + (i-1)\Delta\right)$
Points per decade	$\Delta = 10^{-\frac{\text{FLOOR}\left(\left\lvert \text{LOG}\frac{\text{Start}}{\text{End}} \right\rvert N_{\text{Decade}}\right)}{\text{LOG}\frac{\text{Start}}{\text{End}}}}$	$V_i = \left(\text{Start} \cdot \Delta^{(i-1)}\right)$

NOTE

The **FRA measurement** command provides access to additional options, through the More button *(see Chapter 7.6.3, page 292)*.

7.6.2 FRA single frequency

	This command can be used to perform an impedance spectroscopy measurement at a single frequency. This command requires the optional **FRA32M** or **FRA2** module.

CAUTION

The **FRA single frequency** command requires the optional **FRA32M** or **FRA2** module *(see Chapter 16.3.2.13, page 1091)*.

The details of the properties of the **FRA single frequency** command are shown in *Figure 344*:

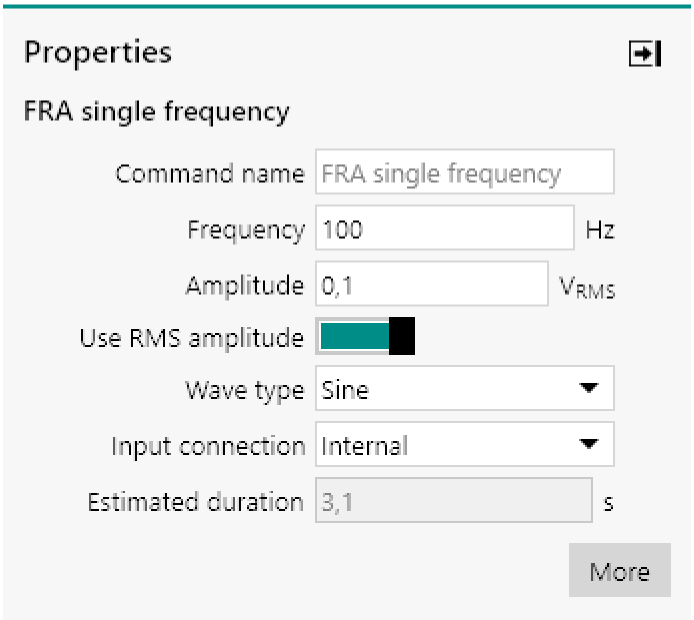


Figure 344 The properties of the FRA single frequency command

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Frequency:** specifies the frequency used in the measurement, in Hz.
- **Amplitude:** specifies the amplitude to apply during the measurement. The units depend on the specified mode and on the *Input connection* property (V, A, or external units).
- **Use RMS amplitude:** a toggle control provided to specify if the amplitude value is the root mean squared (RMS) or top value.
- **Wave type:** specifies the type of signal used during the measurement. The choice is provided between the default single sine or the multi sine wave types (Single sine, 5 sines or 15 sines).
- **Input connection:** specifies if the measurement should be carried out internally (through the PGSTAT) or externally, using the external inputs provided on the front panel of the **FRA32M** or **FRA2** module.
- **Estimated duration:** the estimated duration of the command, in s. This property is defined by the frequency and the acquisition settings and the duration of underlying commands, if applicable.

NOTE

The **FRA single frequency** command provides access to additional options, through the More button *(see Chapter 7.6.3, page 292)*.

7.6.3 Additional properties

Unlike the other measurement command which common additional properties, described in detail in *Chapter 9*, the **FRA measurement** and the **FRA single frequency** commands have specific additional properties, which can be accessed by clicking the More button or by double-clicking the command tile in the procedure editor *(see Figure 345, page 292)*.

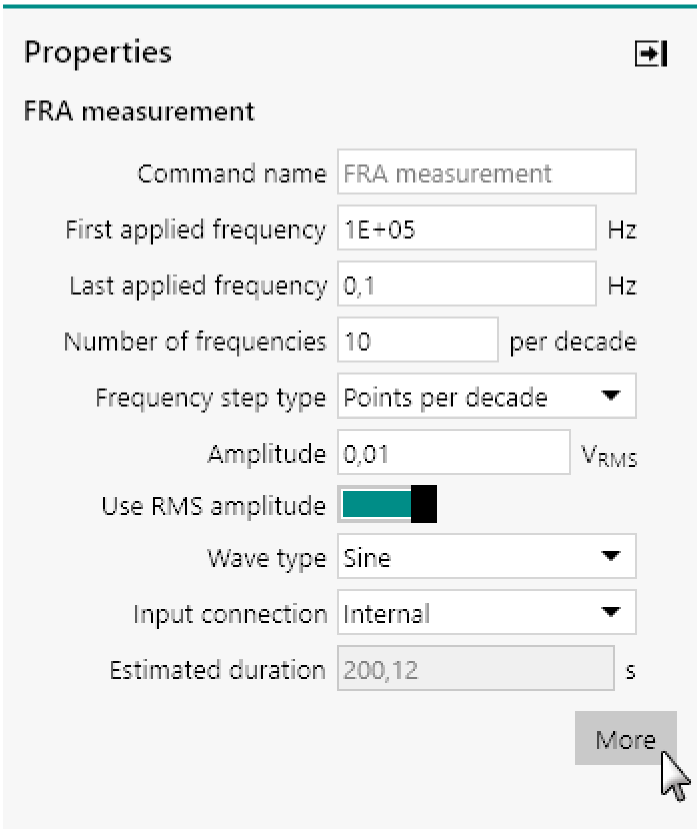


Figure 345 Accessing the additional properties of the FRA measurement command

The additional properties screen will be displayed *(see Figure 346, page 293)*.

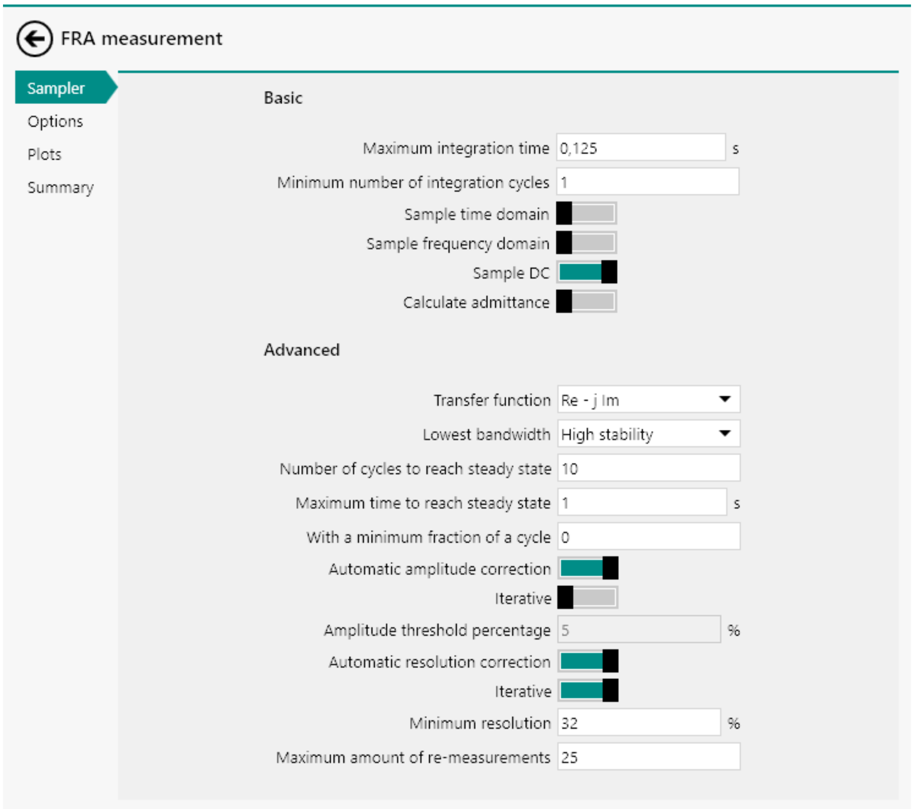


Figure 346 The additional properties

Depending on the properties defined for the **FRA measurement** or the **FRA single frequency** commands, the following sections will be available on the left hand side of the screen:

- **Sampler:** defines the sampling settings for the FRA measurement *(see Chapter 7.6.3.1, page 294)*.
- **Options:** defines the options used in the FRA measurement *(see Chapter 7.6.3.2, page 298)*.
- **Plots:** defines the plots used in the FRA measurement *(see Chapter 7.6.3.2, page 298)*.
- **Summary:** provides an overview of the frequencies used in the FRA measurement. This section is only available for the **FRA measurement** command *(see Chapter 7.6.3.4, page 300)*.
- **External:** defines the transfer function multipliers for measurements using the external inputs of the **FRA32M** or the **FRA2** module *(see Chapter 7.6.3.6, page 309)*.

The **Wave type** property, shown in *Figure 345*, also provides advanced measurement options. *Chapter 7.6.3.5* provides more information on multi sine measurements.

7.6.3.1 Advanced properties - sampler

The **Sampler** section provides sampling settings used by the FRA measurement *(see Figure 347, page 294)*.

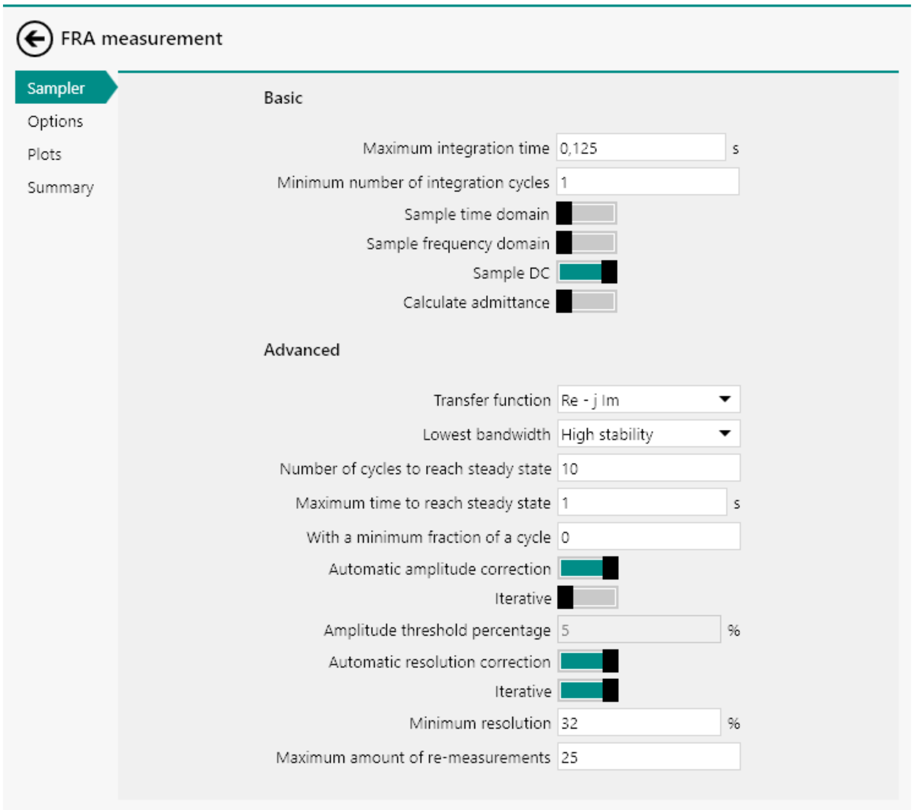


Figure 347 The Sampler settings

The available properties are divided in two sub-sections:

- **Basic:** these are basic acquisition properties that define how FRA measurements are carried out.
- **Advanced:** these are advanced settings, predefined to an optimal value for most measurements. Please read the following section of the manual carefully before adjusting these properties.

The following **Basic** properties are available:

- **Maximum integration time:** the longest time, in seconds, during which the AC response of the cell is recorded for data analysis. Long integration time values increase the duration of the measurement but improve the signal to noise ratio. The default value is 0.125 s.
- **Minimum integration cycles:** defines the minimum cycles of the AC response to record for data analysis (this value overrides the previous property at low frequencies). The minimum value is one cycle and the maximum number is 16. Integrating over a large number of cycles increases the duration of the measurement but improves the signal to noise ratio. This value must be an integer. The default value is 1.

- **Sample time domain:** defines if the time domain information should be sampled during a FRA measurement, using the provided toggle. The time domain information consists of the raw potential and current sine waves. This information can be used to build a Lissajous plot, or to evaluate the signal to noise ratio and to verify the linearity of the cell response. The Time domain, Potential (AC) and Current (AC) and Potential resolution and Current resolution signals are added to the data for each individual frequency when this option is on.
- **Sample frequency domain:** defines if the frequency domain information should be sampled during a FRA measurement, using the provided toggle. The frequency domain information consists of the calculated FFT results obtained from the measured time domain. The frequency domain information can be used to evaluate the measured frequency contributions.
- **Sample DC:** defines if the DC component of the two input signals (Potential and Current, or external signals) information should be sampled during a FRA measurement, using the provided toggle. This option is active by default.
- **Calculate admittance:** specifies if the admittance values must be calculated during the measurement (Y', -Y''), using the provided toggle.

The **Integration time** and **Minimum number of cycles to integrate** define the duration of the data acquisition segment for each frequency. These two properties are competing against one another and depending on the frequency of the applied signal: the measurement duration will be defined by one of these two properties:

- If the *Frequency* is larger than (1/Integration time), the acquisition time will be defined by the **Integration time** property.
- If the *Frequency* is smaller than (1/Integration time), the acquisition time will be defined by the **Minimum number of cycles to integrate** property.

The following **Advanced** properties are available:

- **Transfer function:** this property defines how the impedance is expressed in terms of its real (Re, or Z') and imaginary (Im or Z'') components, using the provided drop-down list. By default, the Re-jIm convention is used. Using this toggle it is possible to acquire impedance data using the alternative convention.

- **Lowest bandwidth:** defines the lowest bandwidth setting used by the Autolab during the measurement (High stability, High speed and Ultra high speed). In High stability, the bandwidth will automatically be set to High stability for frequencies below 10 kHz and to High speed for frequencies below 100 kHz. The Ultra high speed mode is used for frequencies above 100 kHz. When this property is set to High speed, the High stability setting is not used. When this property is set to Ultra high speed, only this bandwidth setting is used. High stability is set by default and is recommended for most measurements.
- **Number of cycles to reach steady state:** defines the number of cycles to apply in between two consecutive frequencies before resuming data acquisition (0 – 30000 cycles). The default value is 10.
- **Maximum time to reach steady state:** defines the maximum amount of time to wait between two consecutive frequencies before resuming data acquisition (this value overrides the previous setting at low frequencies). The maximum value is 30000 s. The default value is 1.
- **With a minimum fraction of a cycle:** defines the minimal fraction of a cycle to wait before the response can be recorded (overrides the previous setting at very low frequencies). This value can be set between 0 and 1. The default value is 0.
- **Automatic amplitude correction:** defines if the automatic amplitude correction algorithm is used during FRA measurements, using the provided toggle. Two automatic amplitude correction modes are available. The selection of the correction algorithm is defined by the position of the **Iterative** toggle:
  - **Normal correction:** this correction mode is enabled when the toggle of the **Iterative** property is set to . In this mode, the amplitude is measured at each frequency during the measurement and the applied amplitude is adjusted for the next frequency point. This means that each frequency value is only adjusted once.
  - **Iterative correction:** this correction mode is enabled when the toggle of the **Iterative** property is set to . In this mode, the amplitude is measured at each frequency and adjusted until the applied amplitude is equal to the expected amplitude within the tolerances specified by the **amplitude threshold percentage property**. This means that each frequency value can be adjusted multiple times before the correct amplitude is measured.
- **Amplitude threshold percentage:** defines the threshold value used to control the applied amplitude. When the **Automatic amplitude correction** property is on and set to *iterative*, the applied amplitude will be considered to be equal to the required amplitude if it fits within the specified **Amplitude threshold percentage**. This value is defined as a percentile value. The default value is 5 %.

- **Automatic resolution correction:** defines if the automatic resolution correction algorithm is used during FRA measurements, using the provided toggle. Two automatic resolution correction modes are available. The selection of the correction algorithm is defined by the position of the **Iterative** toggle:
  - **Normal correction:** this correction mode is enabled when the toggle of the **Iterative** property is set to . In this mode, the resolution is measured at each frequency during the measurement and the gain factors and current ranges are adjusted, if applicable, for the next frequency point. This means that each frequency value is only adjusted once.
  - **Iterative correction:** this correction mode is enabled when the toggle of the **Iterative** property is set to . In this mode, the resolution is measured at each frequency and adjusted until the measured resolution is higher or equal to the specified **minimum resolution**. This means that each frequency value can be adjusted multiple times before the correct resolution is measured.
- **Minimum resolution:** defines the minimum resolution value to reach on both input channels of the impedance analyzer module. When the **Automatic resolution correction** property is on and set to *iterative*, the data will be remeasured until the resolution of both input signals is higher or equal to the specified **Minimum resolution**. This value is defined as a percentile value. The default value is 32 %.
- **Maximum amount of re-measurements:** defines the maximum number of re-measurements allowed if the **Automatic amplitude correction** property is on. The default value is 25.

The **Number of cycles to reach steady state**, **Maximum time to reach steady state**, **With a minimum fraction of a cycle** properties define the duration of the stabilization segment between two consecutive frequencies. These properties are competing against one another, and depending on the frequency of the applied signal, the timing will be defined by one of these three properties:

- If the (Number of cycles to reach steady state/Frequency) is smaller than Maximum time to reach steady state, the settling time will be defined by the **Number of cycles to reach steady state** property.
- If the (Number of cycles to reach steady state/Frequency) is larger than Maximum time to reach steady state, the settling time will be defined by the **Maximum time to reach steady state** property.
- For very low frequencies, if (1/Frequency) is larger than the **Maximum time to reach steady state**, the settling time will be defined by the **With the minimum fraction of a cycle** property.

Figure 348 provides a schematic overview of the five timing properties. At low frequency, the integration time is overruled by the minimum number of cycles to integrate. At low frequency, the number of cycles to reach steady state property is overruled by the maximum time to reach steady state property or the by the with a minimum fraction of cycle property, in a similar way.

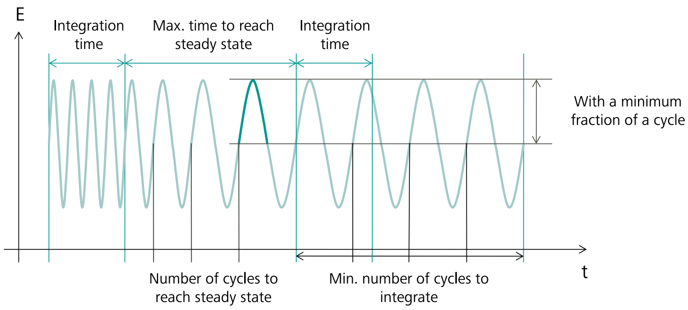


Figure 348 Overview of the timing properties (Integration time: 2 s, Minimum integration cycles: 4, Number of cycles to wait for steady state: 4, Maximum time to reach steady state: 4 s, Minimal cycle fraction of to wait for steady state: 0.5)

7.6.3.2 Advanced properties - options

The **Options** section provides automatic current ranging used by the FRA measurement *(see Figure 363, page 310)*.

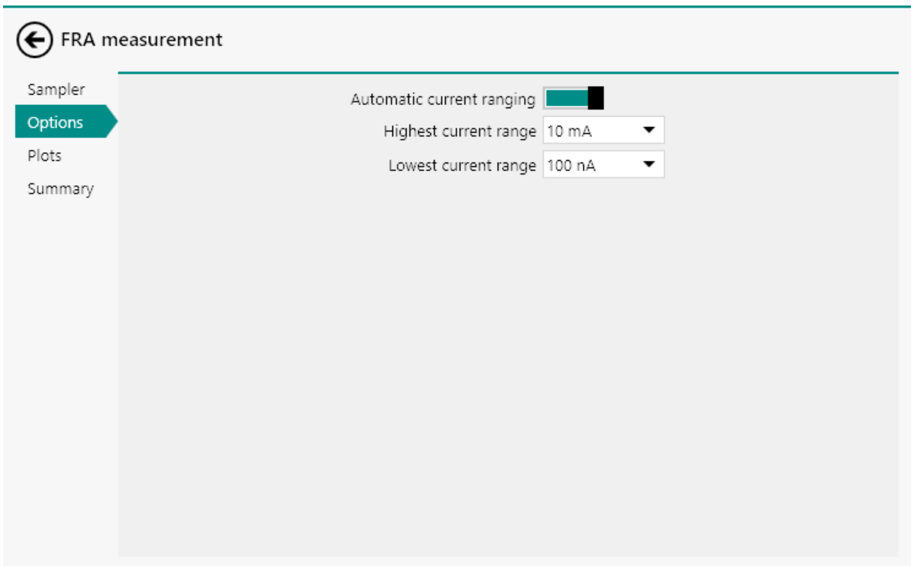


Figure 349 The Options settings

The following properties are available:

- **Automatic current ranging:** sets the automatic current ranging option on or off, using the provided toggle.
- **Highest current range:** defines the highest allowed current range, using the provided drop-down list.
- **Lowest current range:** defines the lowest allowed current range, using the provided drop-down list.

In FRA measurements, using the **FRA measurement** or the **FRA single frequency** commands, it is possible to use the **Booster10A** or **Booster20A** current range in the automatic current ranging option *(see Figure 350, page 299)*.

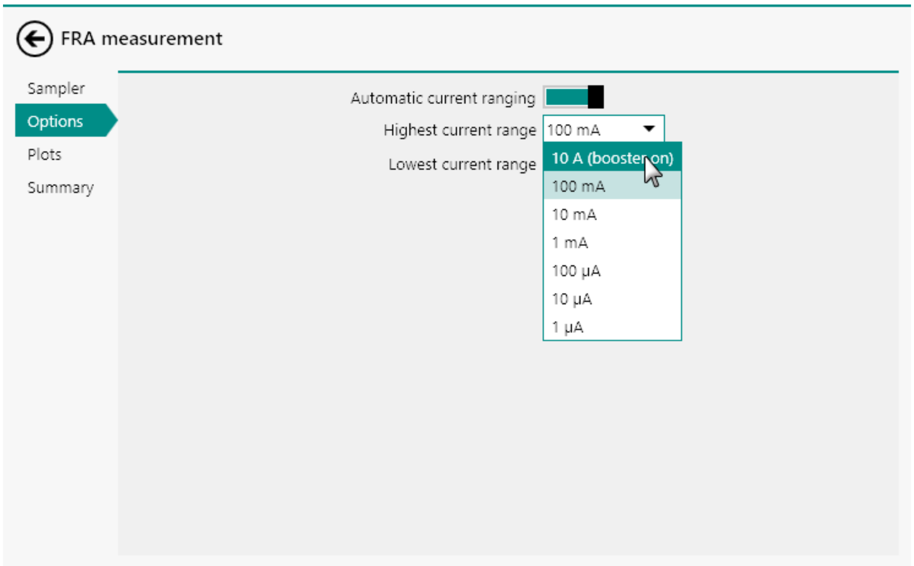


Figure 350 The current range of the Booster10A and Booster20A is available in the Automatic current ranging option

7.6.3.3 Advanced properties - plots

The **Plots** section provides plot settings used by the FRA measurement *(see Figure 351, page 299)*.

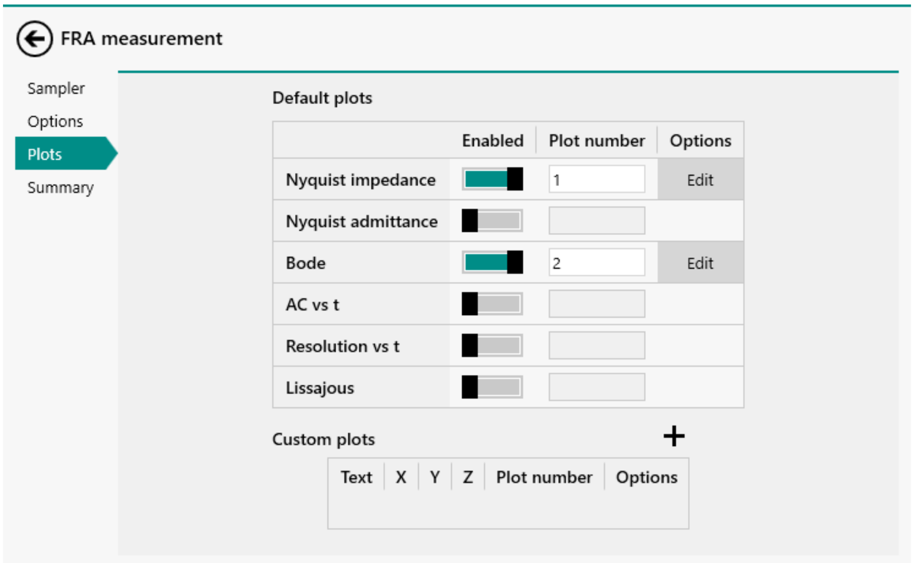


Figure 351 The Plots settings

NOTE

The available plots depend on the properties specified in the **Sampler** section.

The following plots are available:

- **Nyquist impedance:** plots the measured –Z″ values versus the measured Z′ values, using isometric axes.
- **Nyquist admittance:** plots the calculated –Y″ values versus the calculated Y′ values, using isometric axes.
- **Bode:** plots the phase (in opposed values) and the logarithm of the measured impedance (Z), versus the logarithm of the frequency.
- **AC vs t:** plots the raw sinewave amplitudes for the potential and the current signals versus the time.
- **Resolution vs t:** plots the instrumental resolution for the measured potential and current signals versus the time.
- **Lissajous:** plots the raw AC current versus the raw AC potential.

7.6.3.4 Advanced properties - summary

The **Summary** section provides a complete summary of the frequency scan parameters defined in the **FRA measurement** command *(see Figure 352, page 300)*.

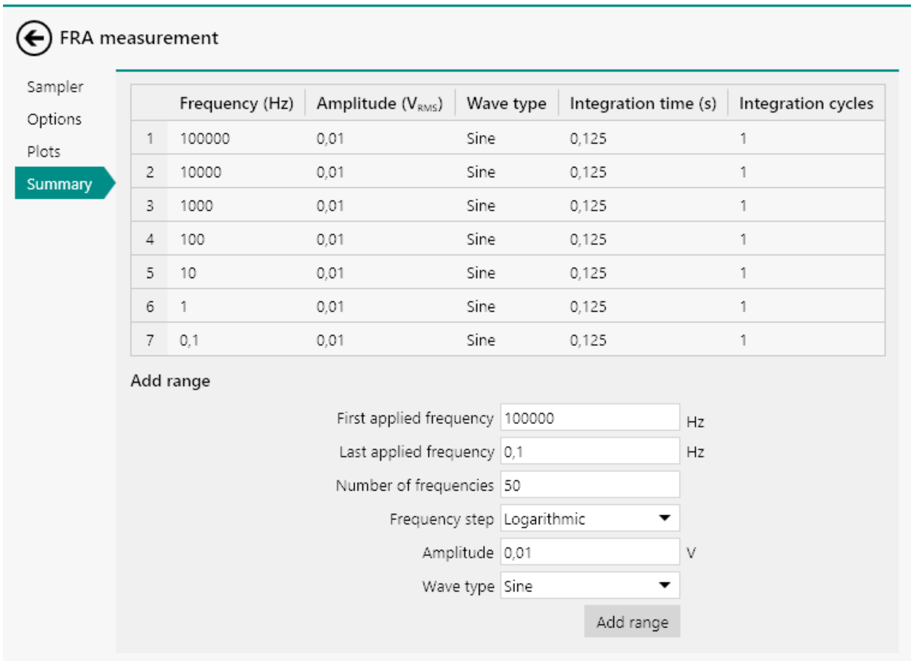


Figure 352 The Summary section of the FRA measurement command

NOTE

This section is only available for the **FRA measurement** command.

Each frequency in the scan is displayed in the table, along with the amplitude values, wave type, minimum integration time and maximum number of cycles to integrate *(see Figure 352, page 300)*.

A tooltip displays information about each individual frequency in the scan *(see Figure 353, page 301)*.

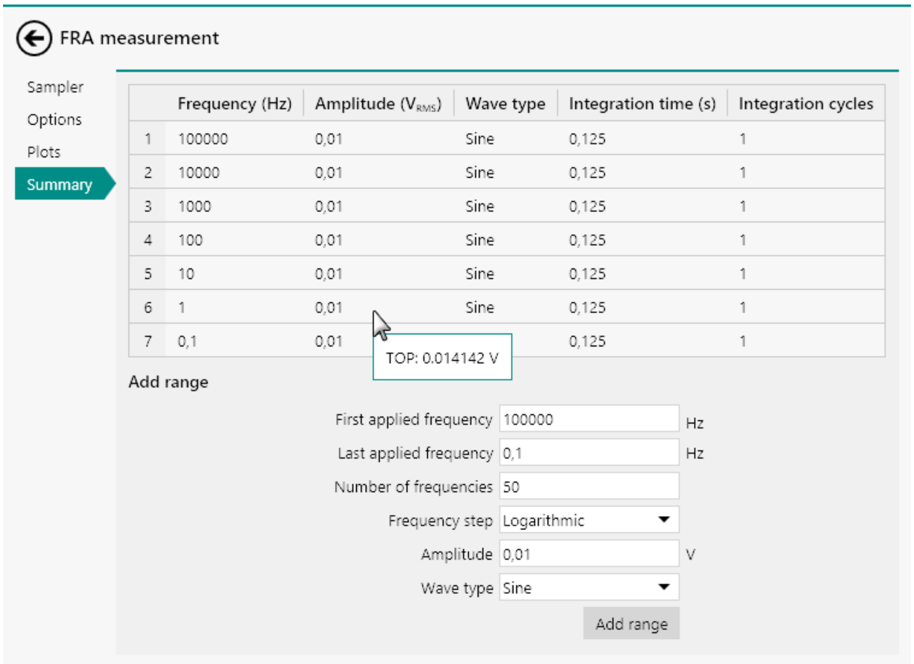


Figure 353 Information on each individual frequency is displayed in a tooltip

The summary section can be used to fine tune the frequency scan in three different ways:

1. The properties of one or more frequencies in the table can be adjusted manually *(see Chapter 7.6.3.4.1, page 302)*.
2. Additional frequencies can be added manually to the table *(see Chapter 7.6.3.4.1, page 302)*.
3. The frequencies can be sorted ascending or descending *(see Chapter 7.6.3.4.3, page 306)*.

7.6.3.4.1 Manual modification of the properties

If needed, one of more properties can be overruled in the summary table shown in the **Summary** section. Double click the field to be edited in the table and modify the value *(see Figure 354, page 302)*.

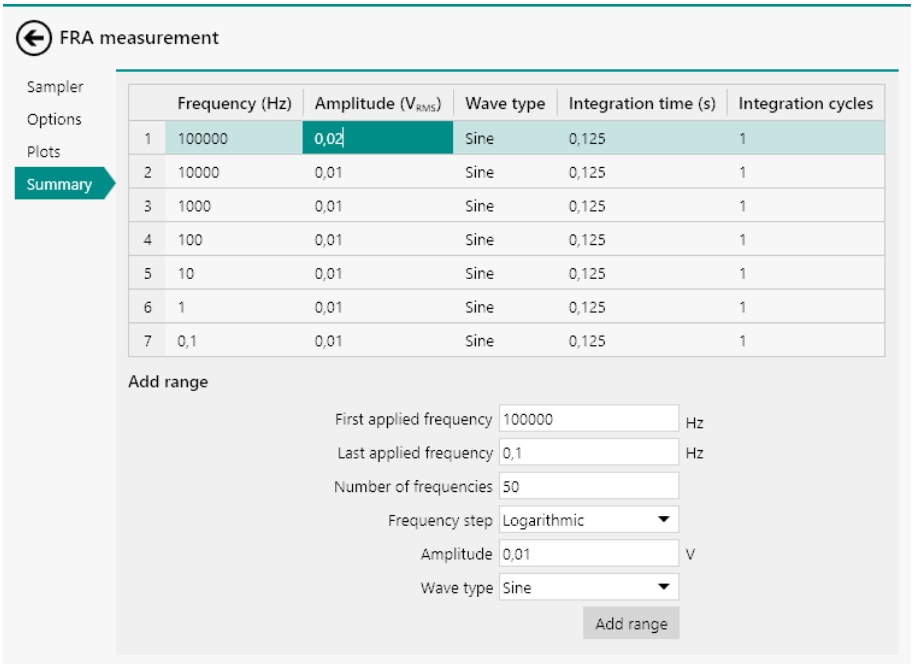


Figure 354 Modifying individual values in the Summary section

Press the **[Enter]** key to validate the new value. The new value will be used instead of the initial value specified in the **FRA measurement** command properties *(see Figure 355, page 303)*.

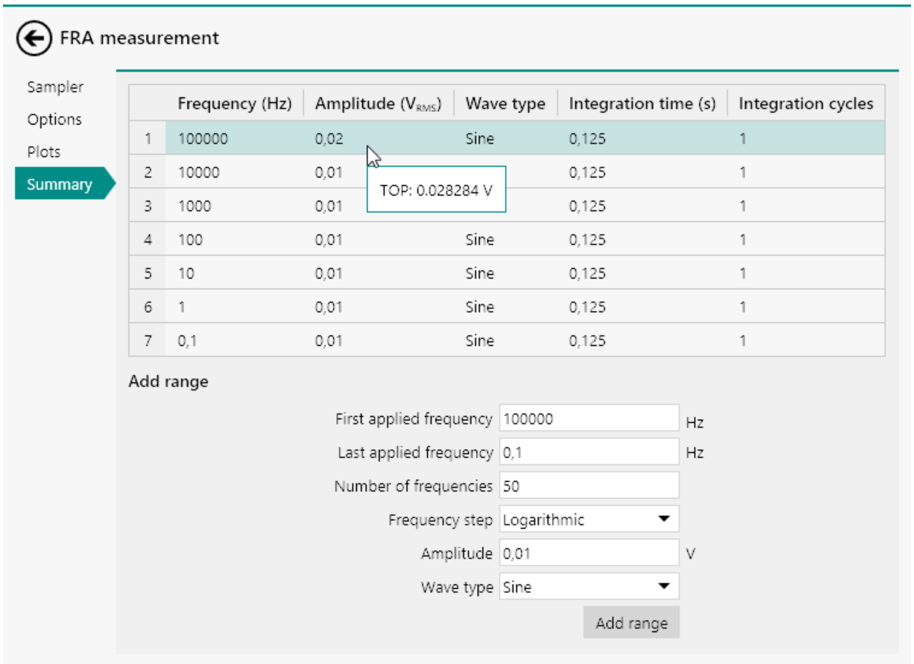


Figure 355 The new value is updated in the FRA editor Summary section

7.6.3.4.2 Adding frequencies to the table

If needed, additional frequencies can be added to the table presented in the **Summary** section, by editing the properties located in the **Add range** sub-panel and clicking the Add range button *(see Figure 356, page 303)*.

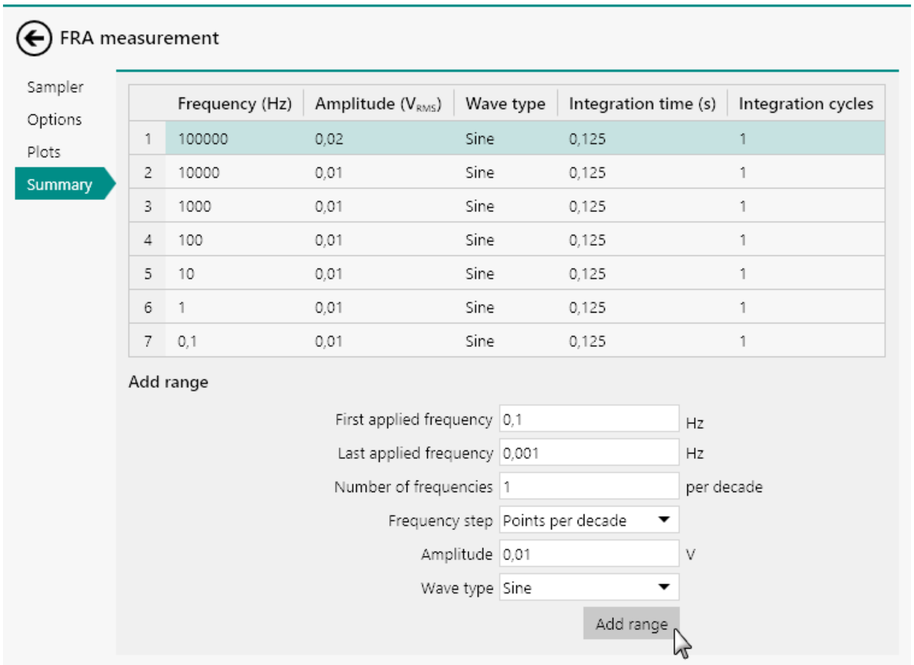
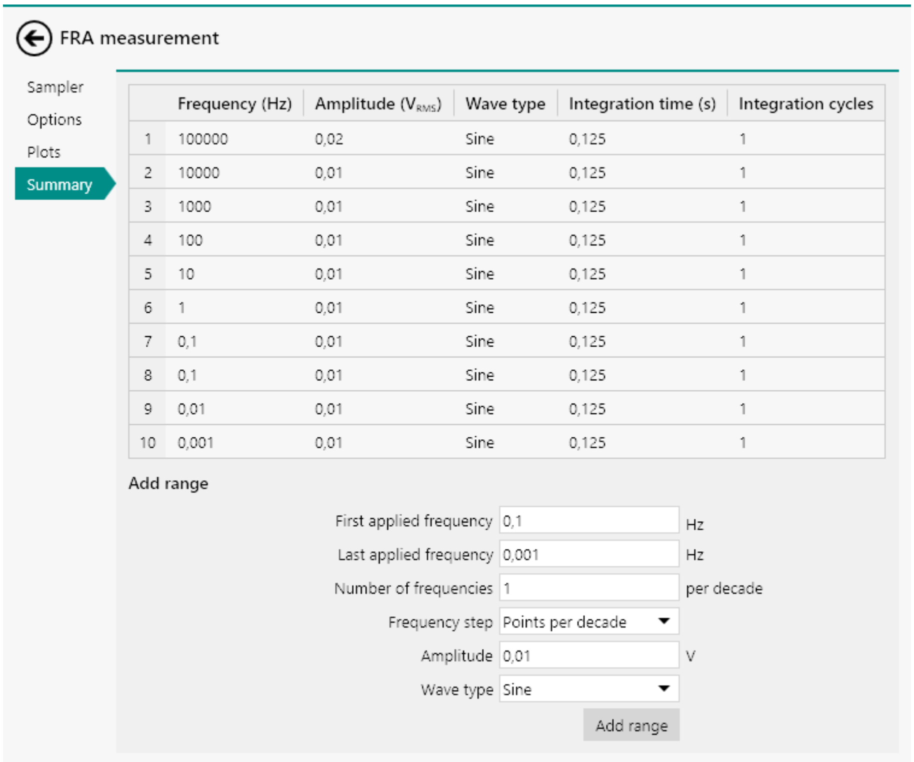


Figure 356 Adding additional frequencies to the range

The **Summary** section will be updated and the new frequencies will be added to the table *(see Figure 357, page 304)*.



	Frequency (Hz)	Amplitude (V_{RMS})	Wave type	Integration time (s)	Integration cycles
1	100000	0,02	Sine	0,125	1
2	10000	0,01	Sine	0,125	1
3	1000	0,01	Sine	0,125	1
4	100	0,01	Sine	0,125	1
5	10	0,01	Sine	0,125	1
6	1	0,01	Sine	0,125	1
7	0,1	0,01	Sine	0,125	1
8	0,1	0,01	Sine	0,125	1
9	0,01	0,01	Sine	0,125	1
10	0,001	0,01	Sine	0,125	1

Figure 357 The updated summary

It is also possible to delete frequencies from the table by selecting one or more rows of the table and pressing the **[Delete]** button *(see Figure 358, page 305)*.

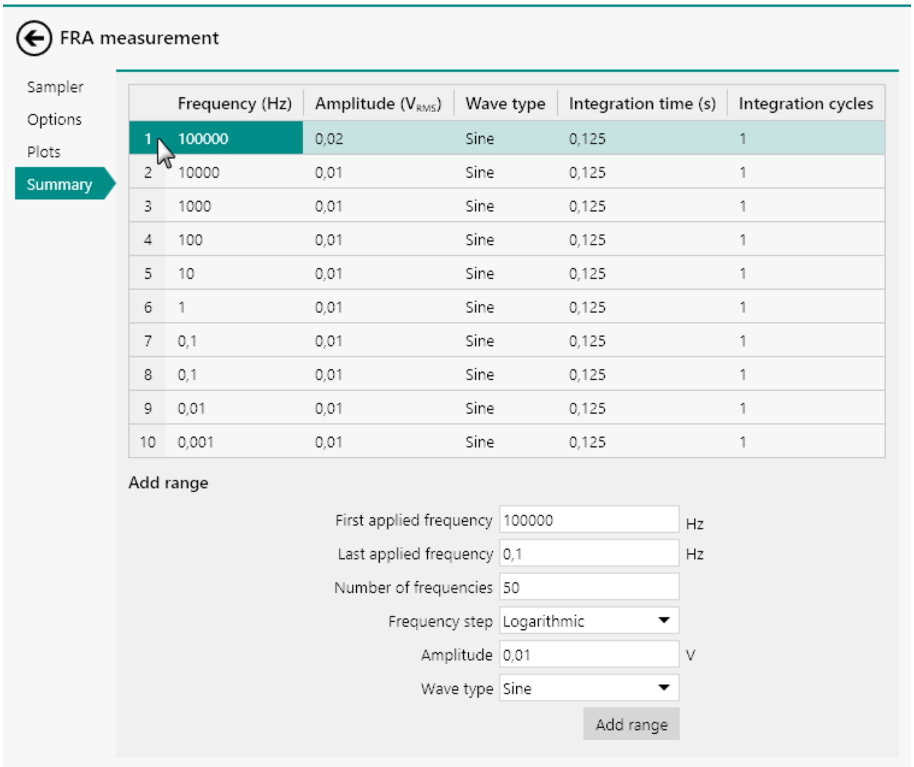


Figure 358 It is possible to delete frequencies from the table

The selected frequencies will be deleted from the table *(see Figure 359, page 306)*.

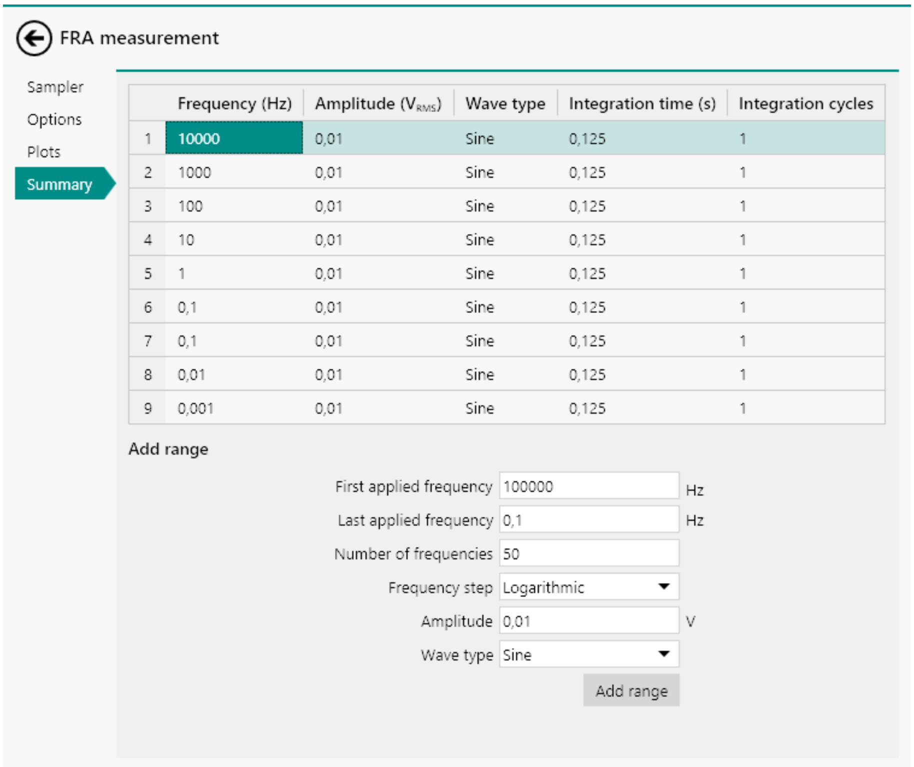


Figure 359 The selected frequencies are removed from the table

7.6.3.4.3 Sorting the table

It is possible to sort the frequencies ascending or descending in the **Summary** section by clicking the Frequency column header. A small arrow will be displayed in the column header indicating the sorting direction. Clicking the header again will cycle between ascending and descending *(see Figure 360, page 307)*.

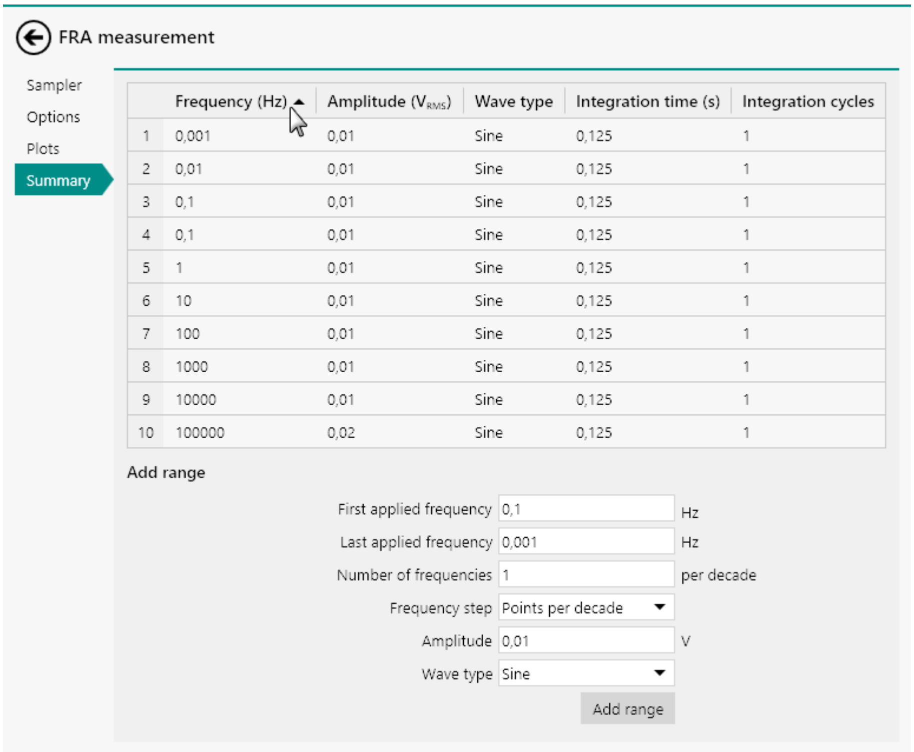


Figure 360 Sorting the frequencies in ascending or descending direction

NOTE

It is only possible to the frequency values in the table.

7.6.3.5 Multi sine measurements

For very low frequency measurements, the multi sine option can be used to increase the acquisition speed. This option allows you to create a frequency scan in which low frequency single sine signals are replaced by a linear combination of five or fifteen frequencies. Each multi sine signal generates a number of data points equal to the number of components in the linear combination.

The multi sine option is **only** available for low frequencies. The maximum frequencies for multi sine measurements are listed in *Table 8*.

Table 8 The frequency limit for multi sine measurement for the FRA32M and FRA2 modules

Module	5 sine frequency limit	15 sine frequency limit
FRA32M	320 Hz	32 Hz
FRA2	3472 Hz	315.2 Hz

To create a multi sine frequency scan, the **Wave type** property can be set in the **Properties** panel *(see Figure 361, page 308)*.

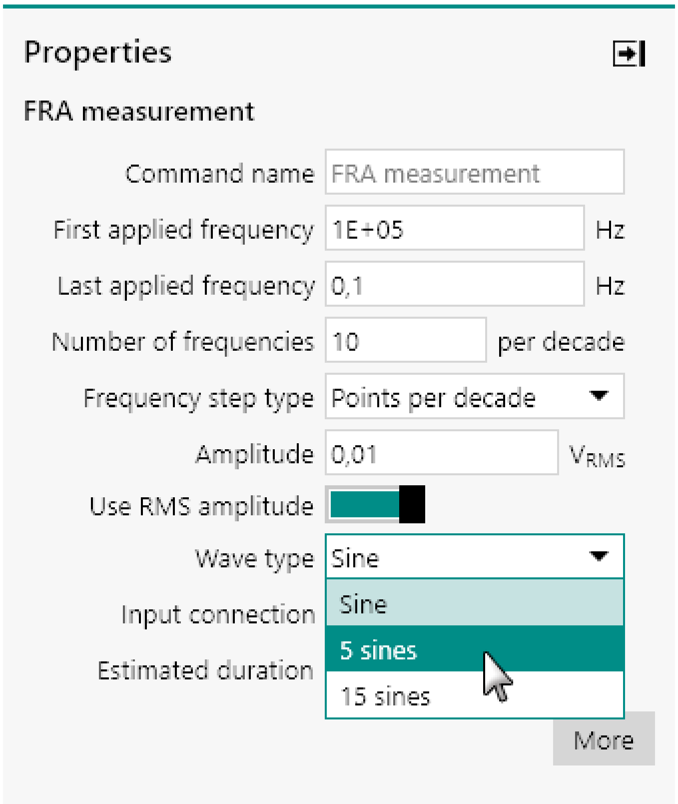


Figure 361 The Wave type property can be used to create a multi sine measurement

Depending on the available hardware (**FRA32M** or **FRA2**), the frequency scan will be generated and the multi sine signals will be used for the low frequencies in the range. The **Summary** section displays the wave type in the table *(see Figure 362, page 309)*.

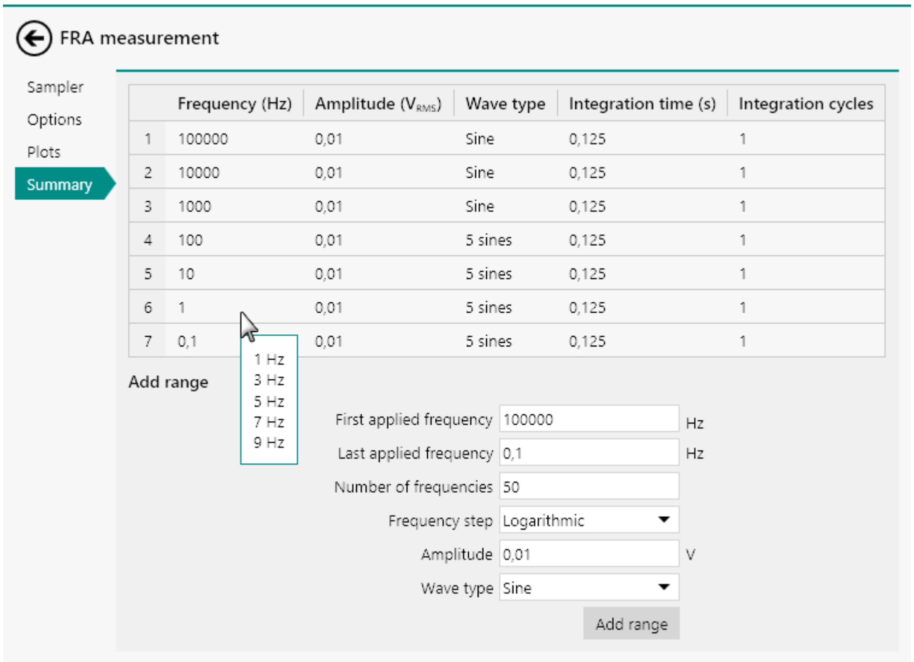


Figure 362 The Summary section displays the wave type

The individual components of a multi sine signal are displayed in a tooltip *(see Figure 362, page 309)*. The frequency displayed in the summary section is the lowest of the value of the linear combination. The values displayed in the tooltip correspond to the higher harmonics in the linear combination.

7.6.3.6 Advanced properties - external

Electrochemical impedance spectroscopy measurements assume that the transfer function of interest is Z, which corresponds to the ratio of the AC potential over the AC current. Electrochemical impedance is in fact a specific form of a more general definition of impedance. The AC perturbation can be of a wide range of parameters, such as the rotation rate of a rotating disc electrode (EHD), or the light intensity of a light source (IMVS/IMPS), etc.

Generalized impedance measurements are possible in NOVA through the SMB connectors located on the front panel of the FRA32M module or through the BNC connectors on the front panel of the FRA2 module. To measure external sine waves on both inputs of the FRA module and to measure the generalized impedance, the **Input connection** property of the **FRA measurement** or the **FRA single frequency** command can be set to External.

The **External** section provides settings used when the **Input connection** property is set to External in the **FRA measurement** or the **FRA single frequency** commands *(see Figure 363, page 310)*.

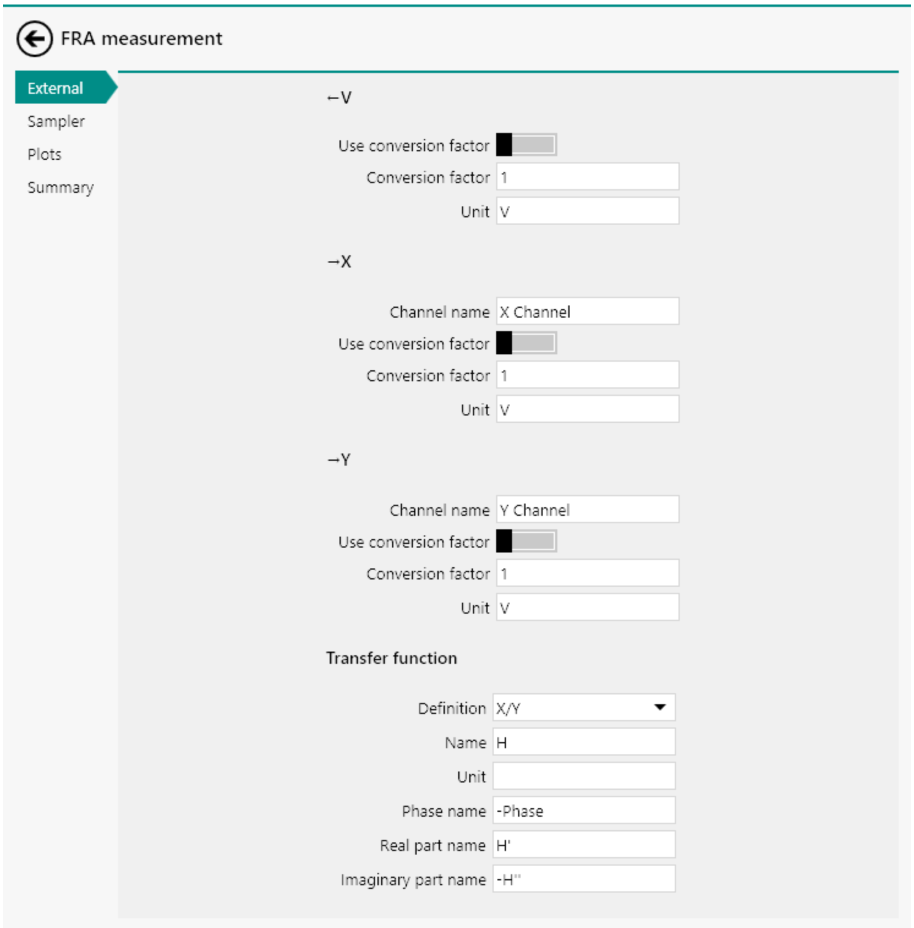
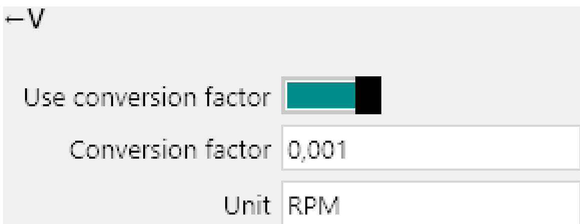


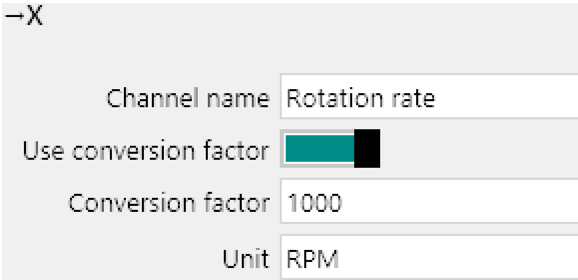
Figure 363 The External settings

The settings shown in the **External** section can be used to specify how external impedance measurements are carried out. Four sub-sections are available in the External section:

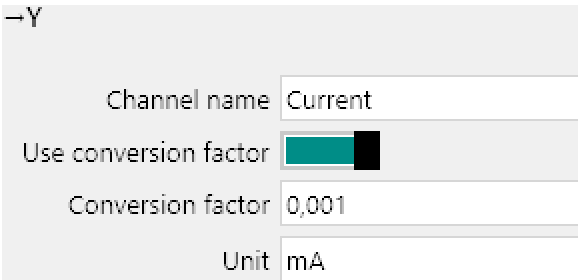
- ← **V**: these settings define the conversion from generated AC voltage to the AC signal used for external impedance measurements. The following properties are available:
  - **Use conversion factor:** defines if a conversion factor must be used, using the provided ▮ toggle.
  - **Conversion factor:** defines the conversion factor, if applicable.
  - **Unit:** defines the units of the external AC signal, if applicable.



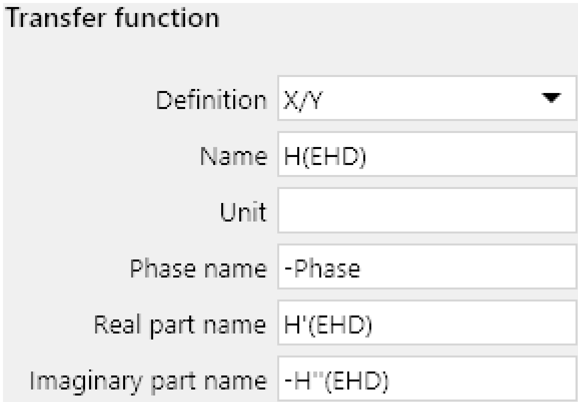
- **→ X:** these settings define the conversion used for the X input of the impedance analyzer module. The following properties are available:
  - **Channel name:** defines the name of the input signal.
  - **Use conversion factor:** defines if a conversion factor must be used, using the toggle.
  - **Conversion factor:** defines the conversion factor used for the input signal.
  - **Unit:** defines the units of the input signal.



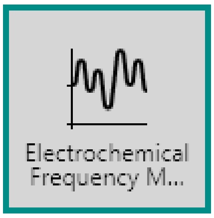
- **→ Y:** these settings define the conversion used for the X input of the impedance analyzer module. The following properties are available:
  - **Channel name:** defines the name of the input signal.
  - **Use conversion factor:** defines if a conversion factor must be used, using the toggle.
  - **Conversion factor:** defines the conversion factor used for the input signal.
  - **Unit:** defines the units of the input signal.



- **Transfer function:** these settings define how the external transfer function is calculated. The following properties are available:
  - **Definition:** defines how the transfer function is calculated, using the provided drop-down list (X/Y or Y/X).
  - **Name:** defines the name of the transfer function.
  - **Unit:** defines the unit of the transfer function.
  - **Phase name:** defines the name of the phase angle.
  - **Real part name:** defines the name of the real part of the transfer function.
  - **Imaginary part name:** defines the name of the imaginary part to the transfer function.



7.6.4 Electrochemical Frequency Modulation

	This command can be used to perform an electrochemical frequency modulation (EFM) measurement. This command requires the optional **FRA32M** module.

CAUTION

The **Electrochemical Frequency Modulation** command requires the optional **FRA32M** module *(see Chapter 16.3.2.13, page 1091)*.

CAUTION

The **Electrochemical Frequency Modulation** command can only be used in **potentiostatic**. Automatic current ranging and other options like cutoffs and counters are not supported by this command.

The details of the properties of the **Electrochemical Frequency Modulation** command are shown in *Figure 364*:

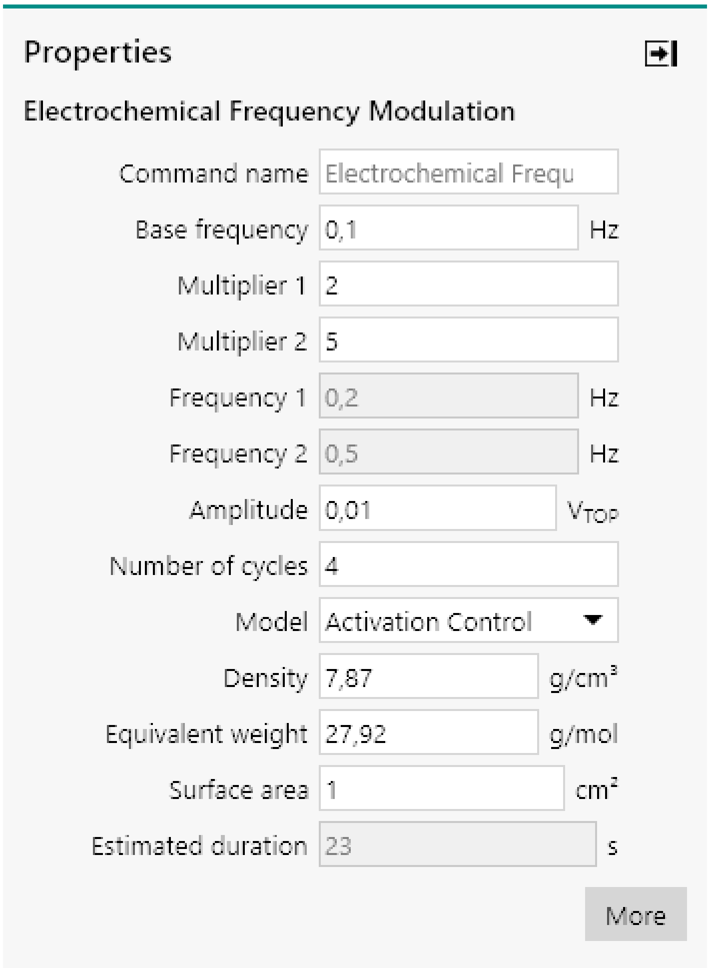


Figure 364 The properties of the Electrochemical Frequency Modulation command

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Base frequency:** specifies the base frequency used in the measurement, in Hz.
- **Multiplier 1:** specifies the value of the base frequency multiplier 1 (default 2).
- **Multiplier 2:** specifies the value of the base frequency multiplier 2 (default 5).
- **Amplitude:** specifies the amplitude to apply during the measurement, in V, as top amplitude.
- **Number of cycles:** specifies the number of cycles to apply during the measurement.

- **Model:** specifies the model used to analyze the data, using the provided drop-down list. Three different models are provided:
  - **Activation control:** a model that provides a general purpose model of a corroding system.
  - **Diffusion control:** a model that can be used for systems for which no cathodic current is observed.
  - **Passivating:** a model that can be used for systems for which no anodic current is observed.
- **Density:** specifies the density of the sample in g/cm^3.
- **Equivalent weight:** defines the equivalent weight of the sample in g/mol of exchanged electrons.
- **Surface area:** defines the area of the sample, in cm^2.
- **Estimated duration:** the estimated duration of the command, in s. This property is defined by the base frequency and the acquisition settings and the duration of underlying commands, if applicable.

NOTE

The **Electrochemical Frequency Modulation** command provides access to additional options, through the More button *(see Chapter 7.6.3, page 292)*.

During a measurement, the **Electrochemical Frequency Modulation** command will apply a multi sine perturbation in potentiostatic conditions using a signal containing two decoupled frequencies. These two frequencies are obtained using the specified **Base frequency**, **Multiplier 1** and **Multiplier 2**. The two frequencies used in the measurement are reported in the **Properties** panel (**Frequency 1** and **Frequency 2**).

CAUTION

Multiplier 1 and **Multiplier 2** must not be integral multiples one another.

The current resulting from the application of the two sinewaves is recorded and converted from the time domain to the frequency domain. The second and third harmonic of the base frequencies are determined as well as the intermodulated frequencies. *Table 7.6.4* provides a summary of the frequencies analyzed in the measured signal.

Table 9 The frequencies used by the Electrochemical Frequency Modulation command

Current spectrum component	Determined from
$i_{2\omega_1}$	Second harmonic of ω_1 (or ω_2)
$i_{3\omega_1}$	Third harmonic of ω_1 (or ω_2)
i_{ω_1,ω_2}	ω_1 (or ω_2)
$i_{2\omega_2\pm\omega_1}$	Second order of ω_1 or ω_2 intermodulated with ω_2 or ω_1
$i^2_{\omega_2\pm\omega_1}$	ω_1 or ω_2 intermodulated with ω_2 or ω_1
$i^2_{\omega_1,\omega_2}$	ω_1 (or ω_2)

Using the measured values, the **Electrochemical Frequency Modulation** command can calculate the following corrosion indicators:

- **b_a:** the anodic Tafel slope, in V/dec.
- **b_c:** the cathodic Tafel slope, in V/dec.
- **i_{corr}:** the corrosion current, in A.

These values are obtained using mathematical expression that depend on the selected model.

1. **Activation control model**

$$i_{corr} = \frac{i^2_{\omega_1,\omega_2}}{2\sqrt{8i_{\omega_1,\omega_2} \cdot i_{2\omega_2\pm\omega_1} - 3i^2_{\omega_2\pm\omega_1}}}$$

$$b_a = 2,303\frac{i_{\omega_1,\omega_2}U_0}{i_{\omega_2\pm\omega_1} + \sqrt{8i_{\omega_1,\omega_2} \cdot i_{2\omega_2\pm\omega_1} - 3i^2_{\omega_2\pm\omega_1}}}$$

$$b_c = 2,303\frac{i_{\omega_1,\omega_2}U_0}{-i_{\omega_2\pm\omega_1} + \sqrt{8i_{\omega_1,\omega_2} \cdot i_{2\omega_2\pm\omega_1} - 3i^2_{\omega_2\pm\omega_1}}}$$

2. **Diffusion control model**

$$i_{corr} = \frac{i^2_{\omega_1,\omega_2}}{2i_{\omega_2\pm\omega_1}}$$

$$b_a = 2,303\frac{i_{\omega_1,\omega_2}U_0}{2i_{\omega_2\pm\omega_1}}$$

$$b_c = \infty$$

3. **Passivating model**

$$i_{corr} = \frac{i^2_{\omega_1,\omega_2}}{2i_{\omega_2\pm\omega_1}}$$

$$b_a = \infty$$

$$b_c = 2,303\frac{i_{\omega_1,\omega_2}U_0}{2i_{\omega_2\pm\omega_1}}$$

U_0 is the applied potential amplitude, in V, in all three models.

Using the calculated values, additional corrosion indicators can be calculated:

- **j_{corr}:** the corrosion current density, in A/cm².
- **Corrosion rate:** the corrosion rate, in mm of material/year.
- **Polarization resistance:** the converted polarization resistance, in Ohm.

Finally, two causality factors can be calculated, regardless of the model:

- **Causality factor (2):** a second order consistency factor. This value should be as close as possible to 2.
- **Causality factor (3):** a third order consistency factor. This value should be as close as possible to 3.

These factors are calculated according to:

$$CF_2 = \frac{i_{\omega_2 \pm \omega_1}}{i_{2\omega_1}}$$

$$CF_3 = \frac{i_{2\omega_2 \pm \omega_1}}{i_{3\omega_1}}$$

NOTE

The values of b_a and b_c are reported in absolute value.

NOTE

The **Electrochemical Frequency Modulation** technique is based on *Electrochemical Frequency Modulation: A New Electrochemical Technique for Online Corrosion Monitoring*, R.W. Bosch, J. Hubrecht, W. F. Bogaerts, and B.C. Syrett, Corrosion, 57 (2001).

7.7 Data handling commands

Data handling commands can be used to process measured data. A full description of the commands provided in this group is provided in this chapter.

The available commands are represented by a shortcut icon *(see Figure 365, page 317)*.

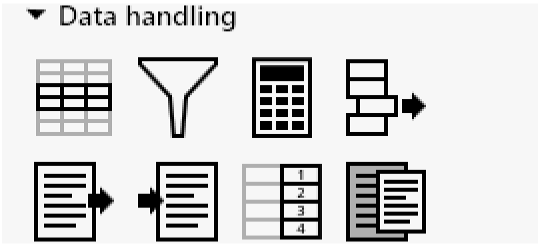
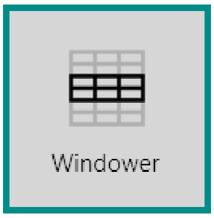


Figure 365 The Data handling commands

The following commands are available:

- **Windower:** a command which can be used to create a subset from measured or calculated signals, using a condition on a source signal *(see Chapter 7.7.1, page 317)*.
- **Build signal:** a command which can be used to create signals based on measured or calculated signals or procedure properties, to be used for further data analysis *(see Chapter 7.7.2, page 322)*.
- **Calculate signal:** a command which can be used to calculate signals based on measured or calculated signals or procedure properties *(see Chapter 7.7.3, page 330)*.
- **Get item:** a command which can be used to get a specific value of a signal *(see Chapter 7.7.4, page 343)*.
- **Import data:** a command which can be used to import data from an external GPES, FRA or ASCII file *(see Chapter 7.7.5, page 343)*.
- **Export data:** a command which be used to export data to an external ASCII or ZView file *(see Chapter 7.7.6, page 347)*.
- **Generate index:** this command can be used to index the source data *(see Chapter 7.7.7, page 349)*.
- **Shrink data:** this command can be used to shrink the source data *(see Chapter 7.7.8, page 350)*.

7.7.1 Windower

	This command can be used to extract a subset of signal values from measured or calculated signals, to be used in the data analysis.

The details of the properties of the **Windower** command are shown in *Figure 366*:

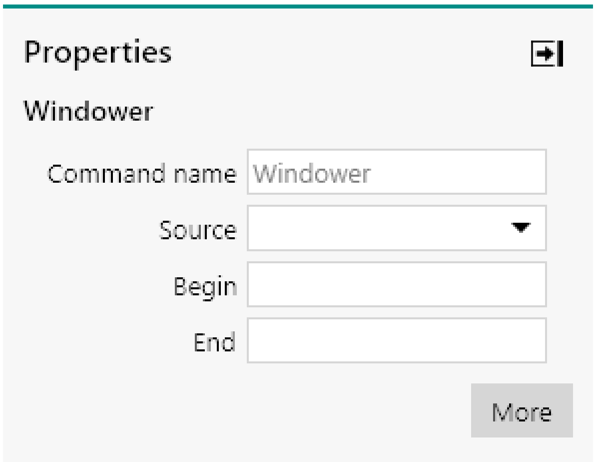


Figure 366 The properties of the Windower command

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Source:** this is the source signal used to window the data. The source signal is one of the available signals available in the data. Only one source signal can be selected.
- **Begin** the start value used by the windower for the specified source signal.
- **End:** the end value used by the windower for the specified source signal.

NOTE

The **Begin** and **End** values can be fine-tuned in the additional properties panel, available using the More button.

NOTE

The **Windower** command can not be used stand alone. This command is designed to work in conjunction with another command providing the source signal used by the **Windower** command. The **Windower** command can be *stacked* onto the command providing the source data *(see Chapter 10.12, page 653)* or *linked* to the command providing the source data *(see Chapter 10.13, page 657)*.

Additional properties are available by clicking the More in the **Properties** panel. A new screen will be displayed, as shown in *Figure 367*.

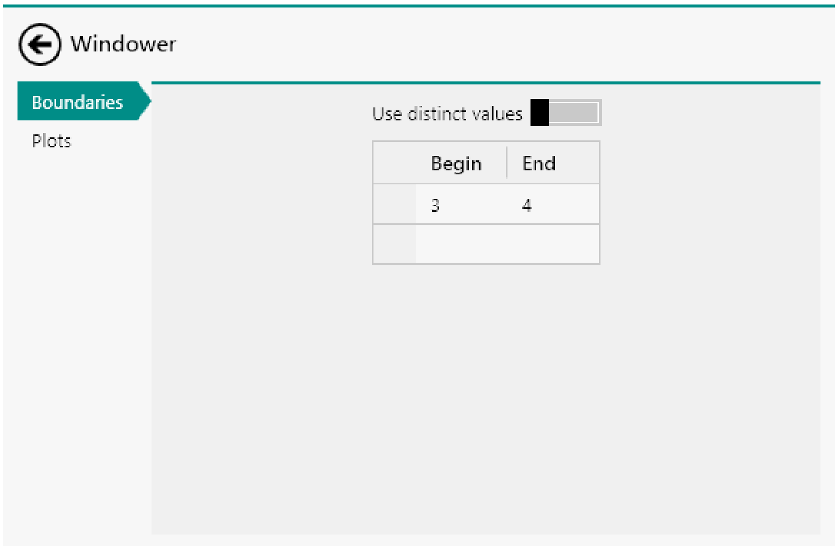


Figure 367 Additional properties are available for the Windower command

Depending on if the **Windower** command is added to a procedure or to data, the following additional settings are available:

- **Use distinct values:** a toggle that can be used to switch the Boundaries from table view to list view. This toggle is only shown when the **Windower** command is used on **data**.
- **Boundaries:** the boundaries of the source signal used by **Windower** command. These boundaries can be edited in the provided table, or in the alternate list view when the **Windower** command is used on **data**.
- **Plots:** the Plot editor can be used to specify how the windowed data should be displayed. The use of the plot editor is explained in *Chapter 9.5*.

The boundaries can be fine-tuned in the additional properties panel of the **Windower** command. In this panel more than one row can be used in order to add more boundaries to the table shown in the panel *Figure 368*.

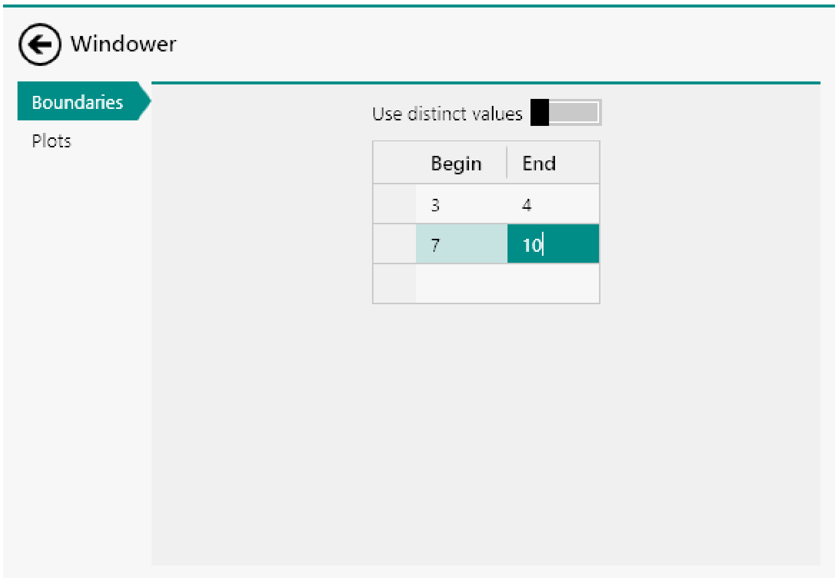


Figure 368 Adding additional boundaries to the Windower command

When the **Windower** command is used on data, the *Use distinct values* toggle is available. When this toggle is activated, the table is replaced by a list of checkboxes of the available value for the selected source signal *(see Figure 369, page 320)*.

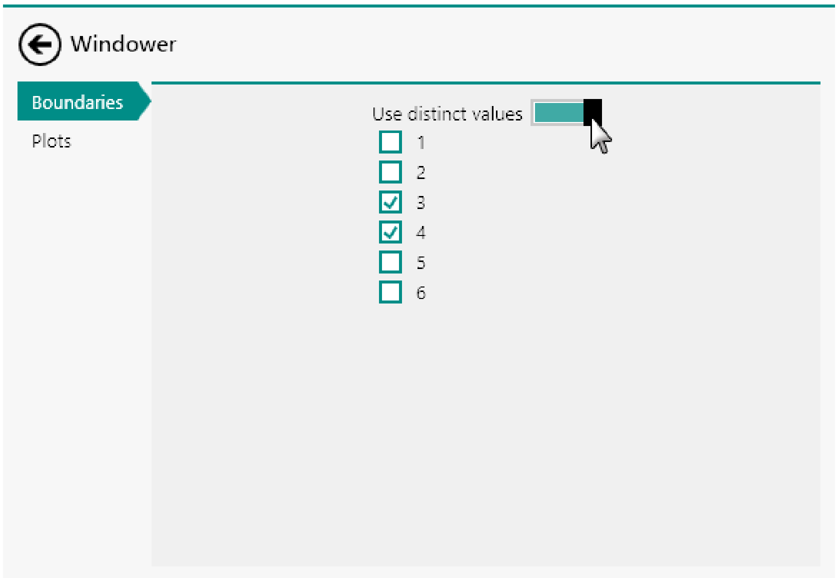


Figure 369 Using the distinct values selection

NOTE

Only the possible values for the selected source signal are shown in the list.

Checking or unchecking the checkboxes adds or remove the matching value for the selected source signal. The values already defined in the table are translated to selected checkboxes *(see Figure 370, page 321)*.

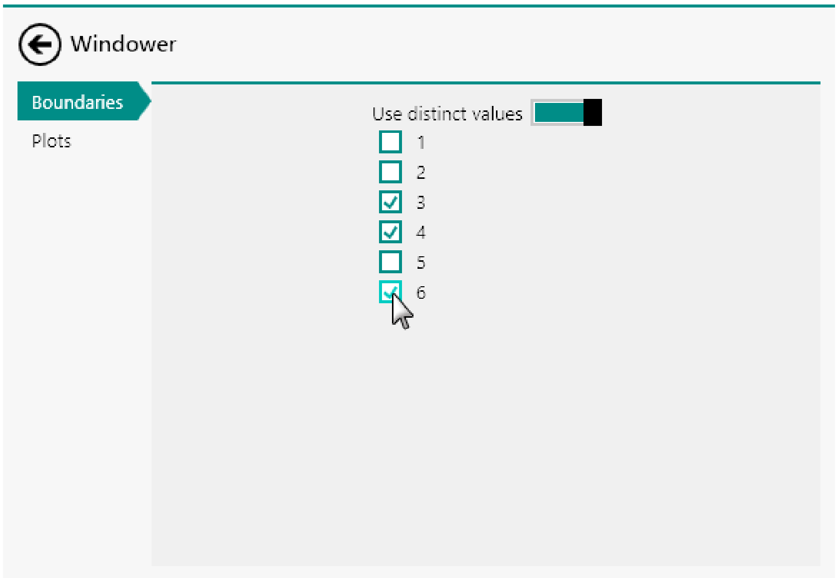


Figure 370 Adding additional values to the boundaries editor

Switching back to the table will translate the selected checkboxes to rows in the table *(see Figure 371, page 322)*.

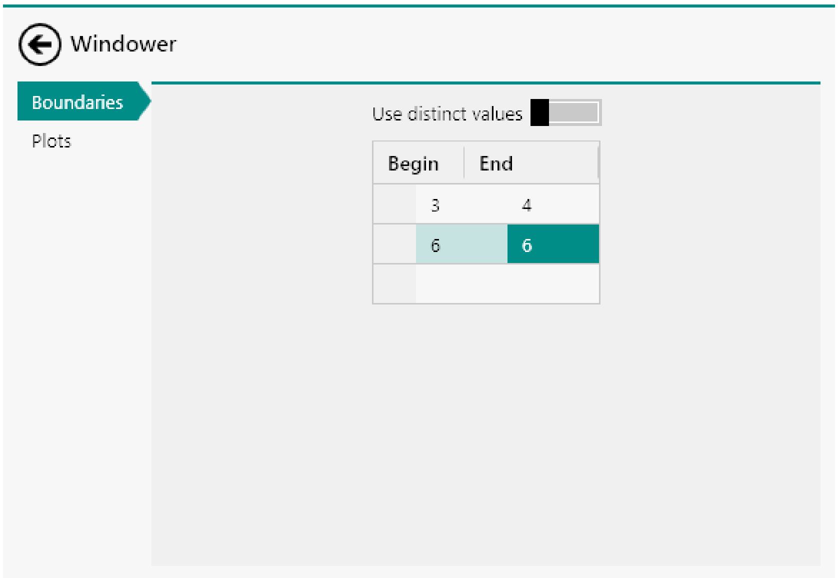
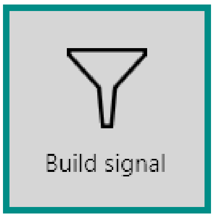


Figure 371 The boundaries can be converted from table view to checkboxes view

7.7.2 Build signal

	This command can be used to create signals based on measured or calculated signals and procedure properties, to be used in the data analysis.

The details of the command properties of the **Build signal** command are shown in *Figure 372*:

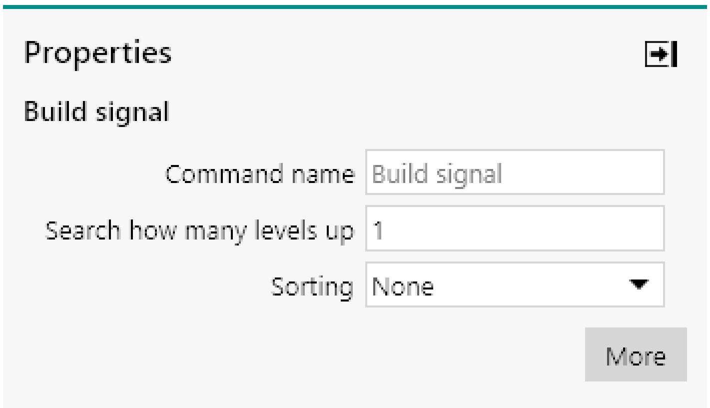


Figure 372 The properties of the Build signal command

The following properties are available:

- **Command name:** a user-defined name for the command.

- **Search how many levels up:** defines the search reach of the **Build signal** command. When this value is 1, the command will search all signals or properties that match the search criteria in the same command track and all sub-tracks. In order to search in command tracks located above the track in which the command is used, the *Search how many levels up* property can be increased.
- **Sorting:** defines if sorting is required, using the provided drop-down list. The command provides the choice between no sorting (None), ascending sorting (Low to high) or descending sorting (High to low). When the sorting option is used, the first signal created by the **Build signal** command will be used to sort all the signals created by the **Build signal** command.

The **Build signal** command can be used to extract command properties and signals for data handling properties. The command uses user-defined search criteria to create one or more signals containing the values of the signals or properties that are matching these search criteria. To define the search criteria, click the More button. The **Build signal** editor will be displayed *(see Figure 373, page 323)*.

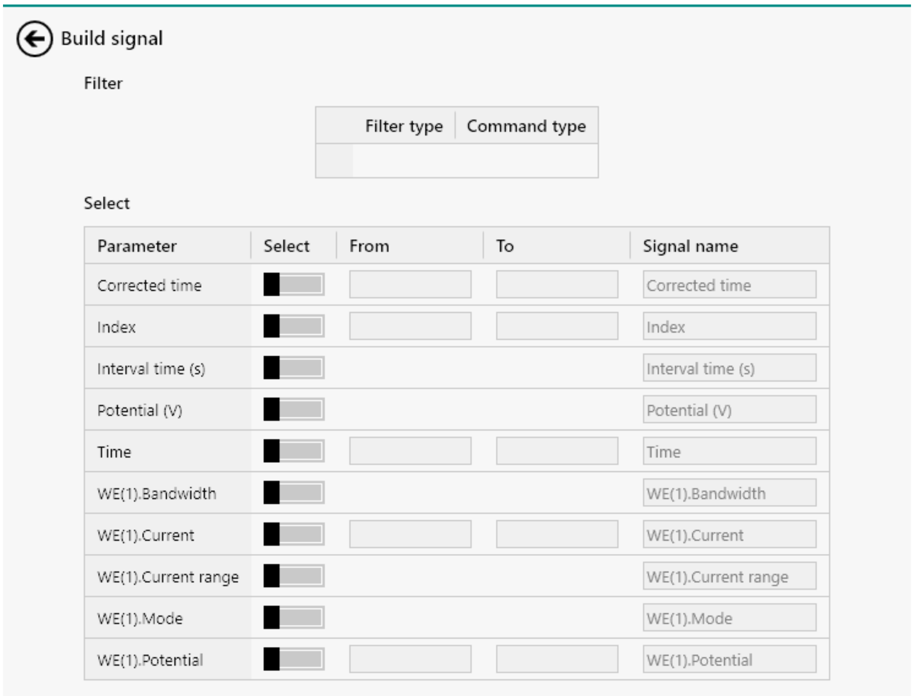


Figure 373 The Build signal editor

The **Build signal** editor provides two sub-panels *(see Figure 373, page 323)*:

- **Filter:** this sub-panel provides a table that can be used to specify the search criteria for creating the signal(s).
- **Select:** this sub-panel provides a list of properties and signals that fit the search criteria specified in the **Filter** sub-panel.

NOTE

By default, the **Build signal** command does not have pre-defined search criteria and it will list all available properties and signals in the **Select** panel.

7.7.2.1 Using the Filter

The **Filter** sub-panel can be used to specify search criteria in order to filter the available signals and properties. To filter the available signals and properties, a filter can be created using the table located in the **Filter** sub-panel. First a *Filter type* can be selected from the first available drop-down list *(see Figure 374, page 324)*.

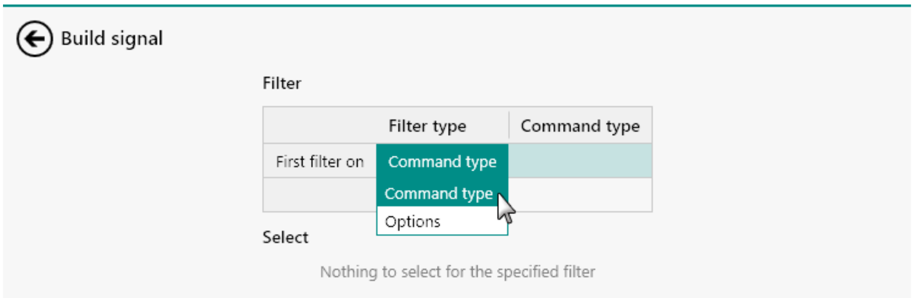


Figure 374 Selecting the Filter type

The drop-down list provides the choice between two possible filters:

- **Command type:** this filter provides the possibility to filter the properties and signals based on the type (or name) of a command.
- **Options:** this filter provides the possibility to filter the properties and signals based on a command option.

After specifying the *Filter type* property in the table, it is possible to select an available argument in first available cell of the second column of the table. The possible arguments are provided in a drop-down list *(see Figure 375, page 324)*.

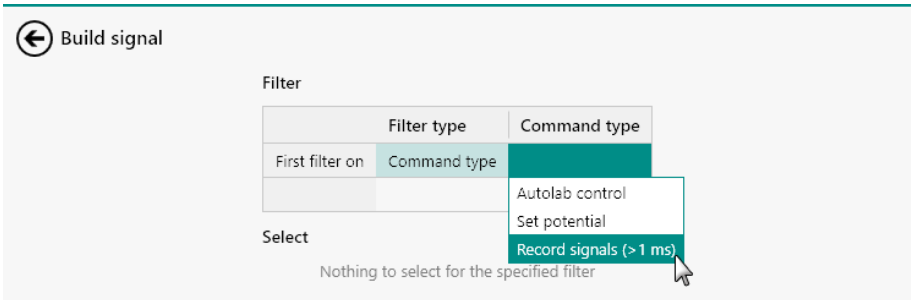


Figure 375 Selecting the Command type

NOTE

The available arguments provided in the drop-down list depend on the commands used in the procedure.

Once the argument is specified, the list of available signals and properties in the **Select** sub-panel will be updated. Only the signals and properties that match the filter criteria specified in the **Filter** sub-panel will be displayed *(see Figure 376, page 325)*.

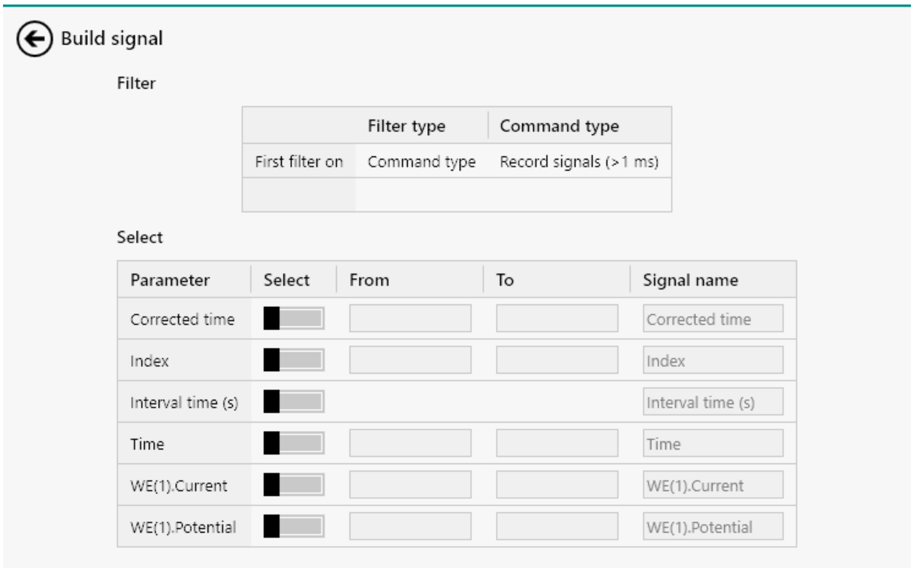


Figure 376 The filtered list of signals and properties

It is possible to create multiple conditions for filtering the properties and signals. For example, it is possible to add a first *Filter type* and *Command type*, shown in *Figure 377*.

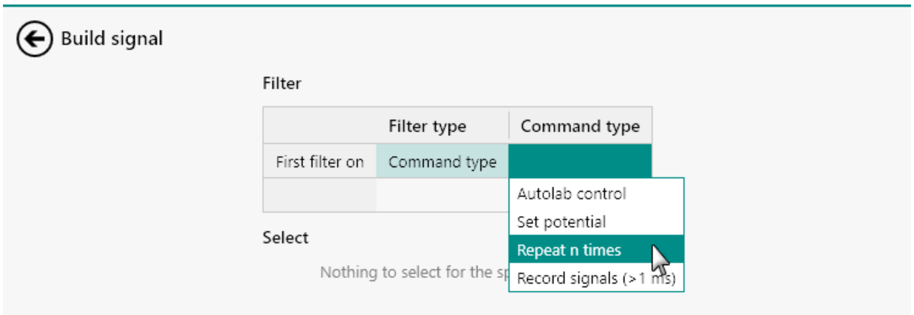


Figure 377 Combining multiple filter conditions

It is then possible to add a second *Filter type* and *Command type*, as shown in *Figure 378*.

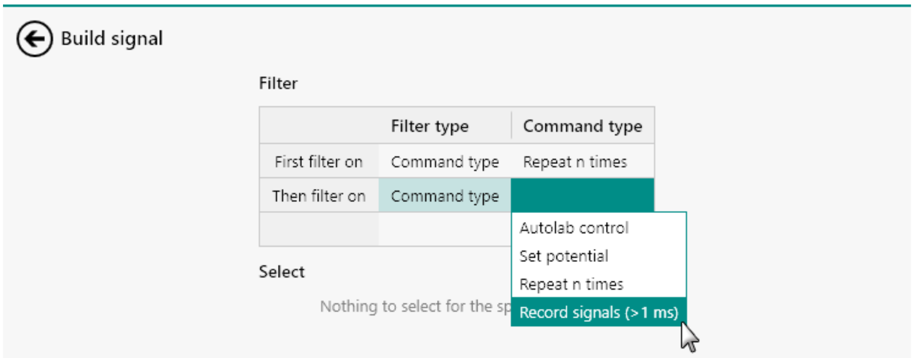


Figure 378 Adding a second condition to the filter

This filter condition will show only the signals and properties provided by all the **Record signals (> 1 ms)** commands located inside a **Repeat n times** command. These signals and properties will be shown in the **Select** sub-panel *(see Figure 379, page 326)*.

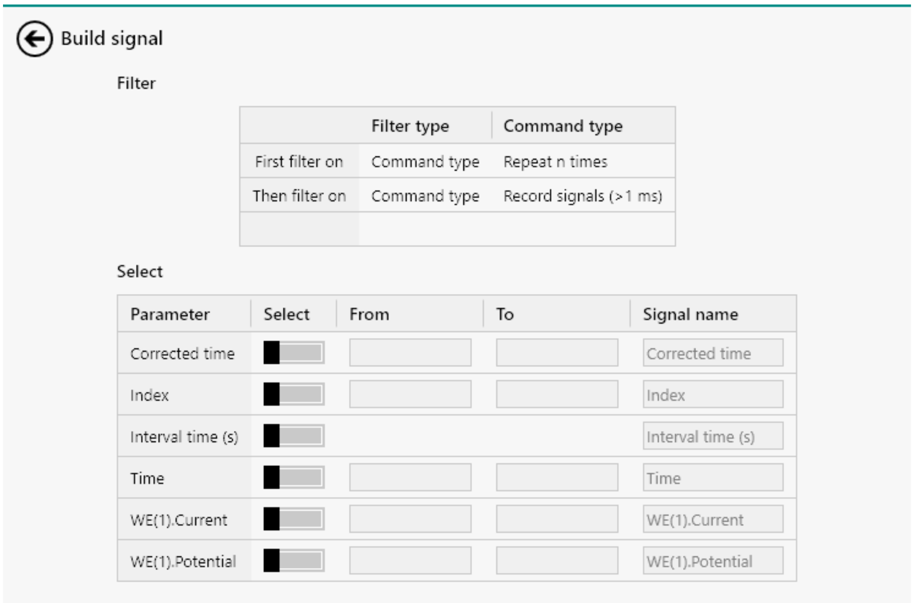


Figure 379 The updated list of signals and properties

7.7.2.2 Selecting the signals

The signals and properties shown in the **Select** panel can be added to the **Build signal** command by using the provided toggle *(see Figure 380, page 327)*.

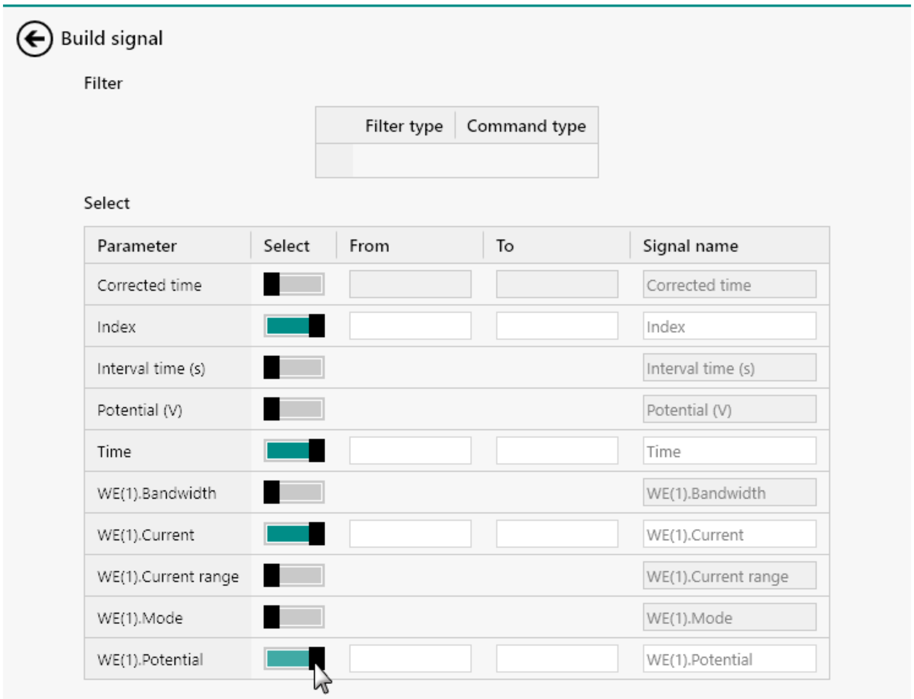


Figure 380 Selecting signals or properties in the Build signal editor

If a filter is used, only the signals and properties provided by the commands or options that fit the selection criteria specified by the filter will be shown *(see Figure 381, page 327)*.

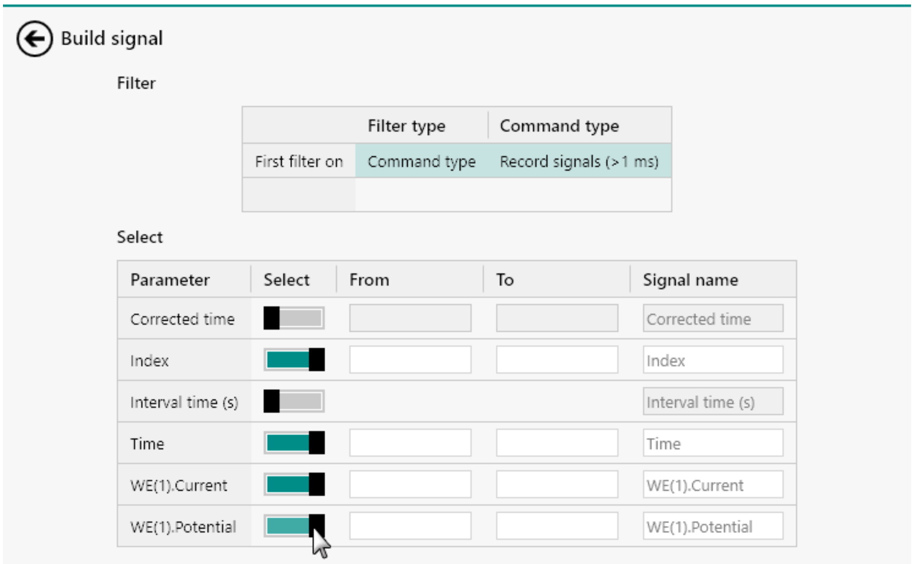


Figure 381 Selecting filtered signals or properties in the Build signal editor

For the signals listed in the **Select** sub-panel, it is also possible to define a range of values, by specifying an index range in the input fields provided next to each of the selected signals *(see Figure 382, page 328)*.

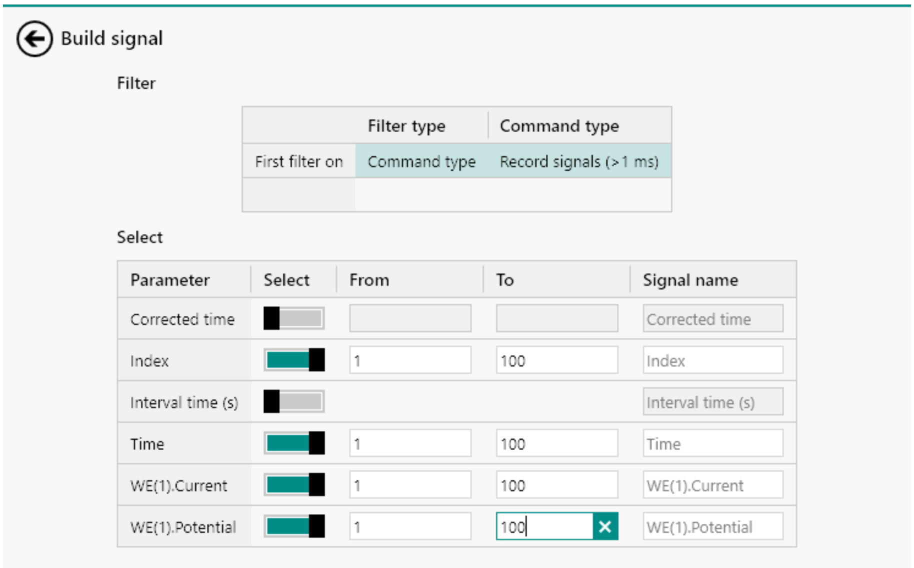


Figure 382   Specifying a range of values for the selected signals

CAUTION

When a range is specified it is necessary to specify two values (From and To).

NOTE

It is only possible to specify a range for signals.

If needed, a specific name can be specified in the Signal name field for each selected signal. By default, the name of the signal is shown in the Signal name column. However, it is possible to specify a custom name by typing it into the provided input field, as shown in *Figure 383*.

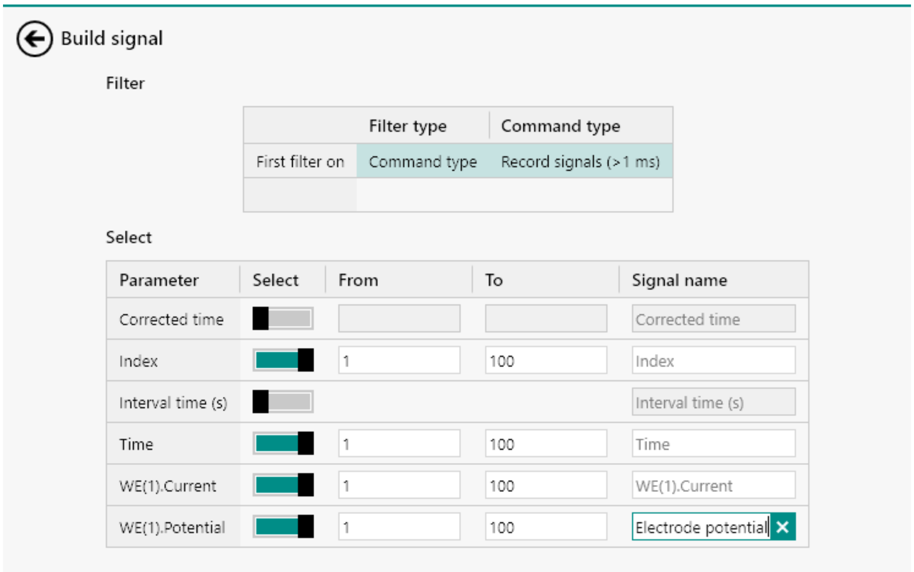


Figure 383 A custom name can be specified if needed

CAUTION

Each user-defined Signal name must be unique.

7.7.2.3 Remove a filter

To remove one of the filter conditions specified in the **Filter** sub-panel, click on the cell to select the complete row *(see Figure 384, page 329)*.

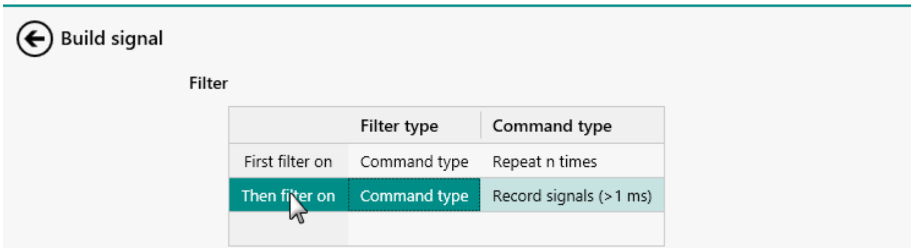


Figure 384 Click the row to select it

With the row selected, press the **[Delete]** key to remove the row from the table. The list of available signals and properties will be updated *(see Figure 385, page 330)*.

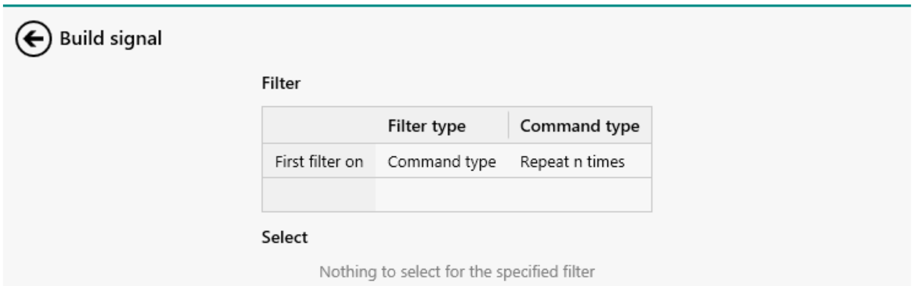
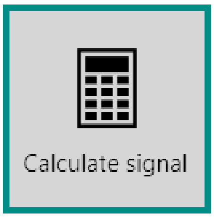


Figure 385 Deleting the row will trigger the filter to be updated

7.7.3 Calculate signal

	This command can be used to calculate signals based on measured or calculated signals and procedure properties, to be used in the data analysis.

The details of the command properties of the **Calculate signal** command are shown in *Figure 386*:

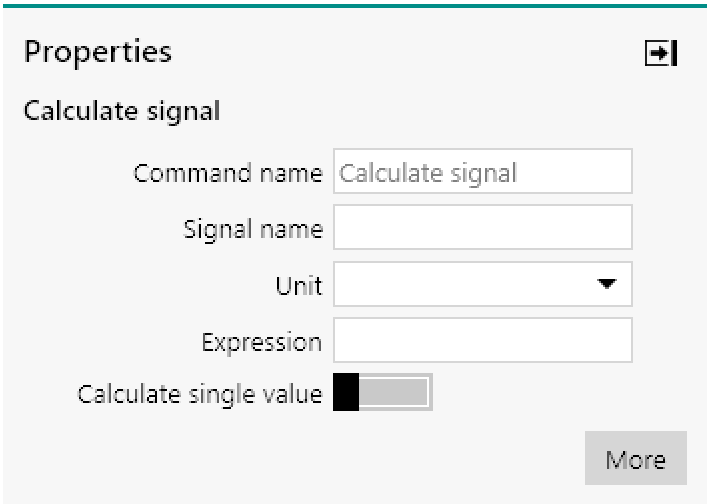


Figure 386 The properties of the Calculate signal command

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Signal name:** the name of the calculated signal.
- **Unit:** the unit of the calculated signal. The unit can either be typed in the input field or it can be picked from the drop-down list.
- **Expression:** the mathematical expression used to calculate the signal.
- **Calculate single value:** a toggle provided to force the **Calculate signal** command to return a single value.

To edit the **Expression** of the **Calculate signal** command, it is possible to use the controls provided in the **Properties** frame or to click the More button to switch to the **Calculate signal** editor *(see Figure 387, page 331)*.

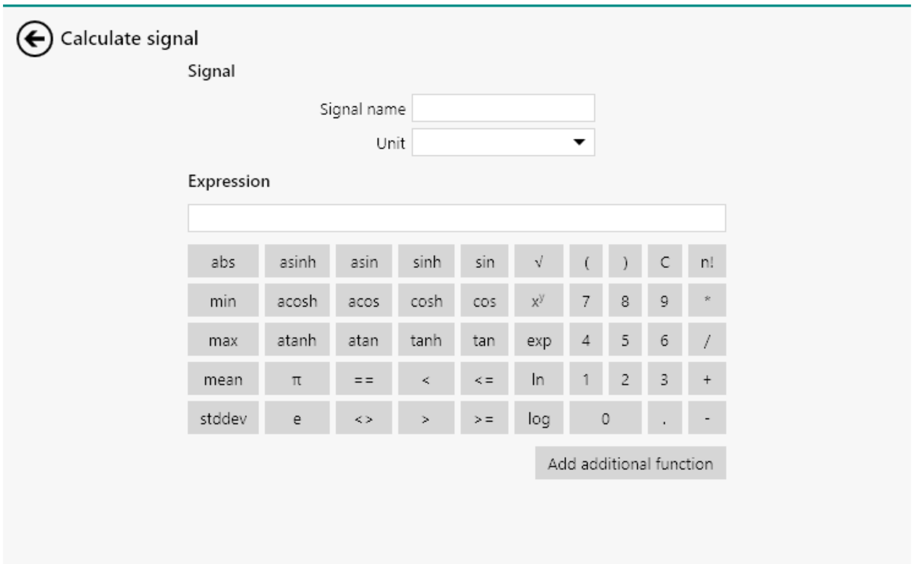


Figure 387 The Calculate signal editor

The **Calculate signal** editor can be used as a scientific calculator to build the expression used to calculate the signal. The mathematical operators provided by the **Calculate signal** editor can be accessed through the buttons or the Add additional function. Clicking this button opens a drop-down menu, displaying additional functions *(see Figure 388, page 332)*.

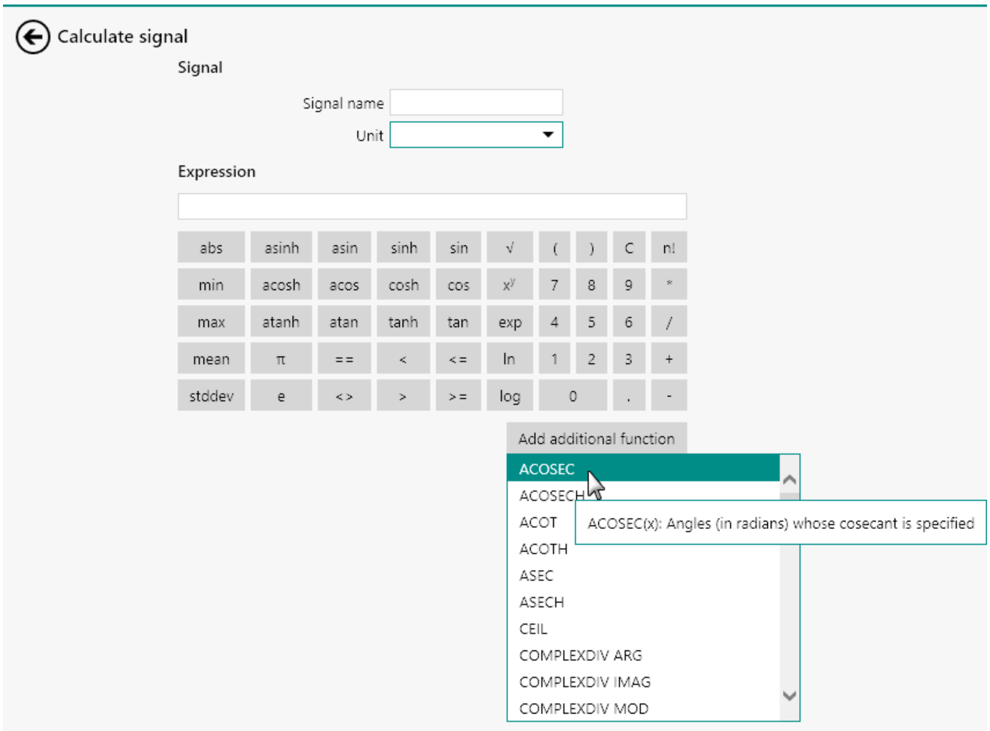


Figure 388 Additional functions can be accessed through using the provided button

The **Calculate signal** command automatically parses the mathematical expression in order to discriminate between mathematical operators and the arguments of the operators. In *Figure 389*, the *Current* string is identified as argument and the **10LOG** and **ABS** strings are identified as mathematical operators.

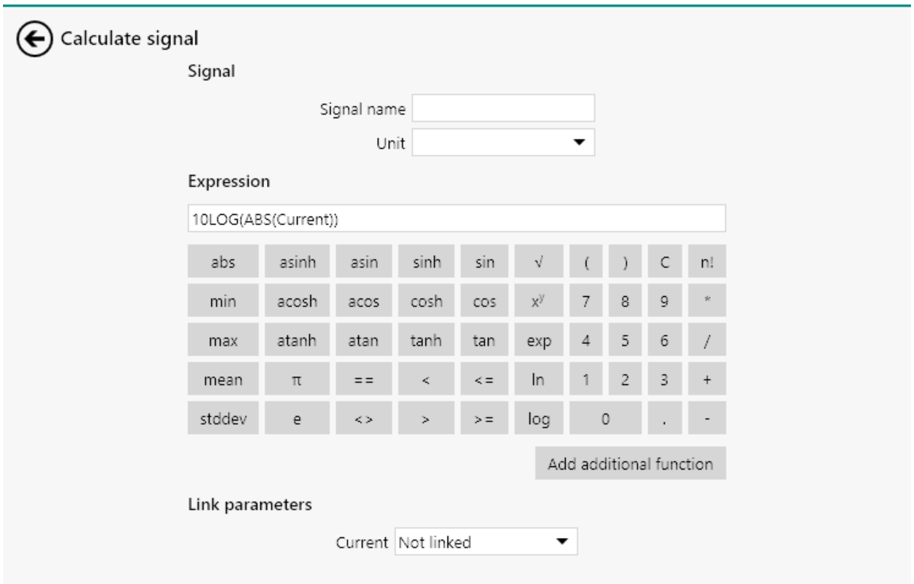


Figure 389 The Calculate signal automatically differentiates between operators and argument

If an error is made in the mathematical expression, the **Expression** field will be highlighted in red and a tooltip will indicate the nature of the error *(see Figure 390, page 333)*.

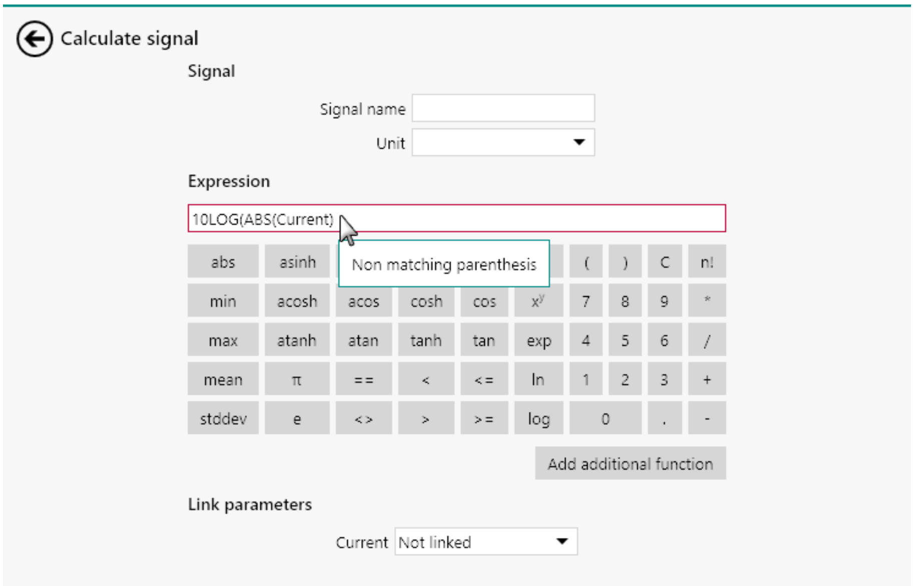


Figure 390 The expression is automatically tested for errors

After the expression is parsed, the identified arguments are listed in the **Link parameters** sub-panel *(see Figure 391, page 333)*.

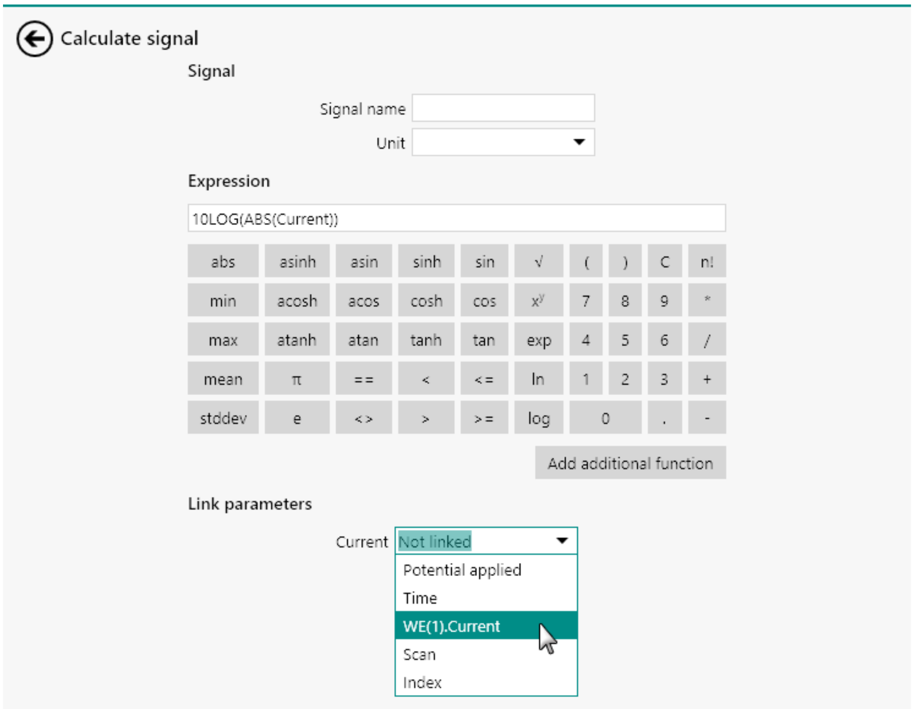


Figure 391 The identified arguments are listed in the Link parameters sub-panel

Depending on how the **Calculate signal** command is used, the arguments of the mathematical expression can be specified in two different ways:

1. Using the provided dropdown list *(see Chapter 7.7.3.1, page 334)*.
2. Using a link *(see Chapter 7.7.3.2, page 335)*.

7.7.3.1 Linking arguments using the drop-down list

If the **Calculate signal** is stacked onto a command that provides linkable properties, as shown in *Figure 392*, the arguments using the mathematical expression of the **Calculate signal** command can be directly linked using the drop-down list provided in the **Link parameter** sub-panel.

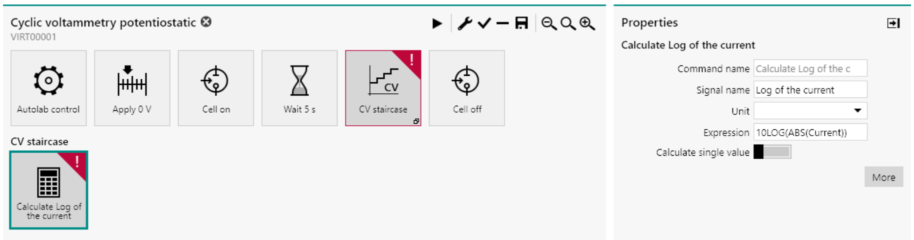


Figure 392 Using the Calculate signal when stacked onto another command

The available linkable properties provided by the command on which the **Calculate signal** is stacked are automatically populated in the drop-down list *(see Figure 393, page 335)*.

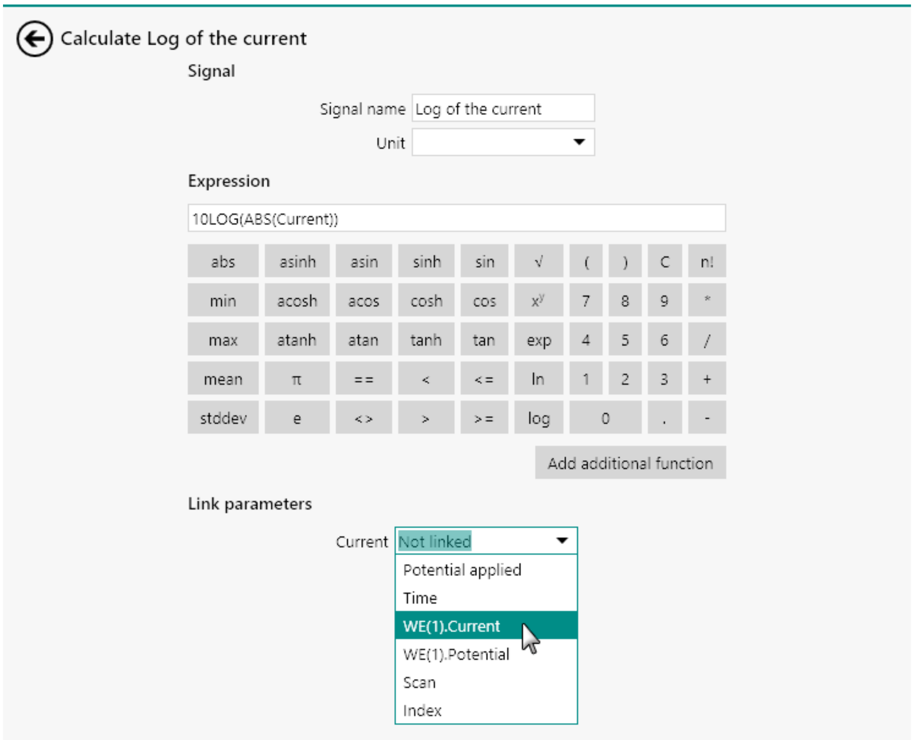


Figure 393 The available linkable properties are shown in the drop-down list

The selected property will be linked to the input argument of the **Calculate signal** command.

NOTE

For more information on stacking commands, please refer to *Chapter 10.12*.

NOTE

It is also possible to link the arguments of the **Calculate signal** command using the link method *(see Chapter 7.7.3.2, page 335)*.

7.7.3.2 Linking arguments with links

If the **Calculate signal** is **not** stacked onto a command that provides linkable properties, as shown in *Figure 394*, the arguments using the mathematical expression of the **Calculate signal** command have to be linked using a link.

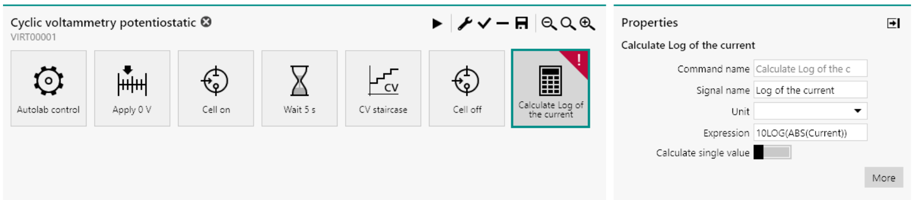


Figure 394 Using the Calculate signal in an arbitrary location in the procedure

In the **Edit links** screen, it is possible to link the argument of the **Calculate signal** command to another property in the procedure *(see Figure 395, page 336)*.

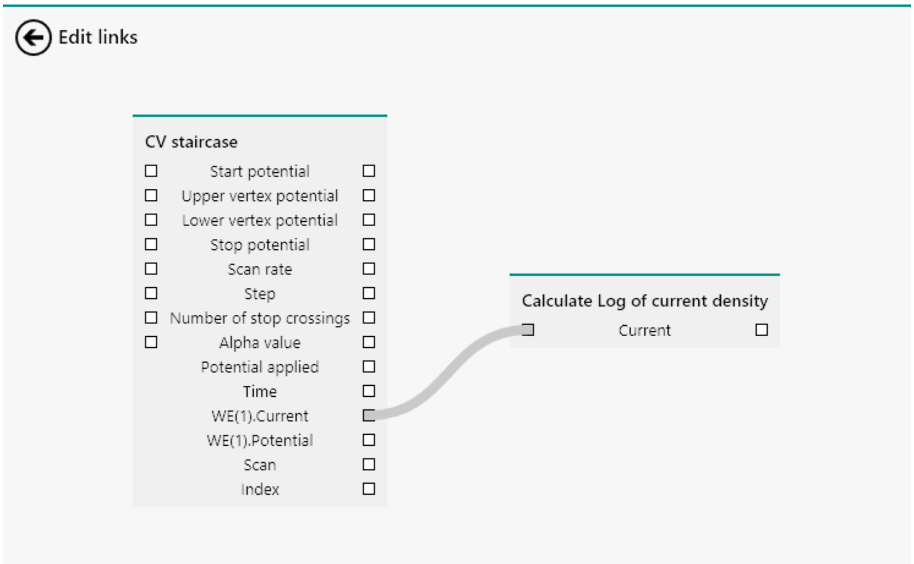


Figure 395 Linking the argument of the Calculate signal command

NOTE

More information on linking commands, please refer to *Chapter 10.13*.

7.7.3.3 Mathematical operators

Table 10 provides an overview of the mathematical or logical operators available using a dedicated **button** in the **Calculate signal** editor.

Table 10 Mathematical and logical operators provided in the Calculate signal editor

Mathematical operator	Button	Explanation
abs(x)	abs	Determines the absolute value of the argument x
asinh(x)	asinh	Determines the inverse hyperbolic sine of the argument x
asin(x)	asin	Determines the inverse sine of the argument x
sinh(x)	sinh	Determines the hyperbolic sine of the argument x
sin(x)	sin	Determines the sine of the argument x
sqrt(x)	√	Determines the square root of the argument x
fac(x)	n!	Determines the factorial of the argument x
min(x)	min	Determines the minimum value of the argument x
acosh(x)	acosh	Determines the inverse hyperbolic cosine of the argument x
acos(x)	acos	Determines the inverse cosine of the argument x
cosh(x)	cosh	Determines the hyperbolic cosine of the argument x
cos(x)	cos	Determines the cosine of the argument x
x^y	x^y	Raises the argument x to the power of y

Mathematical operator	Button	Explanation
max(x)	max	Determines the maximum value of the argument x
atanh(x)	atanh	Determines the inverse hyperbolic tangent of the argument x
atan(x)	atan	Determines the inverse tangent of the argument x
tanh(x)	tanh	Determines the hyperbolic tangent of the argument x
tan(x)	tan	Determines the tangent of the argument x
exp(x)	exp	Determines the exponential function of the argument x
mean(x)	mean	Determines the average value of the argument x
pi	π	The constant number π
(x)==(y)	==	Determines if the argument x is equal to the argument y
(x)<(y)	<	Determines if the argument x is smaller than the argument y
(x)<=(y)	<=	Determines if the argument x is smaller or equal to the argument y
ln(x)	ln	Determines the natural logarithm of the argument x

Mathematical operator	Button	Explanation
stddev(x)	stddev	Determines the standard deviation of the argument x
e(x)	e	Determines the exponential function of the argument x
(x)<>(y)	<>	Determines if the argument x not equal to the argument y
(x)>(y)	>	Determines if the argument x is larger than the argument y
(x)>=(y)	>=	Determines if the argument x is larger or equal to the argument y
log10(x)	log	Determines the 10 base logarithm of the argument x

7.7.3.4 Additional functions

Table 11 provides an overview of the mathematical or logical operators available using the Add additional function button in the **Calculate signal** editor.

NOTE

When the operators use more than one argument, the arguments need to be separated by a semi-colon (**;**).

Table 11 Additional functions provided in the Calculate signal editor

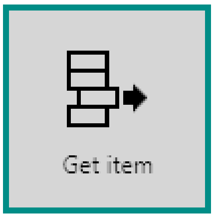
Function	Explanation
ACOSEC(x)	Returns the inverse cosecant of the argument *x*
ACOSECH(x)	Returns the hyperbolic inverse cosecant of the argument *x*
ACOT(x)	Returns the inverse cotangent of the argument *x*

Function	Explanation
ACOTH(x)	Returns the hyperbolic inverse cotangent of the argument *x*
ASEC(x)	Returns the inverse secant of the argument *x*
ASECH(x)	Returns the hyperbolic inverse secant of the argument *x*
CEIL(x)	Rounds the argument *x* to the next available integer
COMPLEXDIV ARG(x1;i1;x2;i2)	Determines the argument of the complex division of (*x1-j**i**1*) by (*x2-j**i**2*)
COMPLEXDIV IMAG(x1;i1;x2;i2)	Determines the imaginary part of the complex division of (*x1-j**i**1*) by (*x2-j**i**2*)
COMPLEXDIV MOD(x1;i1;x2;i2)	Determines the modulus of the complex division of (*x1-j**i**1*) by (*x2-j**i**2*)
COMPLEXDIV REAL(x1;i1;x2;i2)	Determines the real part of the complex division of (*x1-j**i**1*) by (*x2-j**i**2*)
COMPLEXMULT ARG(x1;i1;x2;i2)	Determines the argument of the complex multiplication of (*x1-j**i**1*) by (*x2-j**i**2*)
COMPLEXMULT IMAG(x1;i1;x2;i2)	Determines the imaginary part of the complex multiplication of (*x1-j**i**1*) by (*x2-j**i**2*)
COMPLEXMULT MOD(x1;i1;x2;i2)	Determines the modulus of the complex multiplication of (*x1-j**i**1*) by (*x2-j**i**2*)
COMPLEXMULT REAL(x1;i1;x2;i2)	Determines the real part of the complex multiplication of (*x1-j**i**1*) by (*x2-j**i**2*)
COSEC(x)	Returns the cosecant of the argument *x*
COSECH(x)	Returns the hyperbolic cosecant of the argument *x*

Function	Explanation
COT(x)	Returns the cotangent of the argument *x*
COTH(x)	Returns the hyperbolic cotangent of the argument *x*
DEGTORAG(x)	Converts the angle *x* from degrees to radians
DERIVATIVE(x;y;z)	Returns the z^{th} derivative of the argument *x* against the argument *y*
EXPAND(x;y)	Expands argument *x* by a factor of *y*
FFT FREQUENCY(x)	Returns the frequency of the Fast Fourier Transform of the argument *x*
FFT IMAG(x;bool)	Returns the real component of the Fast Fourier Transform of the argument *x* determined using a normal FFT (bool = 0) or normalized FFT (bool = 1)
FFT REAL(x;bool)	Returns the imaginary component of the Fast Fourier Transform of the argument *x* determined using a normal FFT (bool = 0) or normalized FFT (bool = 1)
FLOOR(x)	Rounds the argument *x* to the previous available integer
FPART(x)	Returns the fractional part of the argument *x*
INDEXER(x)	Indexes the argument *x* starting at a value of 1
INTEGRATE(x;y)	Returns the integral of the argument *x* against the argument *y*
ITEM(x;y)	Returns the y^{th} item of the argument *x*
LENGTH(x)	Return the length of the argument *x*

Function	Explanation
NLOG(x)	Returns the natural logarithm of the argument *x*
RADTODEG(x)	Converts the angle *x* from radians to degrees
ROUND(x)	Returns the rounded value of the argument *x*
SAVITZKY GOLAY(x;left;right;order)	Applies the Savitzky Golay smoothing on the argument *x*, using the specified **left** and **right** points and the specified polynomial **order**
SEC(x)	Returns the secant of the argument *x*
SECH(x)	Returns the hyperbolic secant of the argument *x*
SHRINK STANDARD(x;y)	Shrinks the size of the argument *x* to a new size of *y* keeping one value every *n*, where *n* is *x/y*
SHRINK DIFFERENTIAL (x;y;z)	Shrinks the size of the argument *x* to a new size of *z* using the derivative of the source arguments *dy/dx*, keeping the *z* highest derivative values
SHRINK DIFFERENTIAL ORDINATE(x;y;z)	Shrinks the size of the argument *y* to a new size of *z* using the derivative of the source arguments *dy/dx*, keeping the *z* highest derivative values
SHRINK MEAN(x;y)	Shrinks the size of the argument *x* to a new size of *y* using the average value of *n* values, where *n* is *x/y*
SIGNIFICANTS(x;y)	Formats the argument *x* to *y* significant digits
SPIKEREJECT(x)	Applies a spike rejection algorithm to the argument *x*

7.7.4 Get item

	This command can be used to extract a single value from a source data signal.

The details of the properties of the **Get item** command are shown in *Figure 396*:

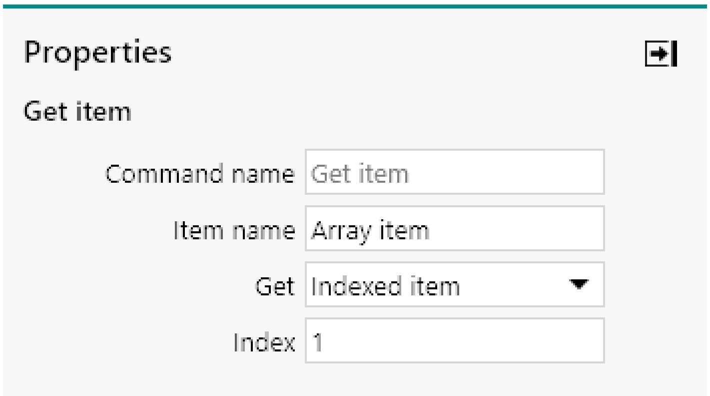


Figure 396 The properties of the Get item command

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Item name:** the name of the extracted item. The extracted item will be identified by the specified **Item name**.
- **Get:** defines which item to get, using the provided drop-down list. The value is returned as a single value. Three settings are provided:
  - **First item:** gets the first item of the source signal.
  - **Last item:** gets the last item of the source signal.
  - **Indexed item:** gets the item corresponding to the specified index value.
- **Index:** the index of the item to get. This property is only shown when the **Get** property is set to *Indexed item*.

7.7.5 Import data

	This command can be used to import data from an external file. The command support data files GPES and FRA as well as ASCII data.

NOTE

This command can be used without an Autolab connected to the computer.

The details of the properties of the **Import data** command are shown in *Figure 397*:

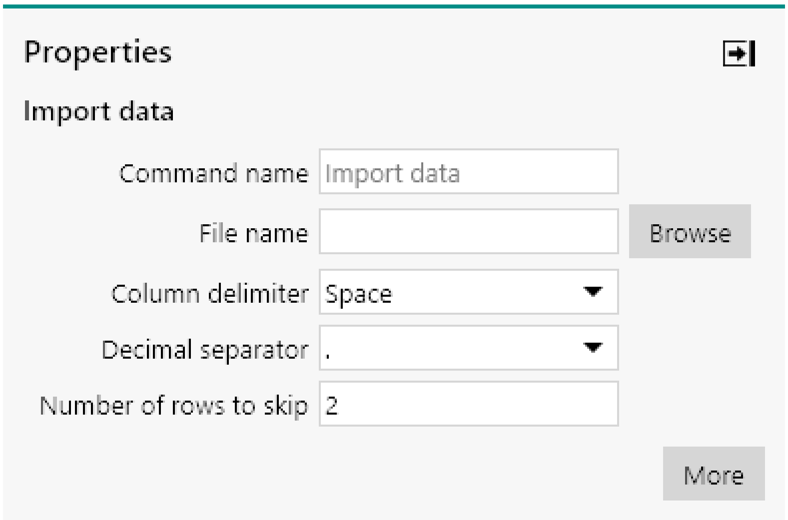


Figure 397 The properties of the Import data command

The following properties are available:

- **Command name:** a user-defined name for the command.
- **File name:** the path to the file containing the data to import. A Browse button is provided in order to specify the location of the file through a Windows Explorer dialog.
- **Column delimiter:** a drop-down list that provides the choice of column delimiter (Space, Tab, Comma (,), Semicolon (;) or Colon (:)). This property only applies to ASCII files.
- **Decimal separator:** a drop-down list that provides the choice of decimal separator (Dot (.) or Comma (,)). This property only applies to ASCII files.
- **Number of rows to skip:** defines the number of rows to skip when importing the data. This only applies to ASCII files.

The file name and location can be specified directly or using Browse button. The Windows Explorer dialog provides the means to import GPES, FRA or any type of file *(see Figure 398, page 345)*.

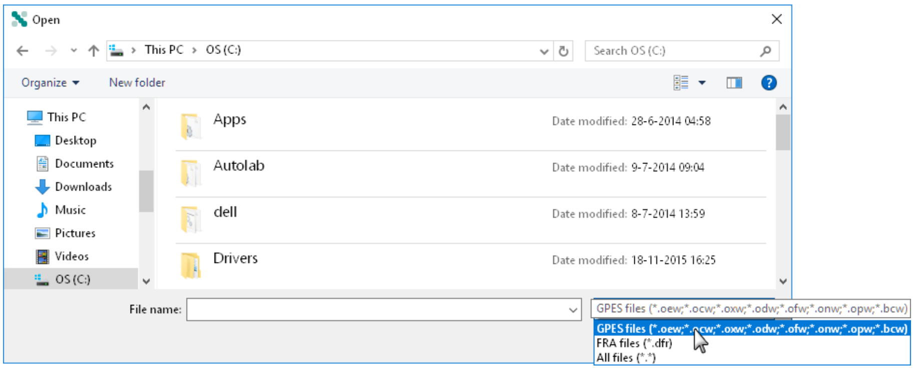


Figure 398 The file type can be adjusted in the Windows Explorer dialog

NOTE

The **Import data** command automatically adjusts the properties displayed based on the extension of the specified file.

Additional settings are available by clicking the More in the **Properties** panel. A new screen will be displayed as shown as *Figure 397*.

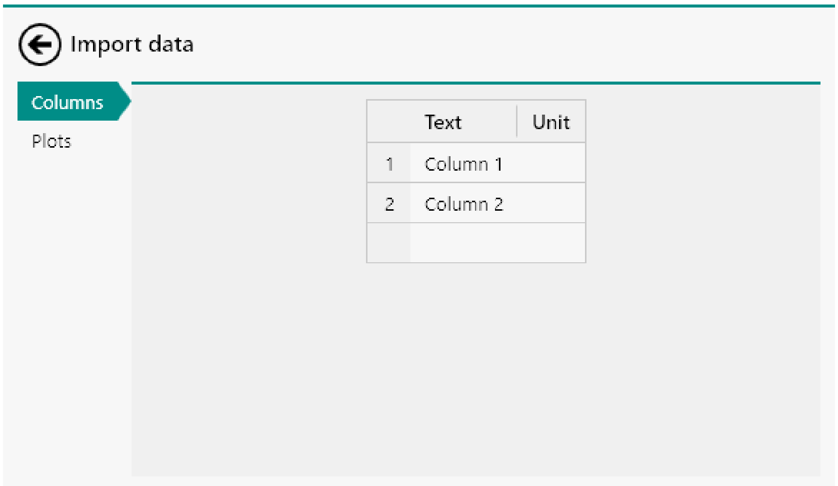


Figure 399 Additional settings are available for the Import data command

Depending on the type of file imported, the following additional settings are available:

- **Columns:** the Columns editor can be used when importing ASCII file to specify the number of columns in the source file, assign a name to each column and specify units for the data in each column, if applicable.

- **Plots:** the Plot editor can be used to specify how the data imported by the file should be displayed. This editor is available for all the file types and the use of the plot editor is explained in *Chapter 9.5*.

To specify the number of columns in the ASCII file import using the **Import Data** command, the Columns editor can be used. By default, two rows are specified in the table and additional rows can be added for additional columns in the table. A signal name can be provided in each text cell in the table and units can be added as well. For example, if the imported file has three columns, with the first one being Time, the second Current and the third one Potential, the Columns editor can be adjusted as shown in *Figure 400*.

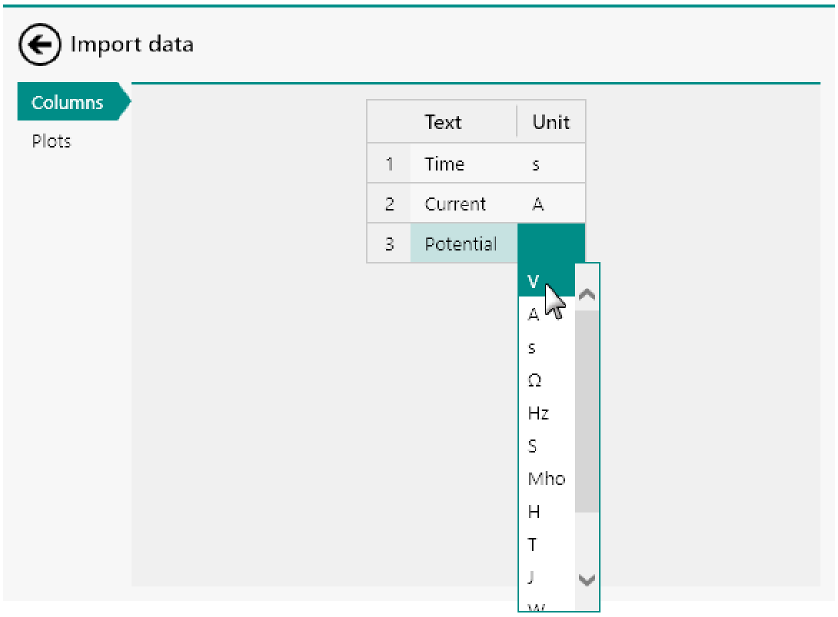


Figure 400 Using the Columns editor

Clicking a cell in the table shows a drop-down list with a number of predefined signal names or units. If needed, a custom name and unit can be specified by typing directly in the selected cell of the table *(see Figure 401, page 347)*.

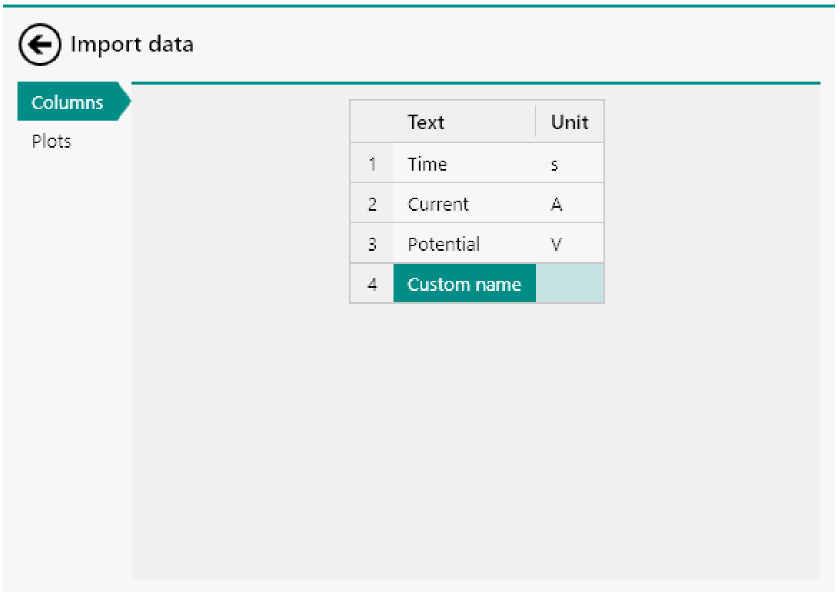
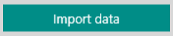


Figure 401 Specifying a custom name

NOTE

GPES and FRA data files can also be imported directly using the 

button located in the **Actions** panel of the **Dashboard** *(see Chapter 4.1, page 74)*.

7.7.6 Export data

	This command can be used to export data from an external ASCII or ZView file.

NOTE

This command can be used without an Autolab connected to the computer.

The details of the properties of the **Export data** command are shown in *Figure 402*:

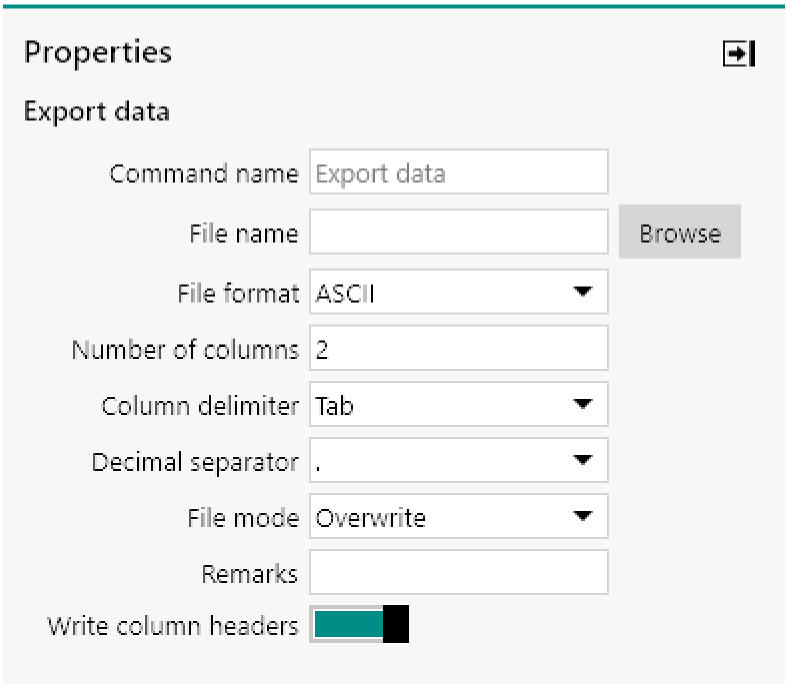


Figure 402 The properties of the Export data command

The following properties are available:

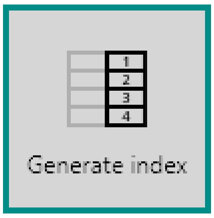
- **Command name:** a user-defined name for the command.
- **File name:** the path and file name of the exported data file. A Browse button is provided in order to specify the location and name of the file through a Windows Explorer dialog.
- **File format:** specifies the type of output file (ASCII or ZView) using the provided drop-down list.
- **Number of columns:** specifies the number of columns in the output file. This property only applies to ASCII files.
- **Column delimiter:** a drop-down list that provides the choice of column delimiter (Space, Tab, Comma (,), Semicolon (;) or Colon (:)). This property only applies to ASCII files.
- **Decimal separator:** a drop-down list that provides the choice of decimal separator (Dot (.) or Comma (,)). This property only applies to ASCII files.

- **File mode:** specifies what should be done is the specified file already exists in the specified location. This property only applies to ASCII files. Using the provided drop-down list, it is possible to choose from:
  - **Overwrite:** using this setting, the content of the file is overwritten.
  - **Append:** using this setting, the new data exported by the command is added to the existing file, immediately below the last row of data in the file.
  - **Make unique:** using this setting, a new file is created, with the same name as specified in the command and an index number between round brackets.
- **Remarks:** a remarks field available for bookkeeping purposes. This property only applies to ASCII files.
- **Write column headers:** defines if the name of the signals written to the file should be included in the header, using the provided toggle. This property is only applies to ASCII files.

NOTE

The actual data to be exported by the **Export data** command is specified by linking.

7.7.7 Generate index

	This command can be used to index the source data this command is added to. The command add an index column to the source data.

The details of the command properties of the **Generate index** command are shown in *Figure 403*.

Figure 403 The properties of the Generate index command

The following properties are available:

- **Command name:** a user-defined name for the command.

NOTE

The **Generate index** command can not be used stand alone. This command is designed to work in conjunction with another command providing the data to index. The **Generate index** command can be *stacked* onto the command providing the source data *(see Chapter 10.12, page 653)*.

7.7.8 Shrink data

	This data can be used to shrink the source data to a smaller set.

The details of the properties of the **Shrink data** command are shown in *Figure 404*:

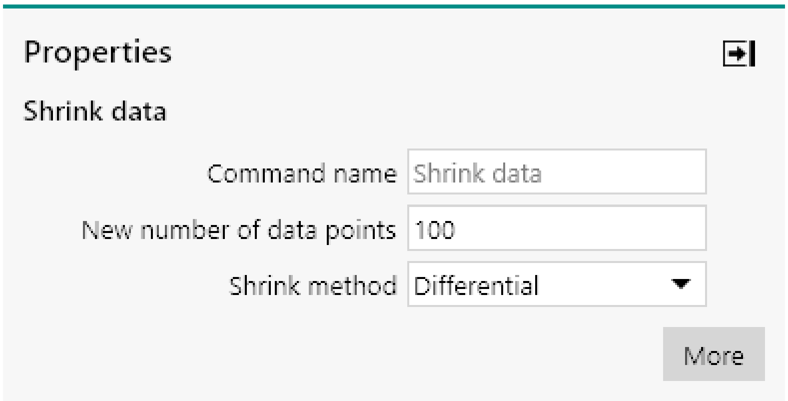


Figure 404 The properties of the Shrink data command

The following properties are available:

- **Command name:** a user-defined name for the command.
- **New number of data points:** specifies the number of data points generated by the shrink algorithm. This value must be smaller or equal than the number of points in the source data. The number of data points in the source data divided by the specified *New number of data points* provides the reduction factor, *n*.

- **Shrink method** specifies the shrink method, using the provided drop-down list. Three shrink methods are available:
  - **Standard:** using this method, points are removed from the source data without a specific selection argument. This method keep a data point out of every *n* data points.
  - **Mean:** using this method, the average value of *n* data points in the source data is determined and stored in the shrinked data.
  - **Differential:** using this method, the selection is based on the differential of the source data, *dY/dX*. The points with the highest differential are kept while the other points are discarded.

NOTE

The **Shrink data** command can not be used stand alone. This command is designed to work in conjunction with another command providing source data used by the **Shrink data** command. The **Shrink data** command can *linked* to the command providing the source data *(see Chapter 10.13, page 657)*.

NOTE

The **Shrink data** needs a *X source* signal and a *Y source* signal.

Additional properties are available by clicking the More in the **Properties** panel. A new screen will be displayed *(see Figure 405, page 351)*.

Figure 405 Additional properties are available for Shrink data command

The following additional settings are available:

- **Custom plots:** the Plot editor can be used to specify how the shrinked data should be displayed. The use of the custom plot editor is explained in *Chapter 9.5*.

7.8 Analysis - general commands

Analysis - general commands can be used to perform data analysis on measured data or to integrate data analysis steps in a procedure.

The available commands are represented by a shortcut icon *(see Figure 406, page 352)*.

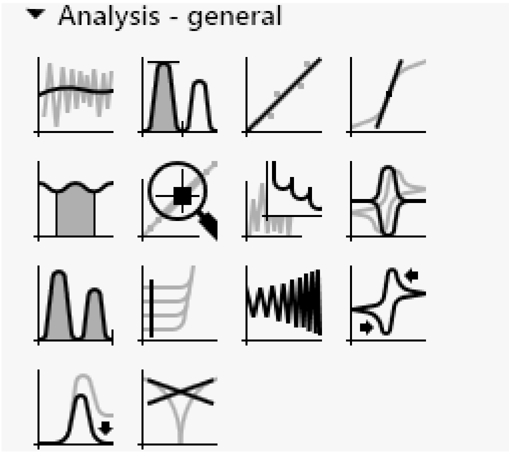


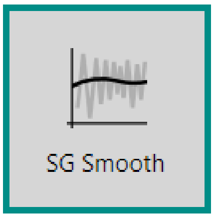
Figure 406 The Analysis - general commands

The following commands are available:

- **Smooth:** a command which can be used to smooth measured data to remove noise or spikes *(see Chapter 7.8.1, page 353)*.
- **Peak search:** a command which can be used to find peaks in measured data *(see Chapter 7.8.2, page 356)*.
- **Regression:** a command which can be used to perform a regression on measured data *(see Chapter 7.8.3, page 358)*.
- **Derivative:** a command which can be used to calculate the first derivative of the provided data *(see Chapter 7.8.4, page 361)*.
- **Integrate:** a command which can be used to calculate the integral of the provided data *(see Chapter 7.8.5, page 363)*.
- **Interpolate:** a command which can be used to determine data points by linear interpolation of measured data *(see Chapter 7.8.6, page 364)*.
- **FFT analysis:** a command which can be used to transform time domain data into frequency domain data by applying a Fast Fourier Transform on the provided data *(see Chapter 7.8.7, page 364)*.
- **Convolution:** a command which can be used to perform a convolution analysis on the provided data *(see Chapter 7.8.8, page 366)*.
- **Calculate charge:** a command which can be used to determine the charge from the measured current *(see Chapter 7.8.9, page 370)*.

- **Hydrodynamic analysis:** a command which can be used to perform a Levich and Koutecký-Levich analysis on measured data recorded using forced convection, using the Autolab rotating disk electrode (RDE) or the Autolab rotating ring disk electrode (RRDE) *(see Chapter 7.8.10, page 371)*.
- **ECN spectral noise analysis:** a command which can be used to analyze electrochemical noise (ECN) data *(see Chapter 7.8.11, page 372)*.
- **iR drop correction:** a command which can be used to correct measured data for ohmic losses *(see Chapter 7.8.12, page 376)*.
- **Baseline correction:** a command which can be used to subtract a baseline from the measured data *(see Chapter 7.8.13, page 377)*.
- **Corrosion rate analysis:** a command which can be used to analyze linear polarization data and determine the corrosion rate *(see Chapter 7.8.14, page 383)*.

7.8.1 Smooth

	This command can be used to smooth data and remove spikes.

The **Smooth** command can be used in two different modes, which can be selected using the provided drop-down list *(see Figure 407, page 353)*:

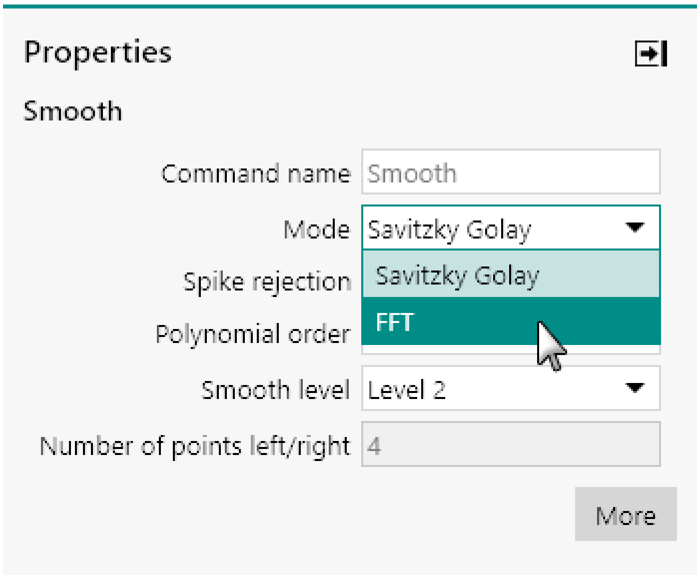


Figure 407 Two modes are provided by the Smooth command

1. Savitzky-Golay (SG) smooth (default mode)
2. FFT smooth

NOTE

The Savitzky-Golay (SG) smoothing method is described in Anal. Chem.,36, 1627 (1964). It involves a polynomial fit through the experimental data. This method is also called weighted moving averaging.

NOTE

The **Smooth** command description in the procedure editor is dynamically adjusted in function of the specified mode.

7.8.1.1 SG Smooth

The following properties are available when the command is used in the *SG Smooth* mode *(see Figure 408, page 354)*:

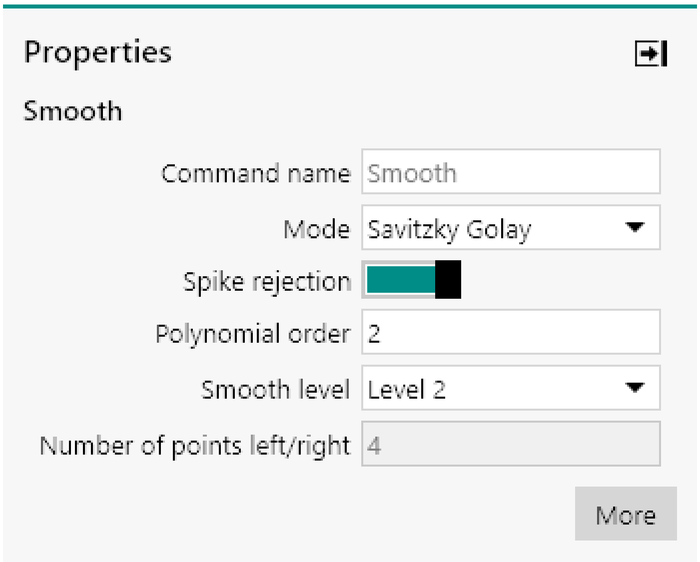


Figure 408 SG Smooth mode properties

- **Command name:** a user-defined name for the command.
- **Spike rejection:** a toggle which can be used to enable or disable spike rejection.
- **Polynomial order:** defines the order of the polynomial function fitted through the data. Small order leads to heavy smoothing (default value: 2).

- **Smooth level:** defines the smoothing level by defining the number of points in the weighted moving average function, using the provided drop-down list. The higher the level, the heavier the smoothing (default level: Level 2). Five pre-defined levels are available:
  - **None:** no smoothing is used.
  - **Level 1:** 5-point weighed moving average (2 point left and right).
  - **Level 2:** 9-point weighed moving average (4 points left and right).
  - **Level 3:** 15-point weighed moving average (7 points left and right).
  - **Level 4:** 23-point weighed moving average (11 points left and right).
- **Number of points left/right:** defines the number of data points left of the center of the weighted moving average when the **Smooth level** property is set to *User defined*. The larger the value, the heavier the smoothing. When the **Smooth level** is set to one of the predefined levels, this property is automatically adjusted.

NOTE

The **Number of points left/right** defines the size of the weighted moving average.

7.8.1.2 FFT Smooth

The following properties are available when the command is used in the *FFT Smooth* mode *(see Figure 409, page 355)*:

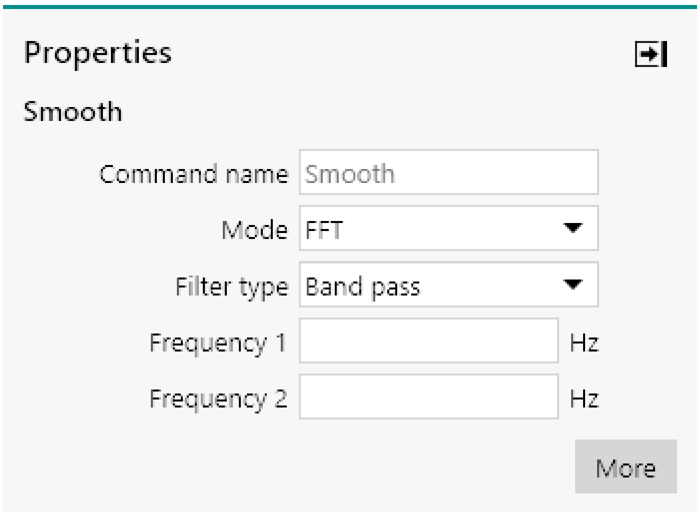


Figure 409 FFT Smooth mode properties

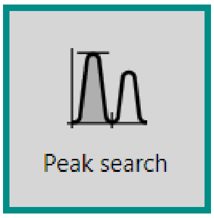
- **Command name:** a user-defined name for the command.

- **Filter type:** defines the type of FFT filter used by the command, using the provided drop-down list. Four different filter types are available for the *FFT Smooth* mode:
  - **Low pass:** all the contributions from frequencies higher than the user-selected cutoff frequency are rejected. This method can be used to remove high frequency noise from a measurement.
  - **High pass:** all the contributions from frequencies lower than the user-selected cutoff frequency are rejected. This method can be used to remove low frequency noise from a measurement.
  - **Band pass:** only the contributions from frequencies within a user-defined frequency range are kept. All frequencies that fall outside of the user defined range are rejected.
  - **Band stop:** all the contributions from frequencies within a user-defined frequency range are rejected. Only the frequencies that fall outside of the user defined range are kept.
- **Frequency 1:** defines the first frequency limit used by the command, in Hz.
- **Frequency 2:** defines the second frequency limit used by the command, in Hz.

NOTE

When the **Filter type** property is set to Low pass or High pass, only one frequency can be specified.

7.8.2 Peak search

	This command can be used to find peaks in the source data. The source data contains X and Y values.

The details of the properties of the **Peak search** command are shown in *Figure 410*.

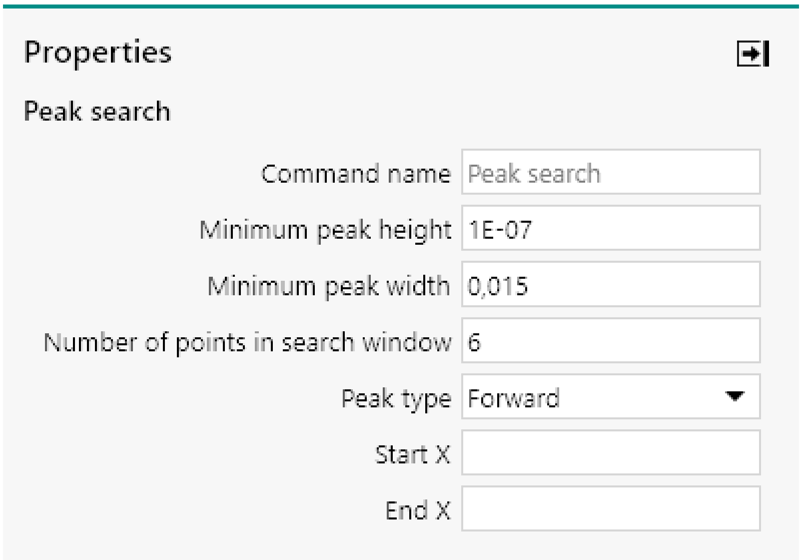


Figure 410 The properties of the Peak search command

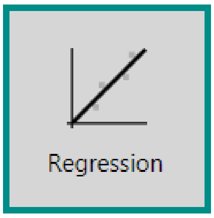
The following properties are available:

- **Command name:** a user-defined name for the command.
- **Minimum peak height:** defines the minimum height of a peak, in units of the Y source data. When this value is set to 0, then this property is not used to find peaks.
- **Minimum peak width:** defines the minimum width of a peak, in units of the X source data. When this value is set to 0, then this property is not used to find peaks.
- **Number of points in search window:** this property defines the number of points that must be located above and below a zero crossing of the first derivative of the signal (dY/dX), in order to qualify as a peak. This setting is useful to discriminate between noise and real peaks. The default value is 6.
- **Peak type:** defines the type of peaks to search, forward or reverse, using the provided dropdown list. Using the forward setting, NOVA will search for regular peaks (anodic peak during the positive going scan or cathodic peak in the opposite direction). The reverse setting allows NOVA to search for peaks in the opposite direction.
- **Start X:** defines the initial abscissa used for the peak search.
- **End X:** defines the final abscissa used for the peak search.

NOTE

When the **Start X** and **End X** properties are not defined, the peaks will be searched in the whole range of X values provided in the source data.

7.8.3 Regression

	This command can be used to perform a regression on the source data. The source data contains X and Y values.

The **Regression** command provides three different modes, which can be selected using the provided drop-down list *(see Figure 411, page 358)*:

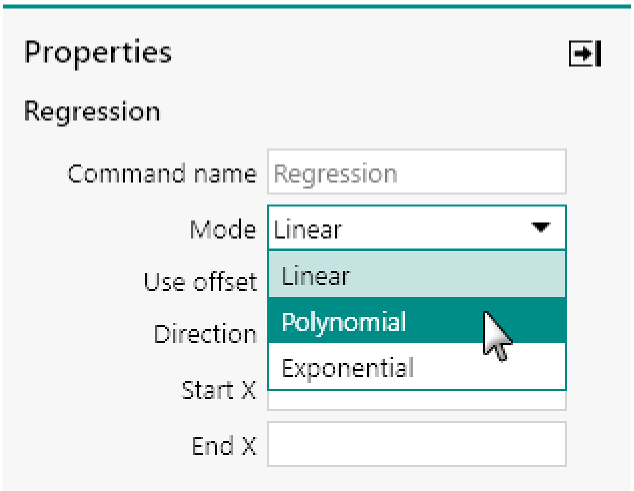


Figure 411 The Regression command provides three regression modes

1. **Linear:** performs a linear regression (default mode)
2. **Polynomial:** performs a polynomial regression
3. **Exponential:** performs an exponential regression

7.8.3.1 Linear regression

The following properties are available when the **Regression** command is used in *Linear* mode *(see Figure 412, page 359)*:

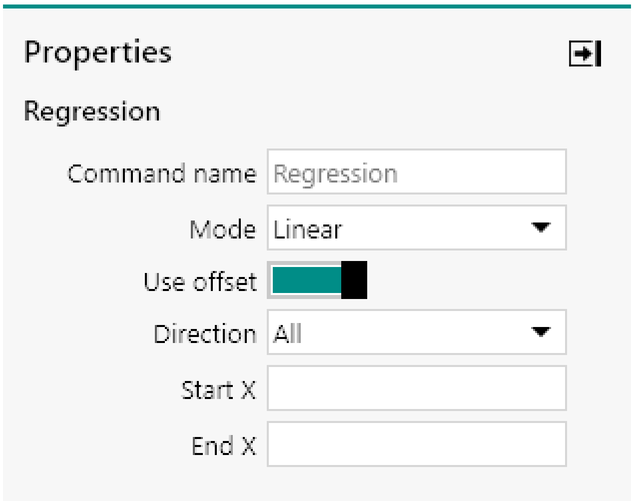


Figure 412 The properties of the Linear mode of the Regression command

- **Command name:** a user-defined name for the command.
- **Use offset:** specifies if an offset should be used in the regression, using the provided  toggle. Depending on this toggle, the following equations are used:
  - **Use offset off:** performs a regression using the equation **y = ax**.
  - **Use offset on:** performs a regression using the equation **y = ax + b**.
- **Direction:** specifies the direction to use in the calculation, using the provided dropdown list. Three directions are available:
  - **All:** all the data provided in the source data is used for the regression. This is the default direction.
  - **Forward:** only the data values in the positive going direction is used for the regression.
  - **Reverse:** only the data values in the negative going direction is used for the regression.
- **Start X:** defines the initial abscissa used for the regression.
- **End X:** defines the final abscissa used for the regression.

7.8.3.2 Polynomial regression

The following properties are available when the **Regression** command is used in *Polynomial* mode *(see Figure 413, page 360)*:

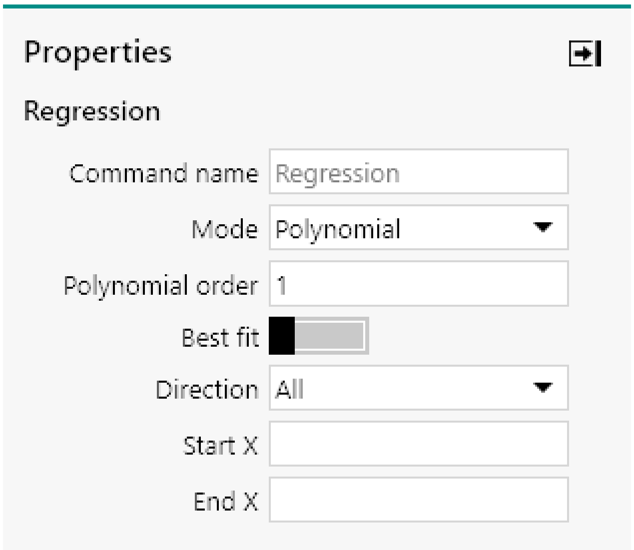


Figure 413 The properties of the Polynomial mode of the Regression command

- **Command name:** a user-defined name for the command.
- **Polynomial order:** specifies the polynomial order used by the regression.
- **Best fit:** specifies if the best fit should be used in the regression, using the provided toggle. Depending on this toggle, the following equations are used:
  - **Best fit off:** performs a regression using the specified polynomial order.
  - **Best fit on:** performs a regression using all the polynomial functions up to the maximum polynomial order. The regression providing the smallest χ^2 (Chi-squared) is automatically selected by the software.
- **Direction:** specifies the direction to use in the calculation, using the provided dropdown list. Three directions are available:
  - **All:** all the data provided in the source data is used for the regression. This is the default direction.
  - **Forward:** only the data values in the positive going direction is used for the regression.
  - **Reverse:** only the data values in the negative going direction is used for the regression.
- **Start X:** defines the initial abscissa used for the regression.
- **End X:** defines the final abscissa used for the regression.

7.8.3.3 Exponential regression

The following properties are available when the **Regression** command is used in *Exponential* mode *(see Figure 414, page 361)*:

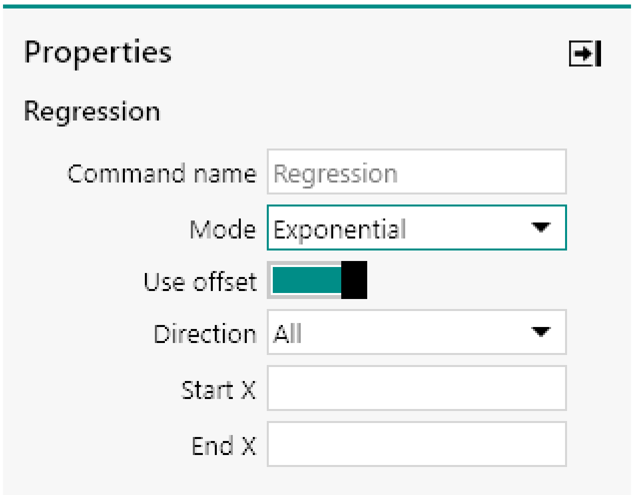
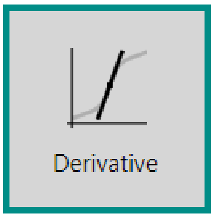


Figure 414 The properties of the Exponential mode of the Regression command

- **Command name:** a user-defined name for the command.
- **Use offset:** specifies if an offset should be used in the regression, using the provided toggle. Depending on this toggle, the following equations are used:
  - **Use offset off:** performs a regression using the equation $\mathbf{y = be^{cx}}$.
  - **Use offset on:** performs a regression using the equation $\mathbf{y = a + be^{cx}}$.
- **Direction:** specifies the direction to use in the calculation, using the provided dropdown list. Three directions are available:
  - **All:** all the data provided in the source data is used for the regression. This is the default direction.
  - **Forward:** only the data values in the positive going direction is used for the regression.
  - **Reverse:** only the data values in the negative going direction is used for the regression.
- **Start X:** defines the initial abscissa used for the regression.
- **End X:** defines the final abscissa used for the regression.

7.8.4 Derivative

	This command can be used to determine the first derivative of a data set.

The details of the command properties of the **Derivative** command are shown in *Figure 415*.

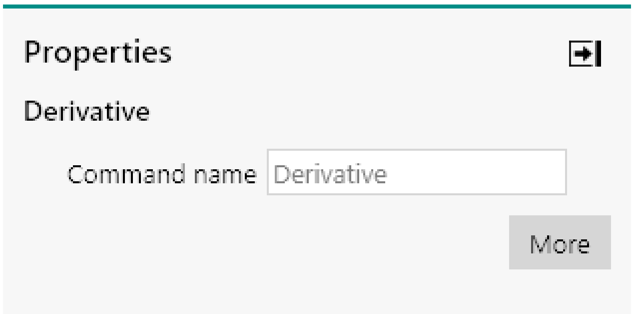


Figure 415 The properties of the Derivative command

The following properties are available:

- **Command name:** a user-defined name for the command.

This command needs to be *linked* to sourced data *(see Chapter 10.13, page 657)*. The **Derivative** command provides two input anchoring points and four output anchoring points *(see Figure 416, page 362)*.

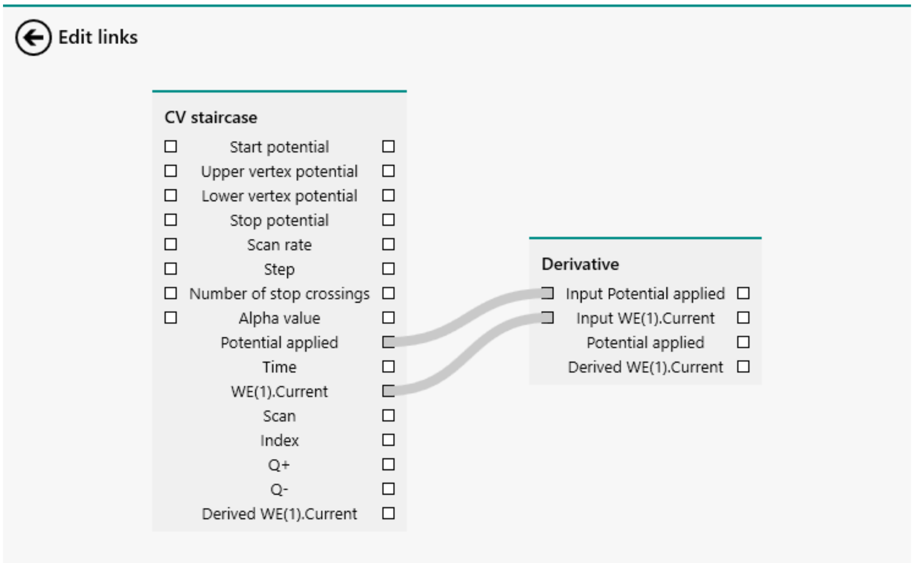
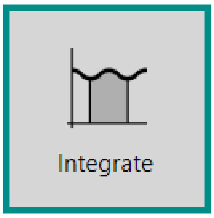


Figure 416 The anchoring points for linking the Derivative command

The command uses the two input signals to calculate the derivative of the second signal versus the first signal.

7.8.5 Integrate

	This command can be used to integrate a curve and determine the area.

The details of the command properties of the **Integrate** command are shown in *Figure 417*.

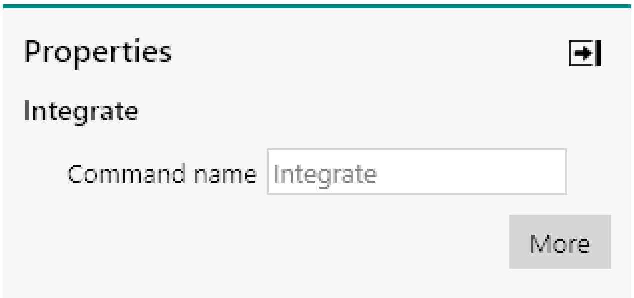


Figure 417 The properties of the Integrate command

The following properties are available:

- **Command name:** a user-defined name for the command.

This command needs to be *linked* to sourced data *(see Chapter 10.13, page 657)*. The **Integrate** command provides two input anchoring points and four output anchoring points *(see Figure 418, page 363)*.

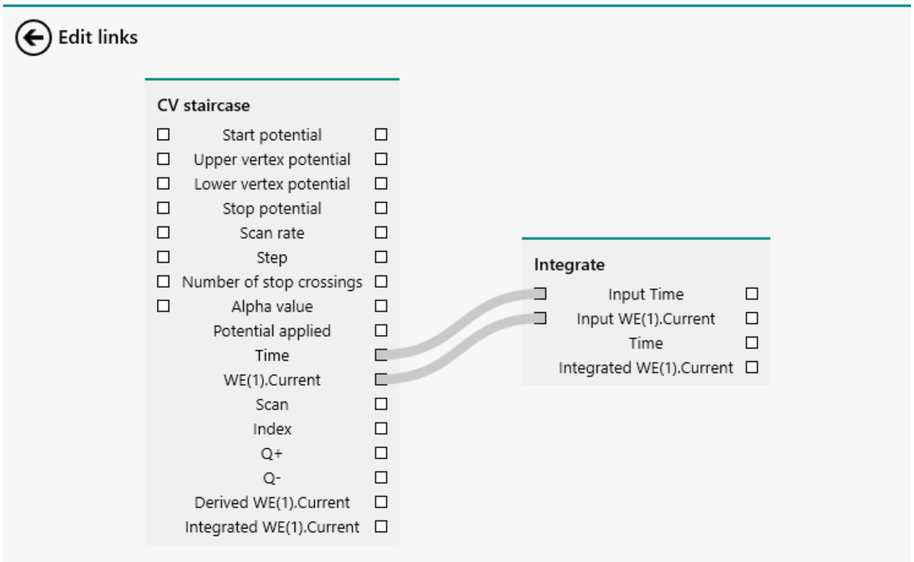
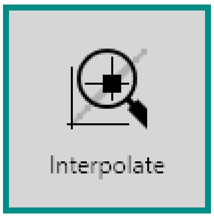


Figure 418 The anchoring points for linking the Integrate command

The command uses the two input signals to calculate the integral of the second signal versus the first signal.

7.8.6 Interpolate

	This command can be used to Interpolate measured data and determine a value at a user-specified location by linear interpolation.

The details of the command properties of the **Interpolate** command are shown in *Figure 419*.

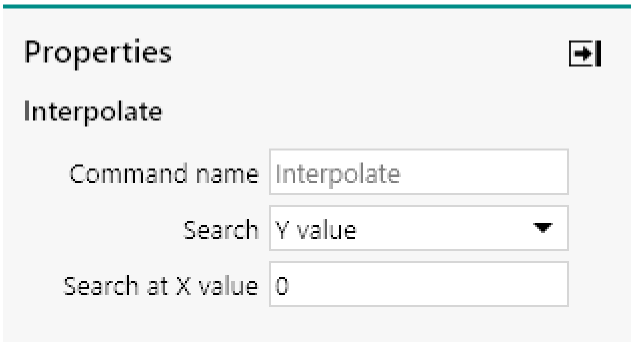
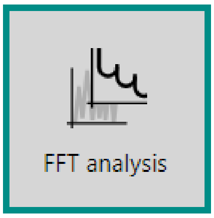


Figure 419 The properties of the Interpolate command

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Search:** a drop-down list allowing the selection of the value to search for (Y value or X value). By default, the command searches for a Y value.
- **Search at X value/Search at Y value:** the location used by the **Interpolate** command. This property is automatically adjusted depending on the **Search** drop-down list.

7.8.7 FFT analysis

	This command can be used to perform a Fast Fourier transformation of source data. Time domain information is converted into frequency domain information.

The details of the command properties of the **FFT analysis** command are shown in *Figure 420*.

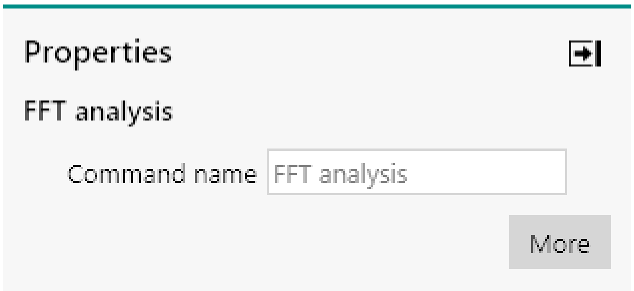


Figure 420 The properties of the FFT analysis command

The following properties are available:

- **Command name:** a user-defined name for the command.

This command needs to be *linked* to sourced data *(see Chapter 10.13, page 657)*. The **FFT analysis** command provides two input anchoring points and six output anchoring points *(see Figure 421, page 365)*.

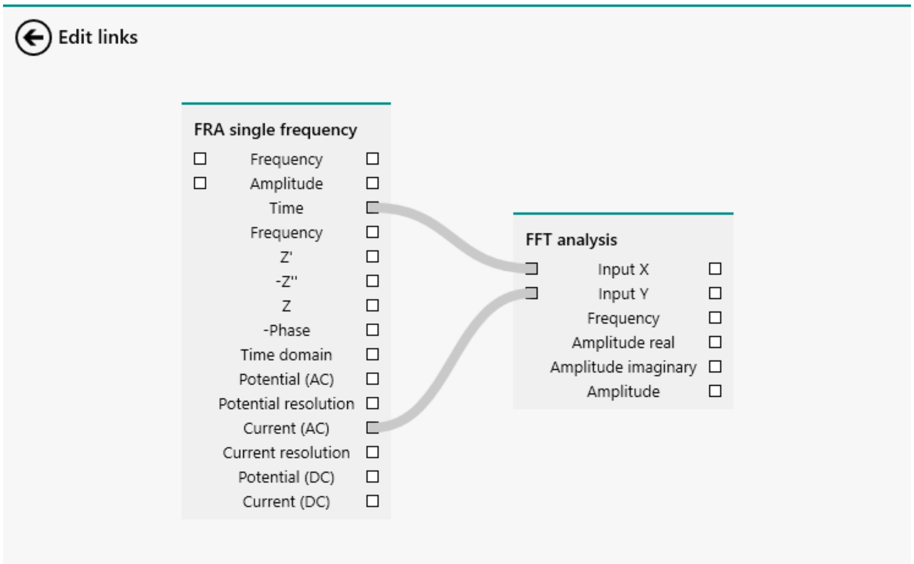


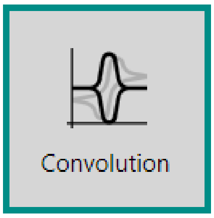
Figure 421 The anchoring points for linking the FFT analysis command

The command uses the two input signals to transform the time domain data to frequency domain data. The frequency, amplitude as well as the real and imaginary parts of the amplitude are returned.

CAUTION

The **FFT analysis** is intended to be used on source data formatted with the *Time* signal the X data. When another signal is used, the **FFT analysis** command will be executed but the *Frequency* signal calculated by the command will no longer be an actual frequency.

7.8.8 Convolution



This command can be used to apply the convolution analysis on the measured data or to integrate this analysis technique in the procedure.

NOTE

This command can only be used on measurements containing the *Time* and *WE(1).Current* signals.

The **Convolution** command can be used in six different modes, which can be selected using the provided drop-down list *(see Figure 422, page 366)*:

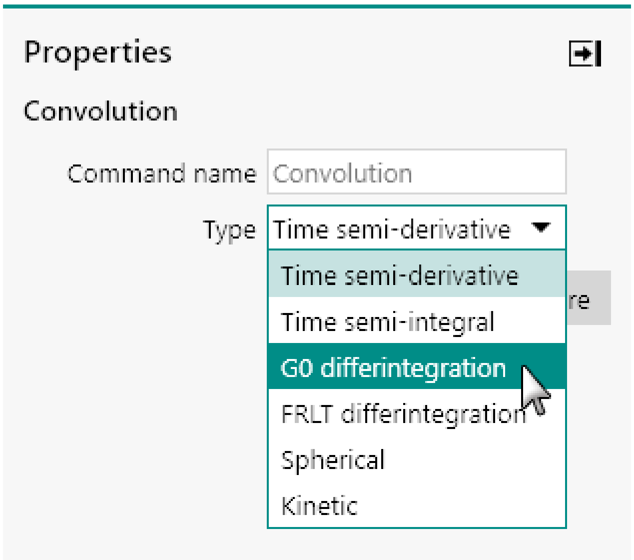


Figure 422 Six modes are provided by the Convolution command

1. Time semi-derivative (default mode)
2. Time semi-integral
3. G0 differintegration
4. FRLT differintegration
5. Spherical convolution
6. Kinetic convolution

NOTE

For a general description of the use of the convolution methods in electrochemistry, we refer the reader to the literature.

7.8.8.1 Time semi-derivative

The following properties are available when the command is used in the *Time semi-derivative* mode *(see Figure 423, page 367)*:

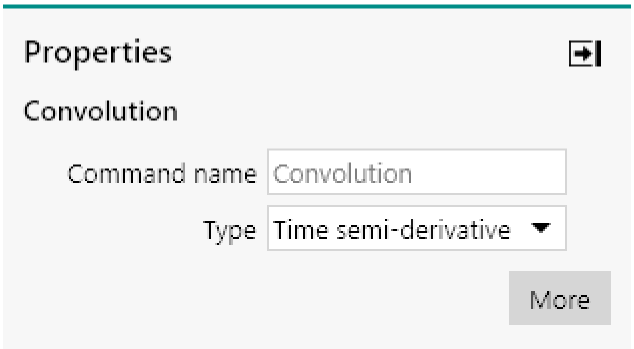


Figure 423 Time semi-derivative mode properties

- **Command name:** a user-defined name for the command.

The time semi-derivative algorithm uses a semi-derivative transformation of a time dependent function, *f(t)*, according to:

$$\frac{d^{1/2}}{dt^{1/2}} f(t)$$

7.8.8.2 Time semi-integral

The following properties are available when the command is used in the *Time semi-integral* mode *(see Figure 424, page 367)*:

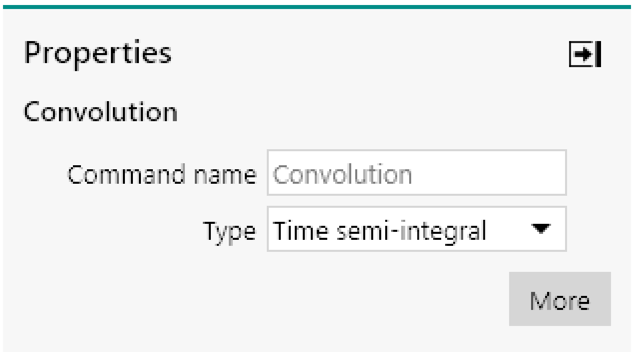


Figure 424 Time semi-integral mode properties

- **Command name:** a user-defined name for the command.

The time semi-integral algorithm uses a semi-integral transformation of a time dependent function, *f(t)*, according to:

$$\frac{d^{-1/2}}{dt^{-1/2}} f(t)$$

7.8.8.3 G0 differintegration (Grünwald-0)

The following properties are available when the command is used in the *G0 differintegration* mode *(see Figure 425, page 368)*:

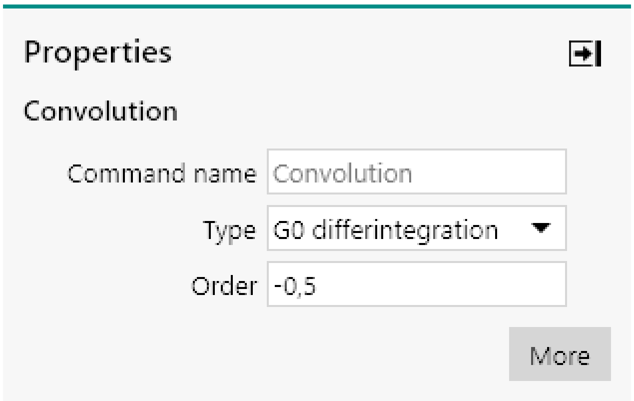


Figure 425 G0 differintegration mode properties

- **Command name:** a user-defined name for the command.
- **Order:** the order used in the G0 differintegration algorithm (default: -0.5).

The G0 differintegration algorithm can be used to carry out differintegration to any user-defined order. Specific *Order* values provide a mathematical equivalence with other transformations:

- For an *Order* value of 1, the operation is the equivalent of a derivative.
- For an *Order* value of -1, the operation is the equivalent of an integration.
- For an *Order* value of 0.5, the operation is the equivalent of a time semi-derivative method.
- For an *Order* value of -0.5, the operation is the equivalent of a time semi-integral method.

Error in results increases with the length of the interval and accumulates, i.e. error in latter points is larger than in earlier ones. Important advantage is that this algorithm does not require the value of the function for $t = 0$, which makes it very well suited for transformation of chronoamperometric data, where $i_{t \to 0} = 0$. The disadvantage of the algorithm is that the total number of operations is proportional to the square of the number of data points, so calculation time grows fast with the length of the data set. The fundamentals of this algorithm are described in Oldham KB, J. Electroanal. Chem. 121 (1981) 341-342.

7.8.8.4 FRLT differintegration (Fast Riemann-Liouville Transform)

The following properties are available when the command is used in the *FRLT differintegration* mode *(see Figure 426, page 369)*:

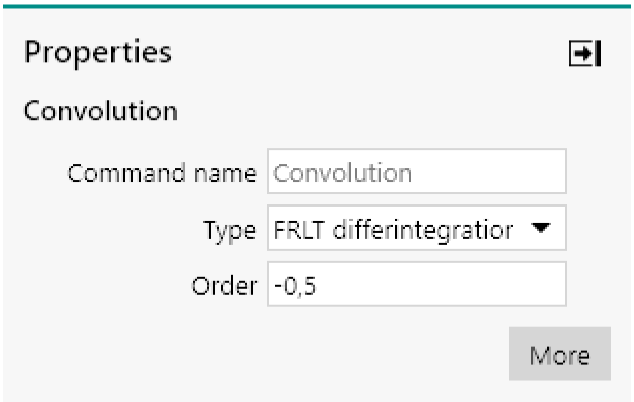


Figure 426 FRLT differintegration mode properties

- **Command name:** a user-defined name for the command.
- **Order:** the order used in the FRLT differintegration algorithm (default: -0.5).

The FRLT differintegration algorithm this is a fast, approximate algorithm based on a recursive digital filter. It is best suited for differintegration *Order* value in the range of 0.0...-0.5 (up to semi-integration). It is less precise than the **G0 differintegration** algorithm, but the number of operations is linearly related to the number of data points. For details refer to Pajkossy T, Nyikos L, J. Electroanal. Chem. 179 (1984) 65-69.

7.8.8.5 Spherical

The following properties are available when the command is used in the *Spherical convolution* mode *(see Figure 427, page 369)*:

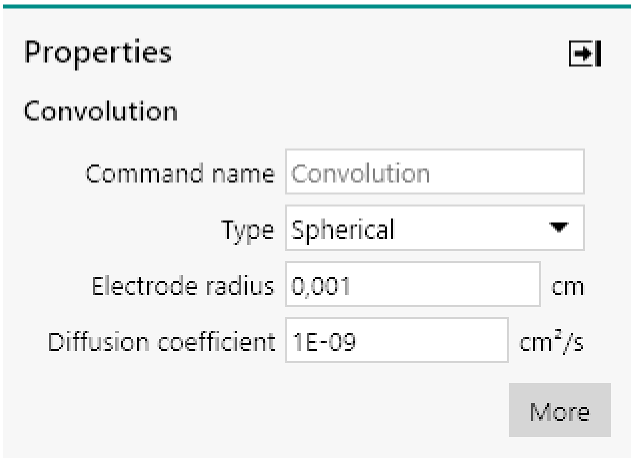


Figure 427 Spherical convolution mode properties

- **Command name:** a user-defined name for the command.

- **Electrode radius:** the radius of the electrode, in cm.
- **Diffusion coefficient:** the diffusion coefficient, in cm²/s.

is used to carry out convolution of the data measured using a spherical electrode and staircase potential waveform. Values of the diffusion coefficient and the electrode radius are necessary. Details of the algorithm can be found in S.O. Englblom, K.B. Oldham, Anal. Chem. 62 (1990) 625-630.

7.8.8.6 Kinetic

The following properties are available when the command is used in the *Kinetic convolution* mode *(see Figure 427, page 369)*:

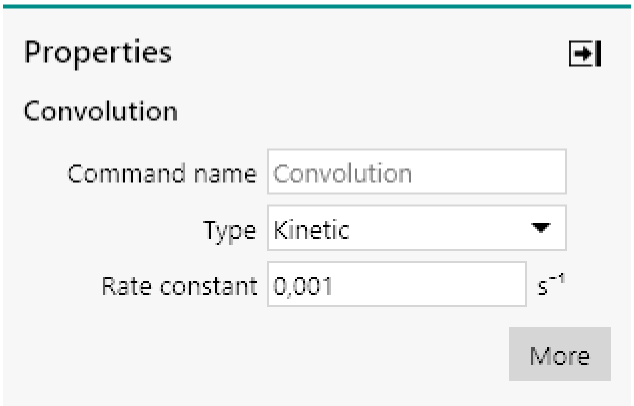
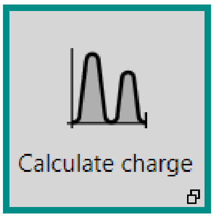


Figure 428 Kinetic convolution mode properties

- **Command name:** a user-defined name for the command.
- **Rate constant:** the rate constant of the chemical reaction, in $^{s-1}$.

The kinetic convolution algorithm carries out kinetic convolution according to F.E. Woodard, R.D. Goodin, P.J. Kinlen, Anal. Chem. 56 (1984) 1920-1923. This convolution requires the value of the rate constant of irreversible homogeneous follow-up reaction (ECi mechanism).

7.8.9 Calculate charge

	This command can be used to calculate the charge by integrating the measured current against time. The total charge is reported in Coulomb (C).

The details of the command properties of the **Calculate charge** command are shown in *Figure 429*.

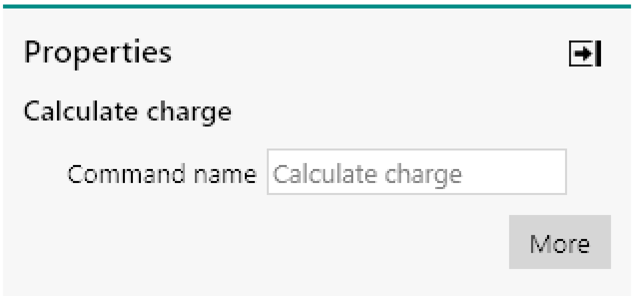
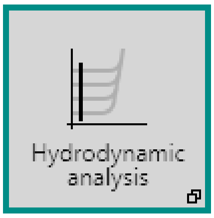


Figure 429 The properties of the Calculate charge command

NOTE

This command can only be used on measurements containing the *Time* and *WE(1).Current* signals.

7.8.10 Hydrodynamic analysis

	This command can be used to perform a Levich and Koutecký-Levich analysis on hydrodynamic data.

NOTE

This command is intended to be used in combination with a rotating disk or rotating ring disk electrode, controlled by NOVA.

The details of the command properties of the **Hydrodynamic analysis** command are shown in *Figure 430*:

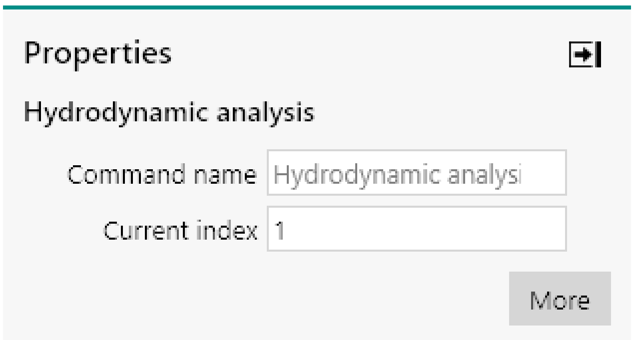


Figure 430 The property of the Hydrodynamic analysis command

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Current index:** the index of the current value used in the by the command. The index of a current value located in the mass transport-limited region should be specified for the Levich analysis and the index of a current value located in the mixed kinetic-mass transported region should be specified for the Koutecký-Levich analysis. For all rotation rates used in the procedure, the current value at the specified index will be used.

The rotation of the electrode creates a convective drag from the bulk of the solution towards the surface of the electrode, resulting in a mixed control of mass transport, involving a convective part which depends on the square root of the angular frequency of the electrode and diffusion layer which also depends on this property. Under these experimental conditions, the limiting current values, i_l and kinetic current i_k, are related to the rotation rate of the working electrode according to the Levich equation and Koutecký-Levich equation:

$$i_l = 0.62 \cdot AnFD^{2/3}\nu^{-1/6}C^{\infty}\sqrt{\omega}$$

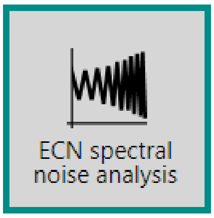
$$\frac{1}{i} = \frac{1}{i_k} + \frac{1}{0.62 \cdot AnFD^{2/3}\nu^{-1/6}C^{\infty}\sqrt{\omega}}$$

Where A is the geometric area of the electrode, in cm², n is the number of electrons involved in the electrochemical reaction, F is the Faraday constant, D is the diffusion coefficient of the electroactive species, in cm²/s, ν is the kinematic viscosity in cm²/s and $\sqrt{\omega}$ is the square root of the angular frequency of the rotating electrode, in $(rad/s)^{1/2}$.

NOTE

The **Hydrodynamic analysis** command automatically carries out two linear regressions using the **Regression** command.

7.8.11 ECN spectral noise analysis

	This command can be used to analyze electrochemical noise measurements (ECN).

The **ECN spectral noise analysis** command can be used in two different modes, which can be selected using the provided drop-down list *(see Figure 431, page 373)*:

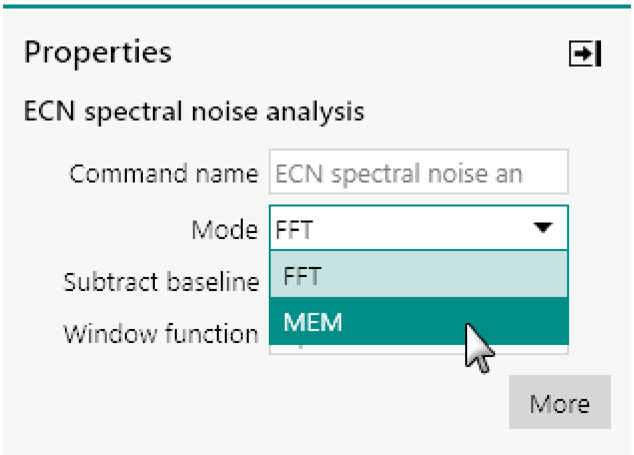


Figure 431 Two modes are provided by the ECN spectral noise analysis command

- **FFT:** a spectral noise analysis that uses the *Fast Fourier Transform* method.
- **MEM:** a spectral noise analysis that uses the *Maximum Entropy* method.

NOTE

This command can only be used on measurements containing the *Time*, *ECN(1).Potential* and *WE(1).Current* signals.

Electrochemical noise data is generally analyzed by computing the spectral density of the measured data. This can be achieved by transforming the time domain information to a frequency domain spectrum, using the **Fast Fourier Transformation (FFT)** or the **Maximum Entropy Method (MEM)**.

Traditional time domain to frequency domain transformation assumes that the data outside of the measured time segment is either zero or that the data in this segment repeats periodically. This hypothesis is not valid for electrochemical noise data. In order to satisfy these requirements and to avoid edge effects in the data, it is common practice to apply a **window function** on the time domain data. This calculation involves the multiplication of the time domain data by a function which is zero at the extremes of the time domain data and rises smoothly to unity value in its center.

Alongside the power spectra determined by the transformation of the data into the frequency domain, the ECN spectral noise analysis also calculates the following statistical indicators:

- Noise resistance, R_n.
- Pitting index (or localization index), PI

- Current and potential skewness
- Current and potential kurtosis

The noise resistance, R_i, is given by:

$$R_n = \frac{\sigma_V}{\sigma_i}$$

Where σ_V and σ_i are the standard deviations of the measured potential and current, respectively. The value of the noise resistance is reported in Ohm.

The pitting index, or localization index, PI, if given by:

$$PI = \frac{\sigma_i}{i_{RMS}} = \sqrt{\frac{\sum_{j=1}^{N}\left(i_j - \bar{i}\right)^2}{\sum_{j=1}^{N} i_j^2}}$$

Where i_{RMS} is the root mean squared value of the measured current. The pitting index can be between 0 and 1. A value close to 0 is observed for systems in which the measured current values show only small deviation with respect to the average current value. On the other hand, the pitting index will be close to 1 when the individual current values are significantly deviating from the average current value. This value is therefore an indication of the distribution of the current values recorded during an electrochemical noise experiment.

Skewness and kurtosis are additional indicators calculated according to the following equations, respectively:

$$\frac{1}{N}\sum_{j=1}^{N}\left(\frac{X_j - \bar{X}}{\sigma}\right)^3$$

$$\frac{1}{N}\sum_{j=1}^{N}\left(\frac{X_j - \bar{X}}{\sigma}\right)^4$$

7.8.11.1 Fast Fourier Transform (FFT)

The following properties are available when the command is used in the *FFT* mode *(see Figure 432, page 375)*:

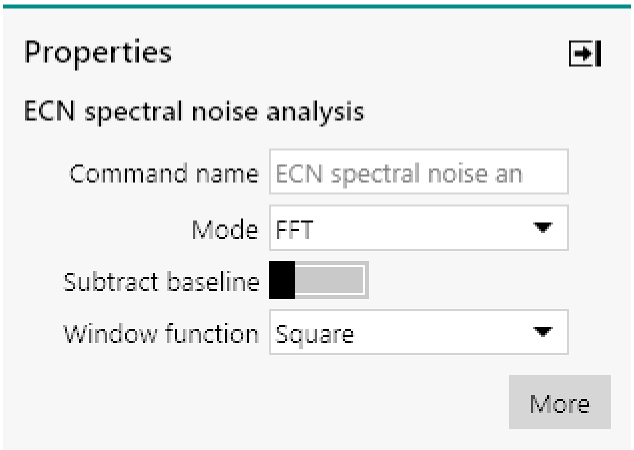


Figure 432 FFT mode properties

- **Command name:** a user-defined name for the command.
- **Subtract baseline:** a toggle which can be used to enable or disable baseline subtraction. When this property is enabled, a linear regression is used to subtract the baseline from the measured potential or current values.
- **Window function:** defines the type of windowing function used for the FFT algorithm, using the provided drop-down list. The default function is the Square function.

NOTE

For a detailed description of the *Window functions* used in NOVA, the reader is invited to refer to W. H. Press, S. A. Teukolsky, W. T. Vetterling, B. P. Flannery, Numerical Recipes – The Art of Scientific Computing, 3rd edition, Cambridge University Press, 2007.

7.8.11.2 Maximum Entropy Method (MEM)

The following properties are available when the command is used in the *MEM* mode *(see Figure 433, page 376)*:

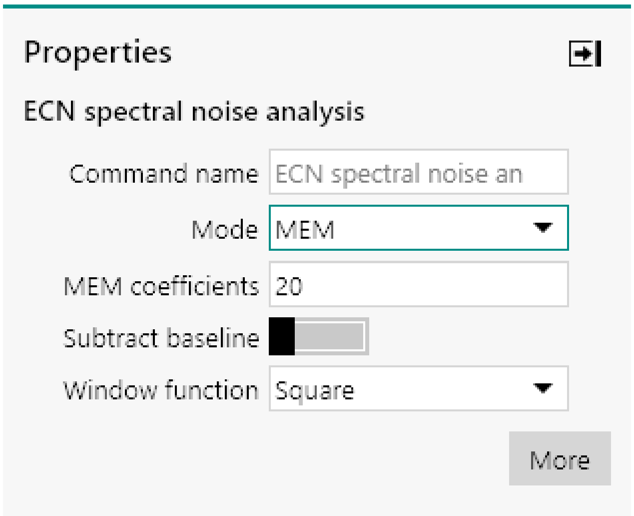


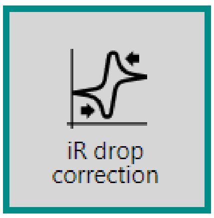
Figure 433 MEM mode properties

- **Command name:** a user-defined name for the command.
- **MEM coefficients:** specifies the number of coefficients to be used in the MEM algorithm. The default value is 20.
- **Subtract baseline:** a toggle which can be used to enable or disable baseline subtraction. When this property is enabled, a linear regression is used to subtract the baseline from the measured potential or current values.
- **Window function:** defines the type of windowing function used for the MEM algorithm, using the provided drop-down list. The default function is the Square function.

NOTE

For a detailed description of the *Window functions* used in NOVA, the reader is invited to refer to W. H. Press, S. A. Teukolsky, W. T. Vetterling, B. P. Flannery, Numerical Recipes – The Art of Scientific Computing, 3rd edition, Cambridge University Press, 2007.

7.8.12 iR drop correction

	This command can be used to correct measured data for ohmic drop losses.

The command properties of the **iR drop correction** command are shown in *Figure 434*:

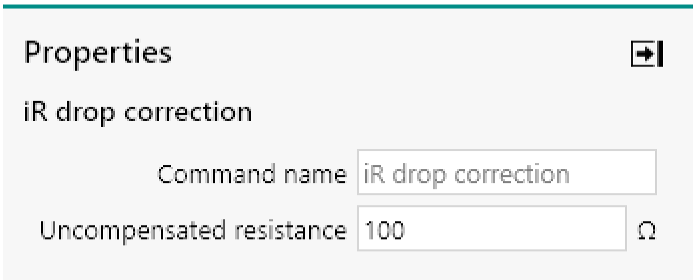


Figure 434 The iR drop correction property

The following properties are available:

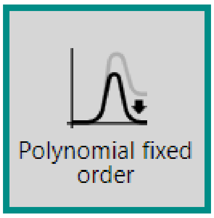
- **Command name:** a user-defined name for the command.
- **Uncompensated resistance:** the value of the uncompensated resistance using the correction, in Ohm.

Using the specified value, the command recalculates a potential scale in using the formula:

$$E_{corrected} = E - iR_u$$

Where $E_{corrected}$ is the recalculated potential, E is the measured potential, i is the measured current and R_u is the specified uncompensated resistance.

7.8.13 Baseline correction

	This command can be used to correct a data set by subtracting a user-defined baseline.

The **Baseline correction** command can be used in four different modes, which can be selected using the provided drop-down list *(see Figure 435, page 378)*:

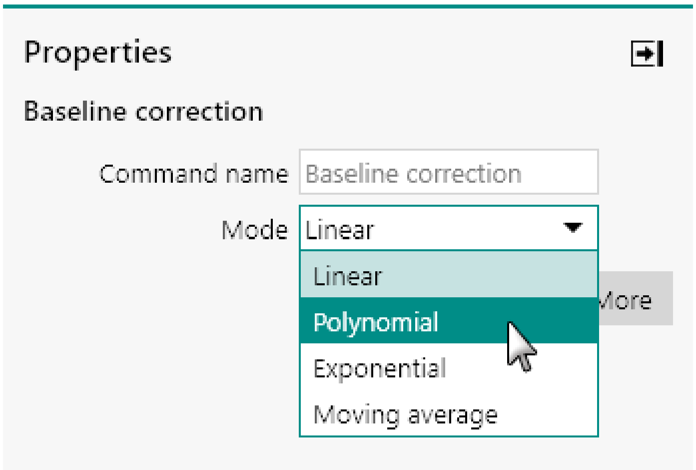


Figure 435 Four modes are provided by the Baseline correction command

1. Linear (default mode)
2. Polynomial
3. Exponential
4. Moving average

NOTE

The **Baseline correction** command description in the procedure editor is dynamically adjusted in function of the specified mode.

For the *Linear*, *Polynomial* and *Exponential* mode, the points defining the location of the baseline used in the correction can be specified using the More button *(see Figure 436, page 378)*.

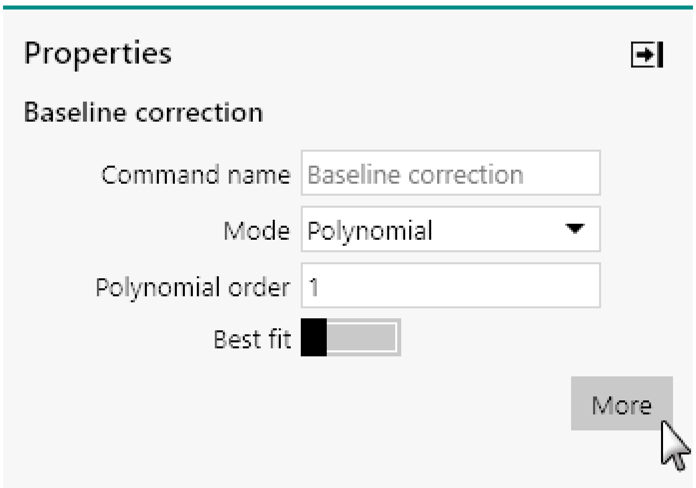


Figure 436 The points defining the baseline are specified in a dedicated editor

A table will be displayed in a new screen *(see Figure 437, page 379)*.

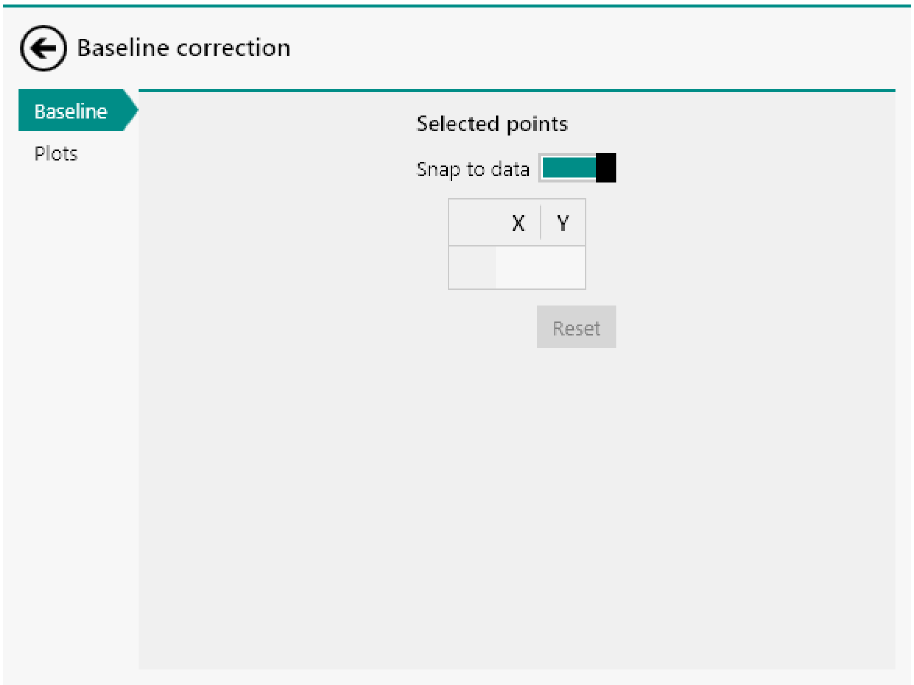


Figure 437 The selected point table

Using the provided editor, it is possible to define the location of the two or more points. The location of each point is defined by specifying a X and Y coordinate *(see Figure 438, page 379)*.

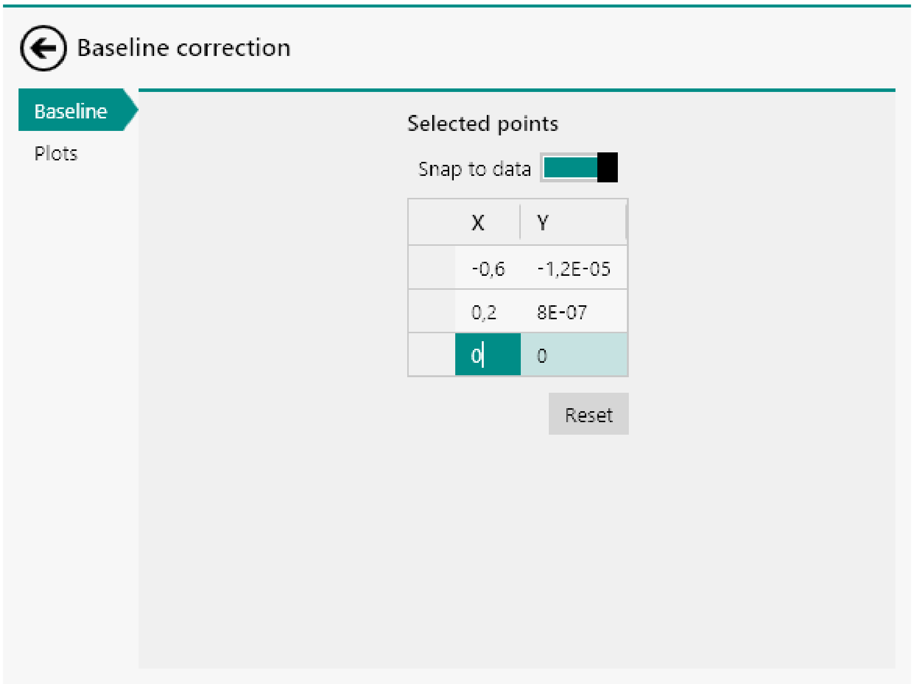


Figure 438 Specifying the baseline points

Depending on mode of the **Baseline correction**, the following number of points need to be defined:

- **Linear mode:** two or more points are required to define a linear baseline.
- **Polynomial:** *n+1* or more points are required to define a polynomial based of order *n*.
- **Exponential:** two or more points are required to define an exponential baseline.

The *Snap to data* toggle can be used to force the baseline to be snapped to the nearest data point in the source data. If this toggle is on, then the Y coordinate will be ignored and the data point nearest to the specified X abscissa will be used.

Clicking the Reset button clears the whole table.

7.8.13.1 Linear

The following properties are available when the command is used in the *Linear* mode *(see Figure 439, page 380)*:

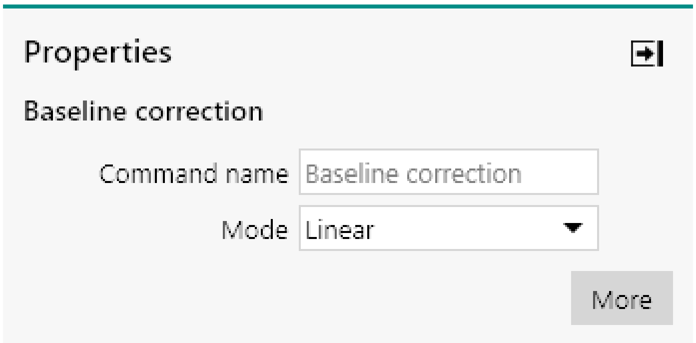


Figure 439 The Linear properties

- **Command name:** a user-defined name for the command.

The points defining the location of the baseline used in the correction can be specified using the More button *(see Figure 436, page 378)*.

7.8.13.2 Polynomial

The following properties are available when the command is used in the *Polynomial* mode *(see Figure 440, page 381)*:

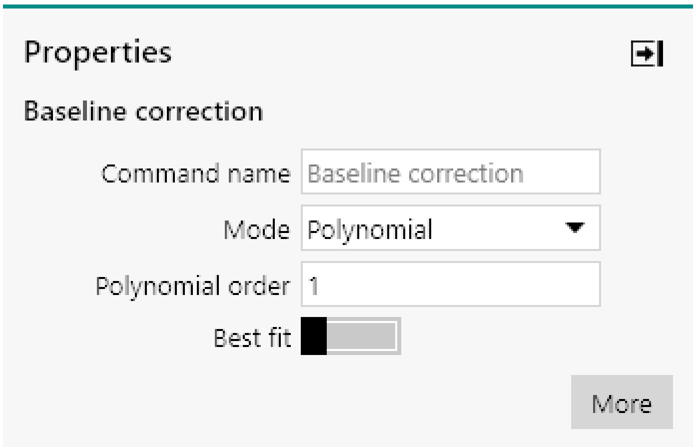


Figure 440 The Polynomial properties

- **Command name:** a user-defined name for the command.
- **Polynomial order:** defines the order of the polynomial baseline (default 1).
- **Best fit:** specifies if the best fit should be used in the polynomial baseline, using the provided toggle. When this toggle is off, the specified polynomial order is used. When this toggle is on, the polynomial order providing the smallest χ^2 (Chi-squared) is automatically selected by the software.

The points defining the location of the baseline used in the correction can be specified using the More button *(see Figure 436, page 378)*.

7.8.13.3 Exponential

The following properties are available when the command is used in the *Exponential* mode *(see Figure 441, page 381)*:

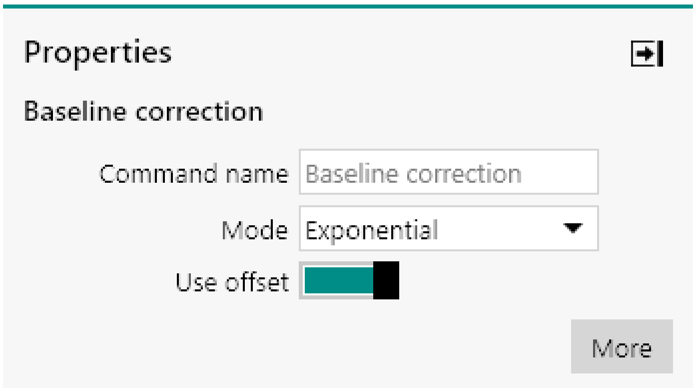


Figure 441 The Exponential properties

- **Command name:** a user-defined name for the command.

- **Use offset:** defines if an offset should be used in the exponential baseline correction, using the provided toggle. Depending on this toggle, the following equations are used:
  - **Use offset off:** performs a baseline correction using the equation $\mathbf{y = be^{cx}}$.
  - **Use offset on:** performs a baseline correction using the equation $\mathbf{y = a + be^{cx}}$.

The points defining the location of the baseline used in the correction can be specified using the More button *(see Figure 436, page 378)*.

7.8.13.4 Moving average

The following properties are available when the command is used in the *Moving average* mode *Figure 442*:

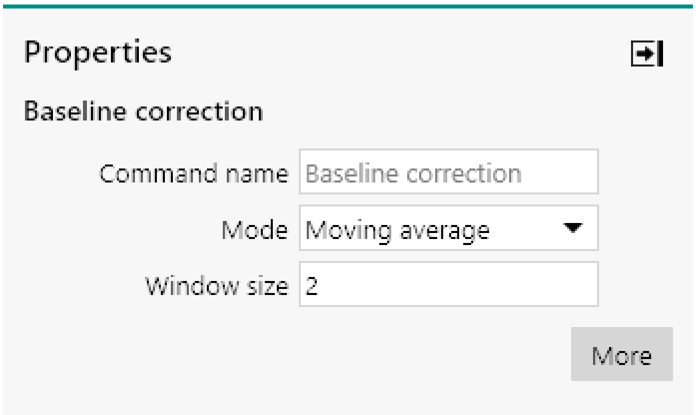


Figure 442 The Moving average properties

- **Command name:** a user-defined name for the command.
- **Window size:** defines the number of points in the moving average window (default: 2).

The moving average baseline correction performs the following steps:

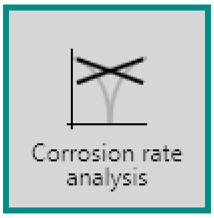
1. The source data is grouped into segments of **n** points; where **n** corresponds to the **Window size** property.
2. The average value of each segment is calculated.
3. The source data is reduced from **m** data points to **m/n** averages.
4. Each $\mathbf{i^{th}}$ average value is compared to the average value of its immediate neighboring values, at **i–1** and **i+1**.
   a. For positive going sweeps, if the $\mathbf{i^{th}}$ average value is higher than the average value of the averages at **i–1** and **i+1**, then the $\mathbf{i^{th}}$ average value is replaced by the average value of the averages at **i–1** and **i+1**.
   b. For negative going sweeps, if the $\mathbf{i^{th}}$ average value is lower than the average value of the averages at **i–1** and **i+1**, then the $\mathbf{i^{th}}$ average value is replaced by the average value of the averages at **i–1** and **i+1**.

5. Step **3** is repeated for a maximum of 1000 iterations or until the baseline does not change anymore.
6. The baseline is interpolated from **m/n** final averages to the original **m** data points.
7. The baseline is subtracted from the source data.

CAUTION

The moving average mode of the baseline correction command can only be used with data which is presented on a **non-reversing X axis**. This means that it cannot be used on a cyclic voltammetry measurement.

7.8.14 Corrosion rate analysis

	This command can be used to determine the corrosion rate and convert the exchange current density in amount of material corroded per year.

The **Corrosion rate analysis** command can be used in two different modes, which can be selected using the provided drop-down list *(see Figure 443, page 383)*:

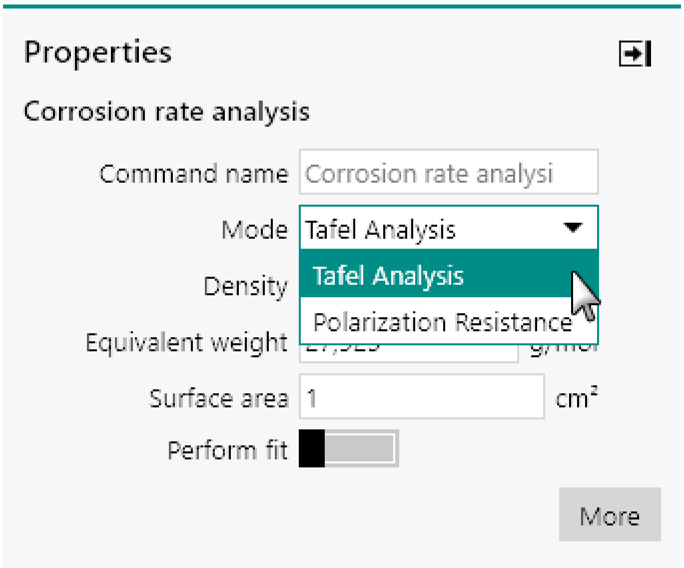


Figure 443 Two modes are provided by the Corrosion rate analysis command

1. Tafel Analysis (default mode)
2. Polarization Resistance

7.8.14.1 Tafel Analysis

The following properties are available when the command is used in the *Tafel Analysis* mode *(see Figure 444, page 384)*:

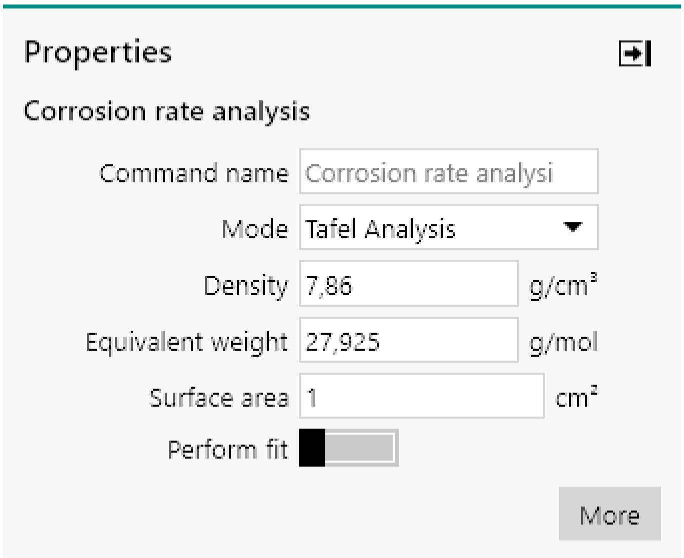


Figure 444 Tafel Analysis mode properties

- **Command name:** a user-defined name for the command.
- **Density:** specifies the density of the sample in g/cm^3.
- **Equivalent weight:** defines the equivalent weight of the sample in g/mol of exchanged electrons.
- **Surface area:** defines the area of the sample, in cm^2.
- **Perform fit:** specifies if the data should be fitted, using the provided  toggle. Depending on this toggle, the corrosion rate analysis command carries out the following analysis:
  - **Perform fit off:** only the Tafel slopes are determined and the corrosion rate analysis is carried out using the results of the Tafel slope analysis.
  - **Perform fit on:** the Tafel slope analysis is carried out and the data is fitted using the **Butler-Volmer** equation. The corrosion rate analysis is carried out based on the results of the fit.

The Butler-Volmer equation is given by:

$$i = i_{corr}\left(e^{2.303\frac{E-E_{corr}}{b_a}} - e^{2.303\frac{E-E_{corr}}{b_c}}\right)$$

Where i is the measured current, i_{corr} is the corrosion exchange current, E is the applied potential, E_{corr} is the corrosion potential and b_a and b_c are the Tafel slopes, in V/decade, respectively.

The **Corrosion rate analysis** command requires the definition of four points, defining the location of the linear parts of the anodic and cathodic branches of the Tafel plot, respectively. These points are defined using the More button *(see Figure 445, page 385)*.

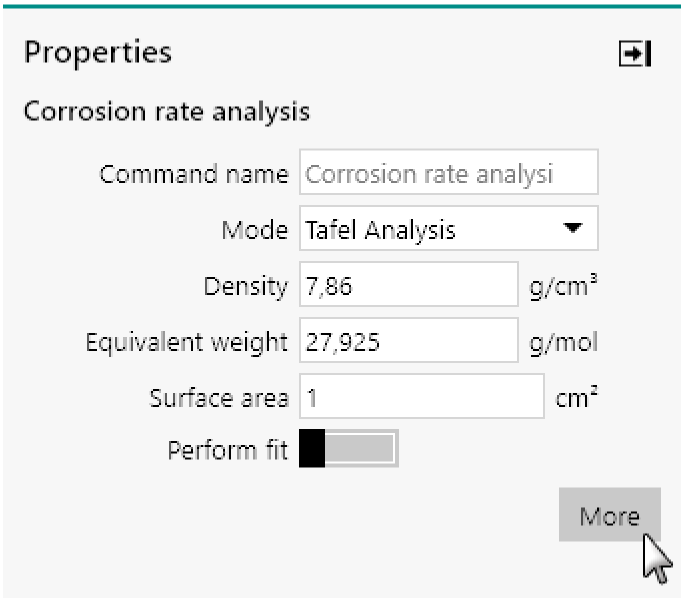


Figure 445 The points defined the linear parts of the Tafel plot are specified in a dedicated editor

A table will be displayed in a new screen *(see Figure 446, page 385)*.

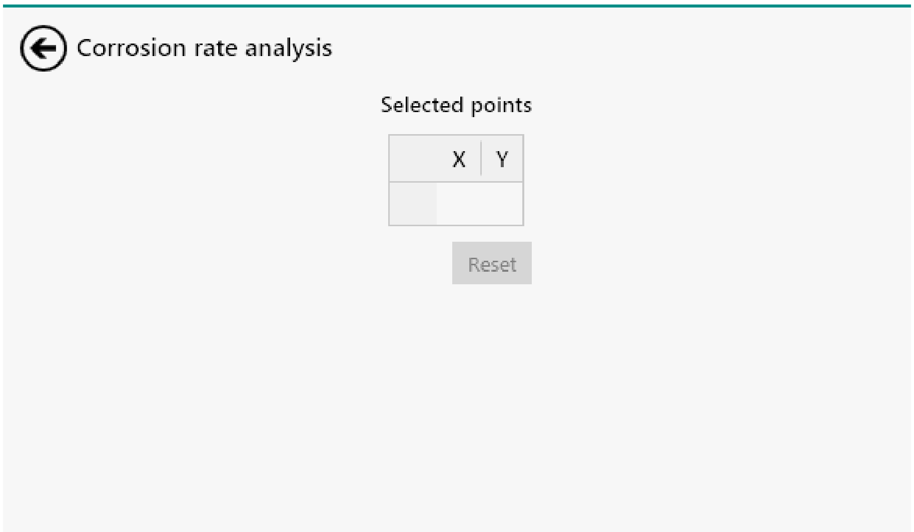


Figure 446 The Selected point table

Using the provided editor, it is possible to define the location of the four points, two for the anodic branch and two for the cathodic branch of the Tafel plot. The location of each point is defined by specifying a X and Y coordinate *(see Figure 447, page 386)*.

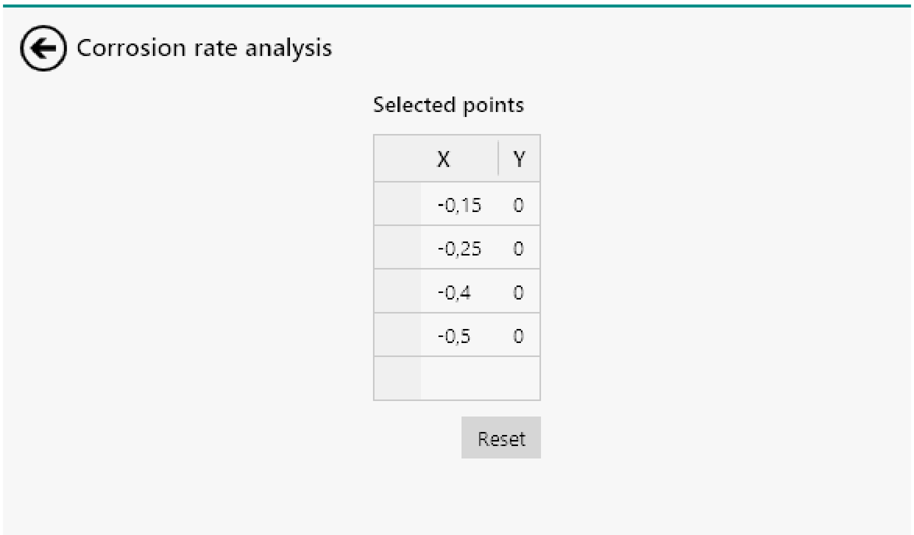


Figure 447 Specifying the four points

NOTE

If needed, the location of these points can be finetuned after the measurement is finished *(see Chapter 12.8, page 776)*.

NOTE

NOVA will automatically select the data points closest to the specified points in the table when the command is executed.

NOTE

The Y coordinates can be set to 0.

Clicking the Reset button clears the whole table.

7.8.14.2 Polarization Resistance

The following properties are available when the command is used in the *Polarization Resistance* mode *(see Figure 448, page 387)*:

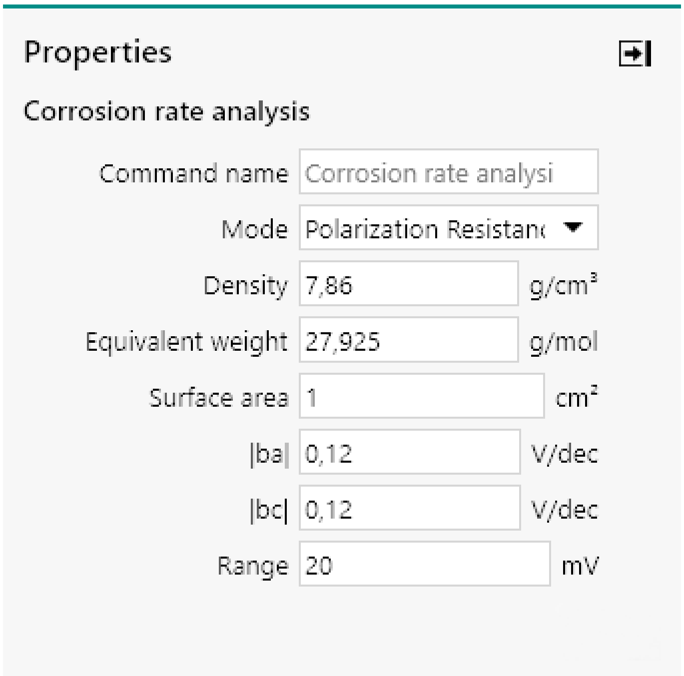


Figure 448 Polarization resistance mode properties

- **Command name:** a user-defined name for the command.
- **Density:** specifies the density of the sample in g/cm^3.
- **Equivalent weight:** defines the equivalent weight of the sample in g/mol of exchanged electrons.
- **Surface area:** defines the area of the sample, in cm^2.
- **|ba|:** defines the absolute value of the anodic Tafel slope value, in V/decade of current.
- **|bc|:** defines the absolute value of the cathodic Tafel slope value, in V/decade of current.
- **Range:** defines the potential range, in mV, around the observed corrosion potential, in which the analysis is carried out. The specified value will be used on both sides of the corrosion potential.

The *Polarization Resistance* mode uses the Stern-Geary equation to the determine the corrosion current, i_{corr}, according to:

$$i_{corr} = \frac{\frac{|b_a| \cdot |b_c|}{2.303\left(|b_a| + |b_c|\right)}}{R_p}$$

Where b_a and b_c are the specified Tafel slopes, in absolute value, and R_p is the inverted slope of the linear regression carried out in the specified Range around the observed corrosion potential.

NOTE

The Polarization Resistance analysis method is based on M. Stern, A. L. Geary, JECS Vol. 104, No. 1, 56-63, 1957.

7.9 Analysis - impedance

Analysis - impedance commands can be used to perform data analysis on measured impedance data or to integrate data analysis steps in a procedure.

The available commands are represented by a shortcut icon *(see Figure 449, page 388)*.

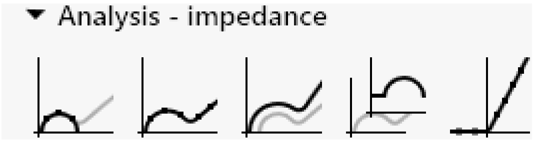
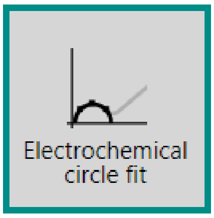


Figure 449 The Analysis - impedance commands

The following commands are available:

- **Electrochemical circle fit:** a command which can be used to quickly fit a semi-circle in a Nyquist plot using a *R(RQ)* equivalent circuit *(see Chapter 7.9.1, page 388)*.
- **Fit and simulation:** a command which can be used to fit measured impedance data with a user-defined equivalent circuit *(see Chapter 7.9.2, page 390)*.
- **Kramer-Kronig test:** a command which can be used to perform the Kramer-Kronig test on measured impedance data *(see Chapter 7.9.3, page 433)*.
- **Include all FRA data:** a command that can be used to calculate additional values from the measured impedance *(see Chapter 7.9.4, page 436)*.
- **Potential scan FRA data:** a command that can be used to calculate values from measured potential scan FRA data to perform a Mott-Schottky analysis *(see Chapter 7.9.5, page 437)*.

7.9.1 Electrochemical circle fit

	This tool can be used to fit a semi-circle in a Nyquist plot with a R(RQ) equivalent circuit using three user defined points.

The details of the command properties of the **Electrochemical circle fit** command are shown in *Figure 450*.

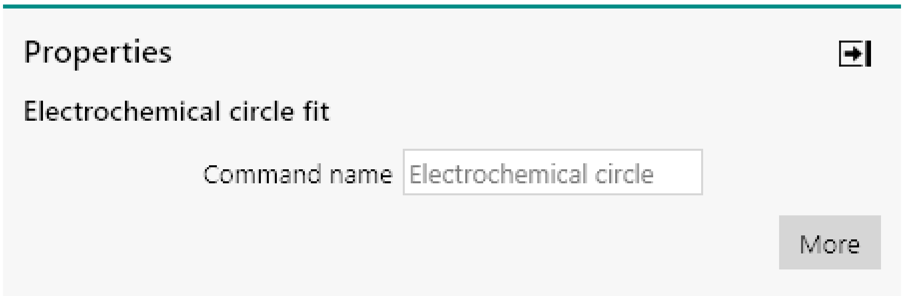


Figure 450 The properties of the Electrochemical circle fit command

The following properties are available:

- **Command name:** a user-defined name for the command.

In order to use this command, three or more point defining the location of the semi-circle in the Nyquist plot need to be defined. These points can be specified using the button *(see Figure 450, page 389)*.

A table will be displayed in a new screen *(see Figure 451, page 389)*.

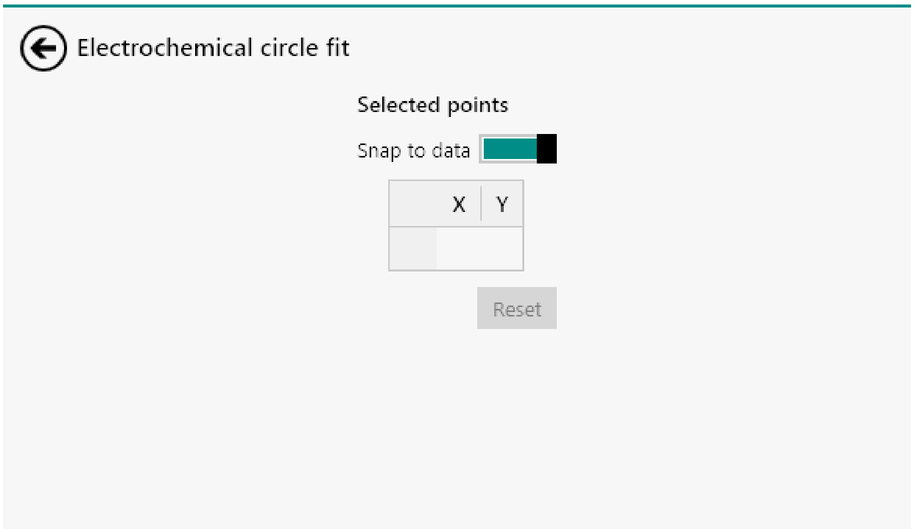


Figure 451 The selected point table

Using the provided editor, it is possible to define the location of three or more points. The location of each point is defined by specifying a X and Y coordinate *(see Figure 452, page 390)*.

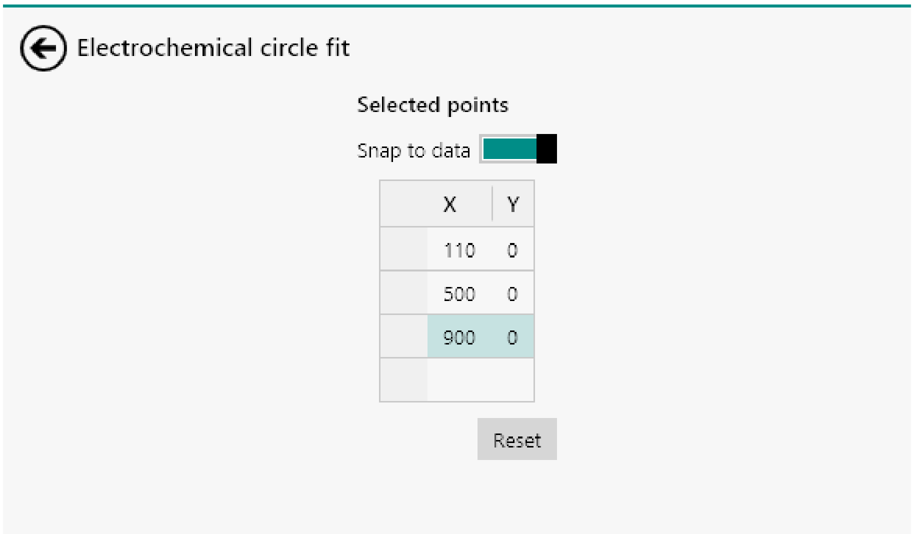
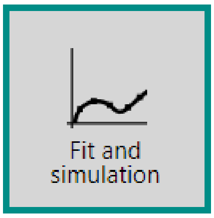


Figure 452   Specifying the points to define the semi-circle

The *Snap to data* toggle can be used to force the points defining the semi-circle to be snapped to the nearest data point in the source data. If this toggle is on, then the Y coordinate will be ignored and the data point nearest to the specified X coordinate.

Clicking the Reset button clears the whole table.

7.9.2 Fit and simulation

	This command allows to use the fit and simulation analysis tool. Measured data can be fitted (or simulated) using a pre-defined equivalent circuit. The equivalent circuit is defined using the dedicated Circuit editor.

The details of the properties of the **Fit and simulation** command are shown in *Figure 453*:

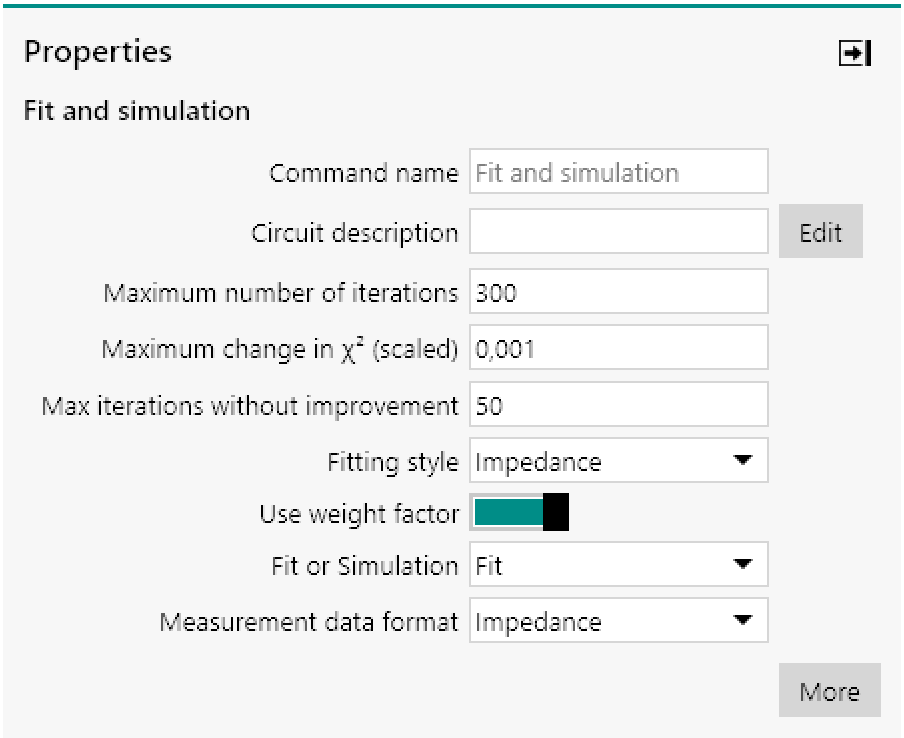


Figure 453 The properties of the Fit and simulation command

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Circuit description:** specifies the equivalent circuit used in the command, either as a properly formatted string or using the dedicated editor which can be accessed by clicking the button.
- **Maximum number of iterations:** defines the number of consecutive calculations used during the fitting calculation. The default value is 300.
- **Maximum change in χ^2 scaled:** defines one of the convergence criteria. The fitting will finish when the absolute change in the χ^2 property (including weight factors) will be lower than this value. The default value is 0.001.
- **Maximum iterations without improvement:** defines a second stop condition for the fitting calculation. This number defines the number of iterations that are allowed during which the χ^2 value does not improve. When this value is reached the fitting calculation is stopped. The default value is 50.
- **Fitting style:** defines whether the calculation should use the impedance or the admittance values during the fit, using the provided dropdown list.

- **Use weight factor:** a toggle which can be used to define whether a weight factor should be used during the calculation. If weight factors are used, each point is multiplied by a weight factor equal to the inverse of the square of the impedance modulus. If this option is not used, the weight factor is the same for each point, i.e. the inverse of the square root of the average of the impedance modulus.
- **Fit or simulation:** defines the calculation method using the provided drop-down list. Using the Fit method, the software will try to find the most suitable values for the parameters of each element defined in the equivalent circuit, starting with initial, user-defined values. The simulation method simply calculates the impedance values for the equivalent circuit, as it is defined by the user.
- **Measurement data format:** defines the type of data of the data, using the provided drop-down list.

The equivalent circuit can be specified either by typing a string in the **Circuit description** field, as shown in *Figure 454*.

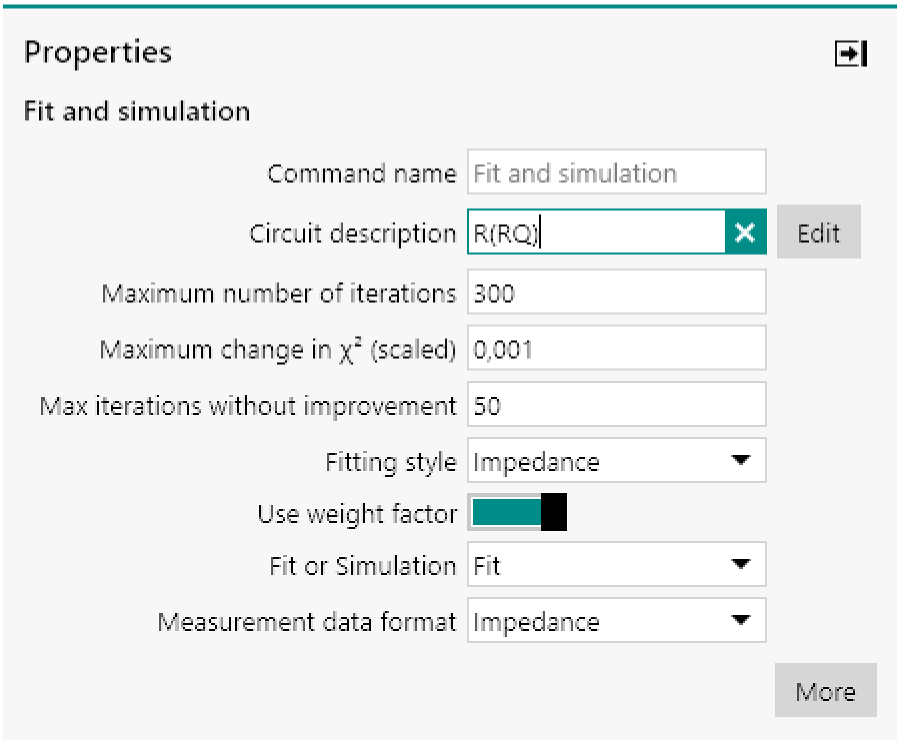


Figure 454 Editing the circuit description

Alternatively, it is possible to define the equivalent circuit in a dedicated editor, by clicking the Edit button *(see Figure 455, page 393)*.

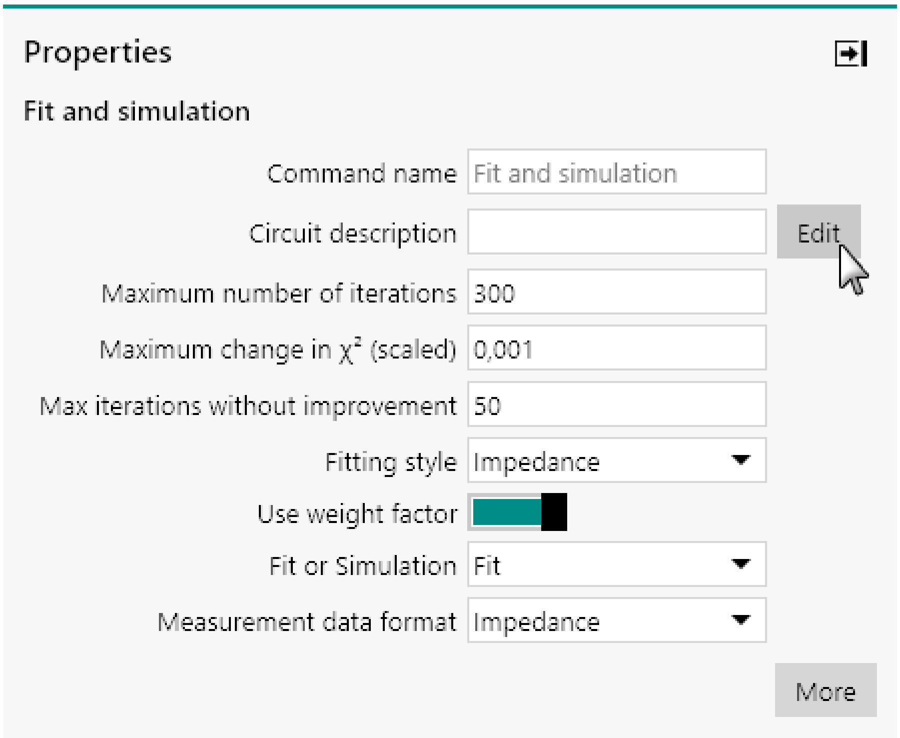


Figure 455 Opening the circuit editor

This opens a new window, in which the equivalent circuit used to fit or simulate the data can be specified *(see Figure 456, page 394)*.

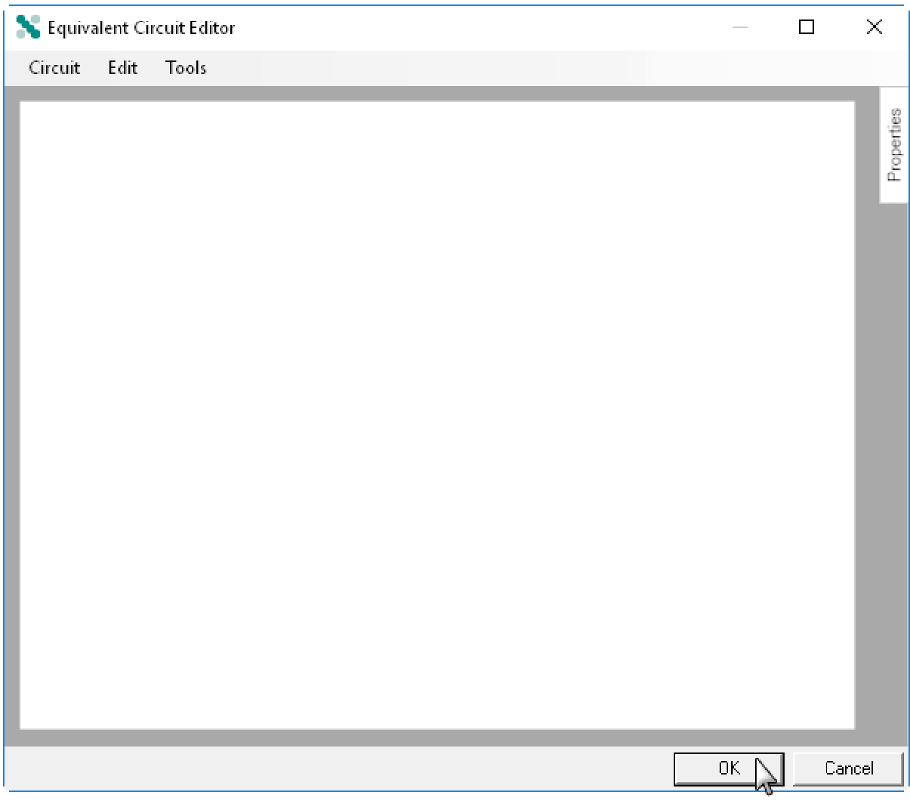


Figure 456 The Equivalent circuit editor

Detailed analysis of the data obtained during an electrochemical impedance measurement is usually performed by fitting the experimental data with an equivalent circuit, based on the Boukamp model. Many circuit elements can be used to fit the experimental data with a model. However, the equivalent circuit must be constructed carefully, since a given experimental data set can be fitted with more than one unique equivalent circuit.

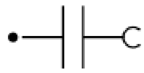
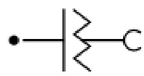
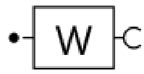
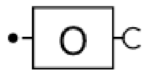
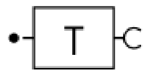
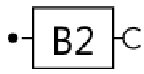
The following tasks can be carried out in the Equivalent circuit editor:

1. Drawing the equivalent circuit using individual circuit elements
2. Generate an equivalent circuit from a CDC string
3. Loading a pre-defined equivalent circuit
4. Importing and exporting equivalent circuits
5. Advanced editing
6. Edit element properties
7. Creating linkable properties
8. Save circuit to Library

7.9.2.1 Circuit elements

The **Fit and simulation** command allows the definition of an equivalent circuit using the elements shown in *Table 12*.

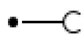
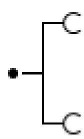
Table 12 Overview of the available equivalent circuit elements

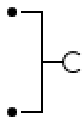
Element	Symbol
R, resistance	
C, capacitance	
L, inductance	
Q, constant phase element	
W, Warburg impedance	
O, cotangent hyperbolic	
T, tangent hyperbolic	
G, Gerischer impedance	
B2, Bisquert #2	

All of the circuit elements are fitted with one input connection and one output connection.

These circuit elements can be arranges in series or in parallel, using the connectors shown in *Table 13*.

Table 13 Overview of the available connectors

Connector	Symbol
Serial	
Parallel split	

Connector	Symbol
Parallel join	

All of the connectors are fitted with one of or more input connections and one or more output connections.

7.9.2.1.1 Resistance, R

The resistance circuit element is represented by the letter **R** and identified by the following symbol:

This element is used to typically represent solution resistance or charge transfer resistance.

The impedance of the resistance is provided by:

$$Z_R = R$$

The properties of the R element are shown in *Figure 457*.

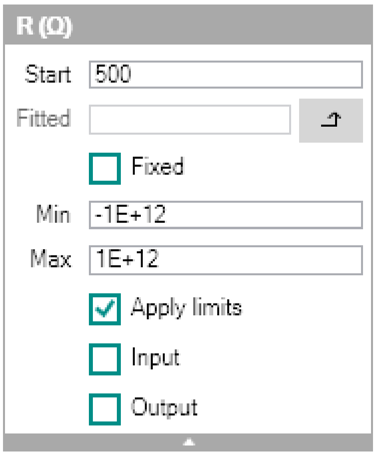


Figure 457 The properties of the R element

The following properties are available:

- **Start:** the start value of the resistance, in Ohm. The default value is 500 Ohm.
- **Fitted:** the fitted value of the resistance, in Ohm. This value is only available after the **Fit and simulation** has been executed.
- **Fixed:** specifies if the value can be modified by the **Fit and simulation** command, using the provided checkbox. Fixed properties are shown in red in the Equivalent circuit editor.
- **Min:** specifies the minimum value for the resistance, in Ohm. The default value is -1 TOhm.

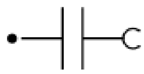
- **Max:** specifies the maximum value for the resistance, in Ohm. The default value is 1 TOhm.
- **Apply limits:** specifies if the Min. and Max. limits should be used by the **Fit and simulation** command, using the provided checkbox. When this property is on, the value of the resistance will be kept between the specified Min. and Max., otherwise, the value will be allowed to take any possible value.
- **Input:** creates an input anchoring point for linking purposes, using the specified checkbox.
- **Output:** creates an output anchoring point for linking purposes, using the specified checkbox.

NOTE

In order to create input and output anchoring points for linking, a **unique** name must be specified for the element. Please refer to *Chapter 7.9.2.7* for more information.

7.9.2.1.2 Capacitance, C

The capacitance circuit element is represented by the letter **C** and identified by the following symbol:



This element is used to typically represent double layer capacitance of the electrochemical interface.

The impedance of the capacitance is provided by:

$$Z_C = \frac{-j}{\omega C}$$

The properties of the C element are shown in *Figure 458*.

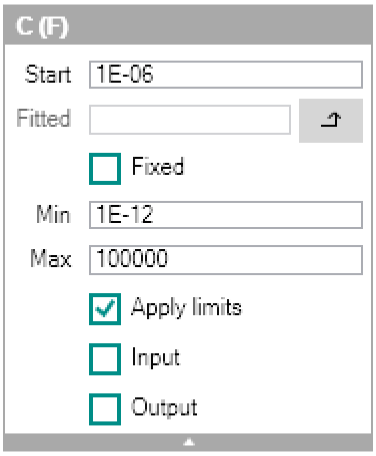


Figure 458 The properties of the C element

The following properties are available:

- **Start:** the start value of the capacitance, in F. The default value is 1 µF.
- **Fitted:** the fitted value of the capacitance, in F. This value is only available after the **Fit and simulation** has been executed.
- **Fixed:** specifies if the value can be modified by the **Fit and simulation** command, using the provided checkbox. Fixed properties are shown in red in the Equivalent circuit editor.
- **Min:** specifies the minimum value for the capacitance, in F. The default value is 1 pF.
- **Max:** specifies the maximum value for the capacitance, in F. The default value is 100 kF.
- **Apply limits:** specifies if the Min. and Max. limits should be used by the **Fit and simulation** command, using the provided checkbox. When this property is on, the value of the capacitance will be kept between the specified Min. and Max., otherwise, the value will be allowed to take any possible value.
- **Input:** creates an input anchoring point for linking purposes, using the specified checkbox.
- **Output:** creates an output anchoring point for linking purposes, using the specified checkbox.

NOTE

In order to create input and output anchoring points for linking, a **unique** name must be specified for the element. Please refer to *Chapter 7.9.2.7* for more information.

7.9.2.1.3 Inductance, L

The inductance circuit element is represented by the letter **L** and identified by the following symbol:

This element is used to typically represent adsorption process on the electrochemical interface.

The impedance of the inductance is provided by:

$$Z_L = j\omega L$$

The properties of the L element are shown in *Figure 459*.

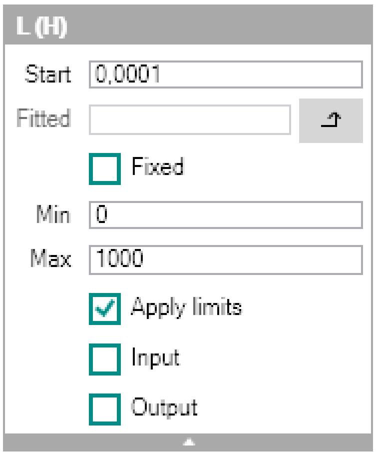


Figure 459 The properties of the L element

The following properties are available:

- **Start:** the start value of the inductance, in H. The default value is 100 µH.
- **Fitted:** the fitted value of the inductance, in H. This value is only available after the **Fit and simulation** has been executed.
- **Fixed:** specifies if the value can be modified by the **Fit and simulation** command, using the provided checkbox. Fixed properties are shown in red in the Equivalent circuit editor.
- **Min:** specifies the minimum value for the inductance, in H. The default value is 0 H.
- **Max:** specifies the maximum value for the inductance, in H. The default value is 1 kH.
- **Apply limits:** specifies if the Min. and Max. limits should be used by the **Fit and simulation** command, using the provided checkbox. When this property is on, the value of the inductance will be kept between the specified Min. and Max., otherwise, the value will be allowed to take any possible value.

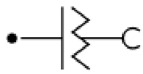
- **Input:** creates an input anchoring point for linking purposes, using the specified checkbox.
- **Output:** creates an output anchoring point for linking purposes, using the specified checkbox.

NOTE

In order to create input and output anchoring points for linking, a **unique** name must be specified for the element. Please refer to *Chapter 7.9.2.7* for more information.

7.9.2.1.4 Constant phase element, Q

The constant phase element circuit element is represented by the letter **Q** and identified by the following symbol:



This element is used to typically represent the non-ideal behavior of the electrochemical double layer.

The impedance of the constant phase element is provided by:

$$Z_Q = \frac{1}{Y_0 (j\omega)^n}$$

The properties of the Q element are shown in *Figure 460*.

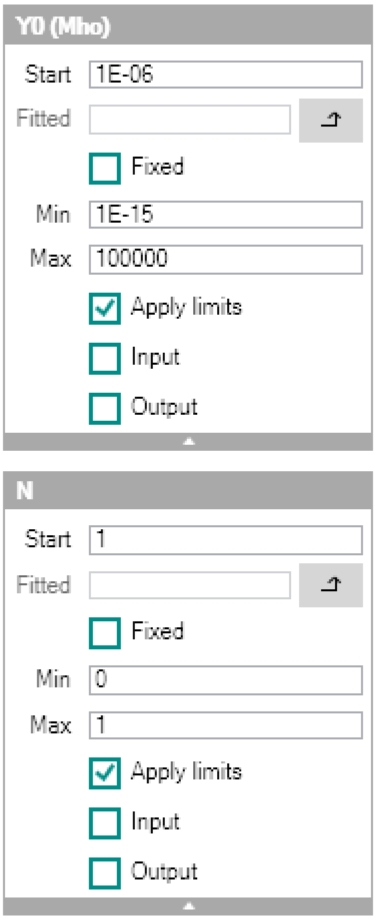


Figure 460 The properties of the Q element

The Q element is defined by two values:

- **Y0:** the admittance value, in Mho.
- **n:** the exponent used in the expression of the constant phase element.

The following **specific** properties are available for the **Y0** value:

- **Start:** the start value of the admittance, **Y0**, of the constant phase element, in Mho. The default value is 1 µMho.
- **Fitted:** the fitted value of the admittance, **Y0**, constant phase element, in Mho. This value is only available after the **Fit and simulation** has been executed.
- **Min:** specifies the minimum value for the admittance, **Y0**, of the constant phase element, in Mho. The default value is 1 fMho.
- **Max:** specifies the maximum value for the admittance, **Y0**, of the constant phase element, in Mho. The default value is 100 kMho.

The following **specific** properties are available for the **n** value:

- **Start:** the start value of the exponent, **n**, of the constant phase element. The default value is 1.

- **Fitted:** the fitted value of the exponent, **n**, of the constant phase element. This value is only available after the **Fit and simulation** has been executed.
- **Min:** specifies the minimum value for the exponent, **n**, of the constant phase element. The default value is 0.
- **Max:** specifies the maximum value for the exponent, **n**, of the constant phase element. The default value is 1.

The following **common** properties are available:

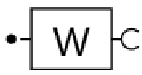
- **Fixed:** specifies if the value can be modified by the **Fit and simulation** command, using the provided checkbox. Fixed properties are shown in red in the Equivalent circuit editor.
- **Apply limits:** specifies if the Min. and Max. limits should be used by the **Fit and simulation** command, using the provided checkbox. When this property is on, the value will be kept between the specified Min. and Max., otherwise, the value will be allowed to take any possible value.
- **Input:** creates an input anchoring point for linking purposes, using the specified checkbox.
- **Output:** creates an output anchoring point for linking purposes, using the specified checkbox.

NOTE

In order to create input and output anchoring points for linking, a **unique** name must be specified for the element. Please refer to *Chapter 7.9.2.7* for more information.

7.9.2.1.5 Warburg, W

The Warburg circuit element is represented by the letter **W** and identified by the following symbol:



This element is used to typically represent the semi-infinite diffusion of electroactive species.

The impedance of the Warburg is provided by:

$$Z_W = \frac{1}{Y_0\sqrt{j\omega}}$$

The properties of the W element are shown in *Figure 461*.

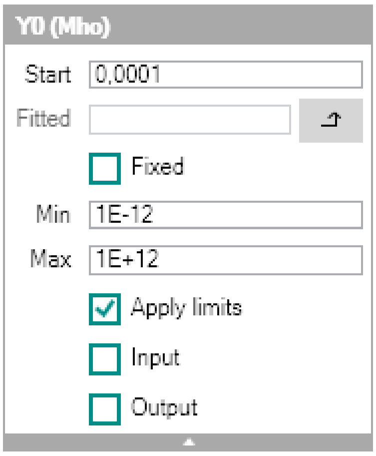


Figure 461 The properties of the W element

The following properties are available:

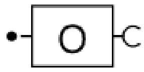
- **Start:** the start value of the admittance, in Mho. The default value is 100 mMho.
- **Fitted:** the fitted value of the admittance, in Mho. This value is only available after the **Fit and simulation** has been executed.
- **Fixed:** specifies if the value can be modified by the **Fit and simulation** command, using the provided checkbox. Fixed properties are shown in red in the Equivalent circuit editor.
- **Min:** specifies the minimum value for the admittance, in Mho. The default value is 1 pMho.
- **Max:** specifies the maximum value for the admittance, in Mho. The default value is 1 TMho.
- **Apply limits:** specifies if the Min. and Max. limits should be used by the **Fit and simulation** command, using the provided checkbox. When this property is on, the value of the admittance will be kept between the specified Min. and Max., otherwise, the value will be allowed to take any possible value.
- **Input:** creates an input anchoring point for linking purposes, using the specified checkbox.
- **Output:** creates an output anchoring point for linking purposes, using the specified checkbox.

NOTE

In order to create input and output anchoring points for linking, a **unique** name must be specified for the element. Please refer to *Chapter 7.9.2.7* for more information.

7.9.2.1.6 Cotangent hyperbolic, O

The cotangent hyperbolic circuit element is represented by the letter **O** and identified by the following symbol:



This element is used to typically represent the limited diffusion of electro-active species.

The impedance of the cotangent hyperbolic is provided by:

$$Z_O = \frac{1}{Y_0\sqrt{j\omega}} \tanh\left(B\sqrt{j\omega}\right)$$

The properties of the O element are shown in *Figure 462*.

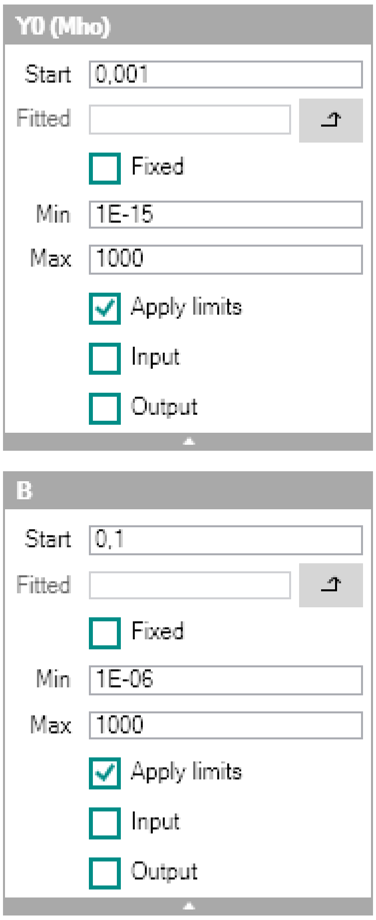


Figure 462 The properties of the O element

The O element is defined by two values:

- **Y0:** the admittance value, in Mho.
- **B:** the factor associated with the thickness of the diffusion layer.

The following **specific** properties are available for the **Y0** value:

- **Start:** the start value of the admittance, **Y0**, of the cotangent hyperbolic, in Mho. The default value is 1 mMho.
- **Fitted:** the fitted value of the admittance, **Y0**, cotangent hyperbolic, in Mho. This value is only available after the **Fit and simulation** has been executed.
- **Min:** specifies the minimum value for the admittance, **Y0**, of the cotangent hyperbolic, in Mho. The default value is 1 fMho.
- **Max:** specifies the maximum value for the admittance, **Y0**, of the cotangent hyperbolic, in Mho. The default value is 1 kMho.

The following **specific** properties are available for the **B** value:

- **Start:** the start value of the thickness factor, **B**, of the cotangent hyperbolic. The default value is 0.1.
- **Fitted:** the fitted value of the thickness factor, **B**, of the cotangent hyperbolic. This value is only available after the **Fit and simulation** has been executed.
- **Min:** specifies the minimum value for the thickness factor, **B**, of the cotangent hyperbolic. The default value is 1 µ.
- **Max:** specifies the maximum value for the thickness factor, **B**, of the cotangent hyperbolic. The default value is 1000.

The following **common** properties are available:

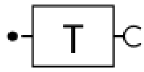
- **Fixed:** specifies if the value can be modified by the **Fit and simulation** command, using the provided checkbox. Fixed properties are shown in red in the Equivalent circuit editor.
- **Apply limits:** specifies if the Min. and Max. limits should be used by the **Fit and simulation** command, using the provided checkbox. When this property is on, the value will be kept between the specified Min. and Max., otherwise, the value will be allowed to take any possible value.
- **Input:** creates an input anchoring point for linking purposes, using the specified checkbox.
- **Output:** creates an output anchoring point for linking purposes, using the specified checkbox.

NOTE

In order to create input and output anchoring points for linking, a **unique** name must be specified for the element. Please refer to *Chapter 7.9.2.7* for more information.

7.9.2.1.7 Tangent hyperbolic, T

The tangent hyperbolic circuit element is represented by the letter **T** and identified by the following symbol:



This element is used to typically represent the limited diffusion of electro-active species.

The impedance of the tangent hyperbolic is provided by:

$$Z_T = \frac{1}{Y_0\sqrt{j\omega}}\coth\left(B\sqrt{j\omega}\right)$$

The properties of the T element are shown in *Figure 463*.

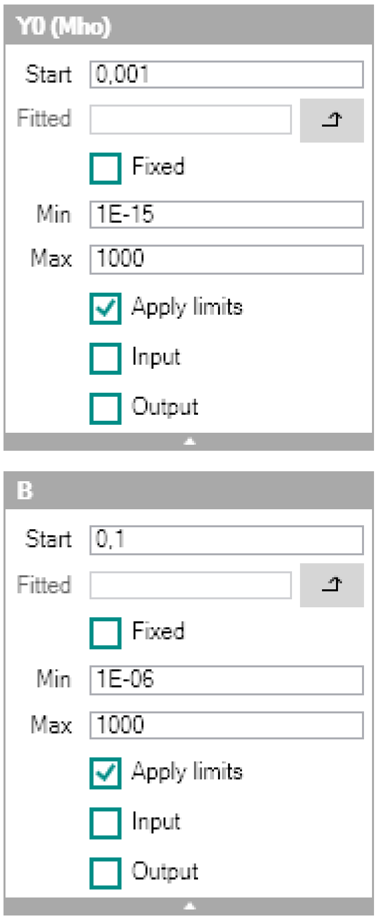


Figure 463 The properties of the T element

The T element is defined by two values:

- **Y0:** the admittance value, in Mho.
- **B:** the factor associated with the thickness of the diffusion layer.

The following **specific** properties are available for the **Y0** value:

- **Start:** the start value of the admittance, **Y0**, of the tangent hyperbolic, in Mho. The default value is 1 mMho.
- **Fitted:** the fitted value of the admittance, **Y0**, tangent hyperbolic, in Mho. This value is only available after the **Fit and simulation** has been executed.
- **Min:** specifies the minimum value for the admittance, **Y0**, of the tangent hyperbolic, in Mho. The default value is 1 fMho.
- **Max:** specifies the maximum value for the admittance, **Y0**, of the tangent hyperbolic, in Mho. The default value is 1 kMho.

The following **specific** properties are available for the **B** value:

- **Start:** the start value of the thickness factor, **B**, of the tangent hyperbolic. The default value is 0.1.
- **Fitted:** the fitted value of the thickness factor, **B**, of the tangent hyperbolic. This value is only available after the **Fit and simulation** has been executed.
- **Min:** specifies the minimum value for the thickness factor, **B**, of the tangent hyperbolic. The default value is 1 µ.
- **Max:** specifies the maximum value for the thickness factor, **B**, of the tangent hyperbolic. The default value is 1000.

The following **common** properties are available:

- **Fixed:** specifies if the value can be modified by the **Fit and simulation** command, using the provided checkbox. Fixed properties are shown in red in the Equivalent circuit editor.
- **Apply limits:** specifies if the Min. and Max. limits should be used by the **Fit and simulation** command, using the provided checkbox. When this property is on, the value will be kept between the specified Min. and Max., otherwise, the value will be allowed to take any possible value.
- **Input:** creates an input anchoring point for linking purposes, using the specified checkbox.
- **Output:** creates an output anchoring point for linking purposes, using the specified checkbox.

NOTE

In order to create input and output anchoring points for linking, a **unique** name must be specified for the element. Please refer to *Chapter 7.9.2.7* for more information.

7.9.2.1.8 Gerischer, G

The Gerischer circuit element is represented by the letter **G** and identified by the following symbol:

This element is used to typically represent a coupled chemical and electrochemical reaction (CE mechanism).

The impedance of the Gerischer circuit element is provided by:

$$Z_G = \frac{1}{Y_0\sqrt{k_a + j\omega}}$$

The properties of the G element are shown in *Figure 464*.

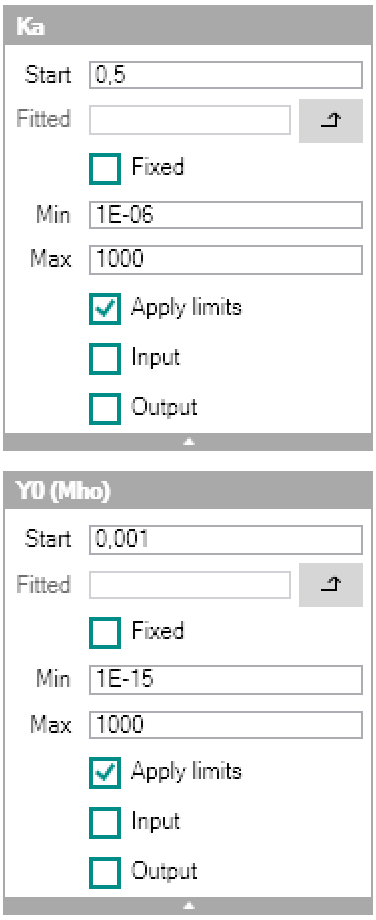


Figure 464 The properties of the G element

The G element is defined by two values:

- **Ka:** the kinetic constant of the chemical reaction.
- **Y0:** the admittance value, in Mho.

The following **specific** properties are available for the **Ka** value:

- **Start:** the start value of the kinetic constant, **Ka**, of the Gerischer. The default value is 0.5.
- **Fitted:** the fitted value of the kinetic constant, **Ka**, of the Gerischer. This value is only available after the **Fit and simulation** has been executed.
- **Min:** specifies the minimum value for the kinetic constant, **Ka**, of the Gerischer. The default value is 1 µ.
- **Max:** specifies the maximum value for the kinetic constant, **Ka**, of the Gerischer. The default value is 1000.

The following **specific** properties are available for the **Y0** value:

- **Start:** the start value of the admittance, **Y0**, of the Gerischer, in Mho. The default value is 1 mMho.
- **Fitted:** the fitted value of the admittance, **Y0**, Gerischer, in Mho. This value is only available after the **Fit and simulation** has been executed.
- **Min:** specifies the minimum value for the admittance, **Y0**, of the Gerischer, in Mho. The default value is 1 fMho.
- **Max:** specifies the maximum value for the admittance, **Y0**, of the Gerischer, in Mho. The default value is 1 kMho.

The following **common** properties are available:

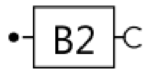
- **Fixed:** specifies if the value can be modified by the **Fit and simulation** command, using the provided checkbox. Fixed properties are shown in red in the Equivalent circuit editor.
- **Apply limits:** specifies if the Min. and Max. limits should be used by the **Fit and simulation** command, using the provided checkbox. When this property is on, the value will be kept between the specified Min. and Max., otherwise, the value will be allowed to take any possible value.
- **Input:** creates an input anchoring point for linking purposes, using the specified checkbox.
- **Output:** creates an output anchoring point for linking purposes, using the specified checkbox.

NOTE

In order to create input and output anchoring points for linking, a **unique** name must be specified for the element. Please refer to *Chapter 7.9.2.7* for more information.

7.9.2.1.9 Bisquert #2, B2

The Bisquert #2 circuit element is represented by the letter **B2** and identified by the following symbol:



This element is a transmission line element derived from the classical model for a porous or mixed-phase electrode of thickness L. The model is represented in *Figure 465*.

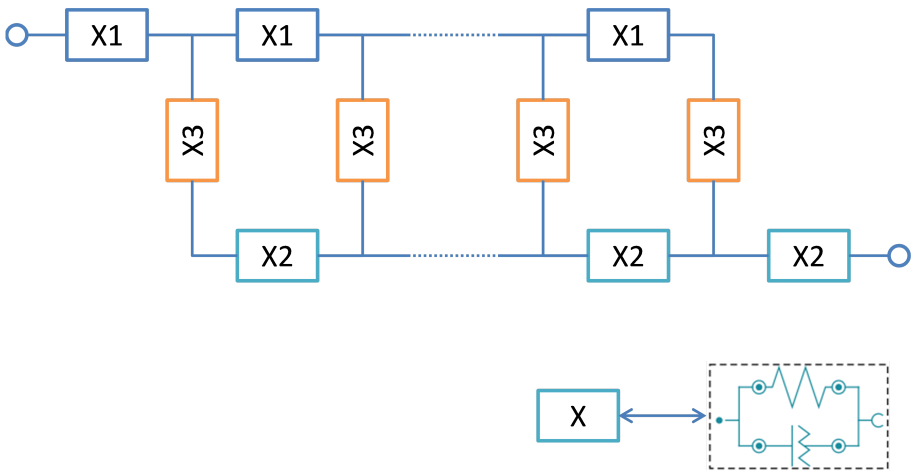


Figure 465 Overview of the general transmission line model used in the B2 element

In the **B2** element, the **X** element used in the transmission line is represented by a parallel combination of a resistor (R) and a constant phase element (Q).

This transmission line is often used in the world of dye-sensitized solar cells (DSC) and in general systems that analyze the combination of charge transport, accumulation and recombination.

The impedance of this equivalent circuit element may be written as:

$$Z_{B2} = \frac{X_1 X_2}{X_1 + X_2}\left(L + \frac{2\lambda}{\sinh\left(\frac{L}{\lambda}\right)}\right) + \lambda\frac{X_1^2 + X_2^2}{X_1 + X_2}\operatorname{cotanh}\left(\frac{L}{\lambda}\right)$$

Where λ is given by:

$$\lambda = \sqrt{\frac{X_3}{X_1 + X_2}}$$

This element is a composite element, consisting of three types of parallel (RQ) element combinations. For the properties of the R element and the Q element, please refer to *Chapter 7.9.2.1.1* and *Chapter 7.9.2.1.4*, respectively.

The **B2** element provides one additional property, L, representing the length of the transmission line, shown in *Figure 466*.

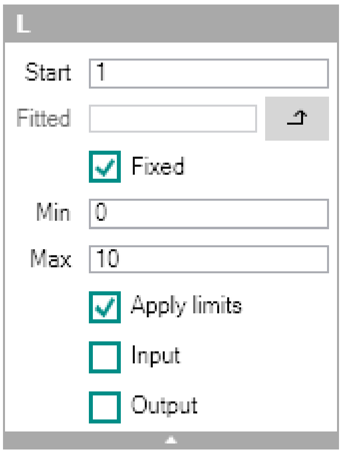


Figure 466 The additional property of the B2 element

The following properties are available:

- **Start:** the start value of the transmission line length. The default value is 1.
- **Fitted:** the fitted value of the transmission line length. This value is only available after the **Fit and simulation** has been executed.
- **Fixed:** specifies if the value can be modified by the **Fit and simulation** command, using the provided checkbox. Fixed properties are shown in red in the Equivalent circuit editor.
- **Min:** specifies the minimum value for the transmission line length. The default value is 0.
- **Max:** specifies the maximum value for the transmission line length. The default value is 10.
- **Apply limits:** specifies if the Min. and Max. limits should be used by the **Fit and simulation** command, using the provided checkbox. When this property is on, the value of the transmission line length will be kept between the specified Min. and Max., otherwise, the value will be allowed to take any possible value.
- **Input:** creates an input anchoring point for linking purposes, using the specified checkbox.
- **Output:** creates an output anchoring point for linking purposes, using the specified checkbox.

NOTE

In order to create input and output anchoring points for linking, a **unique** name must be specified for the element. Please refer to *Chapter 7.9.2.7* for more information.

NOTE

For more information on the Bisquert #2 transmission line model, please refer to J. Bisquert, G. Garcia-Belmonte, F. Fabregat-Santiago, A. Compte, Electrochemistry Communications 1999, 1:9:429-435 and J. Bisquert; Phys. Chem. Chem. Phys., Vol. 2 (2000), pp. 4185-4192.

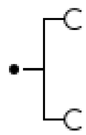
7.9.2.1.10 Serial connection

The **Serial connection** can be used to place two circuit elements in **series**. The **Serial connection** is represented by the following symbol:

The **Serial connection** has one input connection and one output connection.

7.9.2.1.11 Parallel split connection

The **Parallel split connection** can be used to place two or more circuit elements in **parallel**. The **Parallel split** connection creates a parallel arrangement. The **Parallel split** connection is represented by the following symbol:



The **Parallel split** connection has one input connection and two output connections.

If needed additional output connections can be created, by right-clicking the connection and selecting the *Add output* option from the context menu *(see Figure 467, page 413)*.

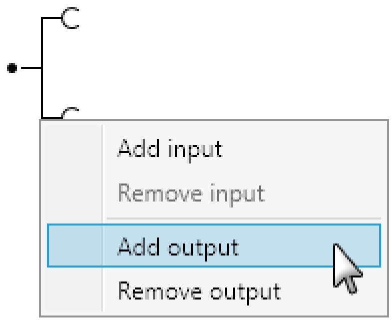


Figure 467 Adding additional output connections

The additional output will be added to the element *(see Figure 468, page 413)*.

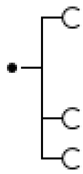


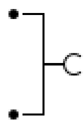
Figure 468 The additional output is added to the Parallel split connection

NOTE

The same menu can be used to remove outputs from the element.

7.9.2.1.12 Parallel join connection

The **Parallel join connection** can be used to place two or more circuit elements in **parallel**. The **Parallel join** connection closes a parallel arrangement. The **Parallel join** connection is represented by the following symbol:



The **Parallel join** connection has two input connections and one output connection.

If needed additional input connections can be created, by right-clicking the connection and selecting the *Add input* option from the context menu *(see Figure 469, page 414)*.

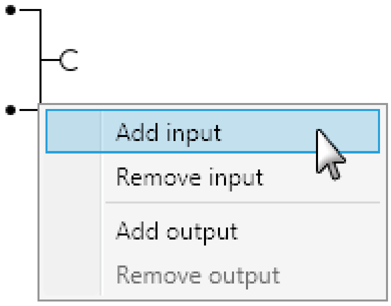


Figure 469 Adding additional input connections

The additional input be added to the element *(see Figure 470, page 414)*.

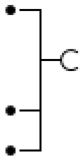


Figure 470 The additional input is added to the Parallel join connection

NOTE

The same menu can be used to remove inputs from the element.

7.9.2.2 Build a custom equivalent circuit

The Equivalent Circuit Editor window can also be used to draw the equivalent circuit by connecting individual element to one another, graphically. It is possible to add a circuit element to the editor from the *Insert* option available in the **Edit** menu *(see Figure 471, page 415)*.

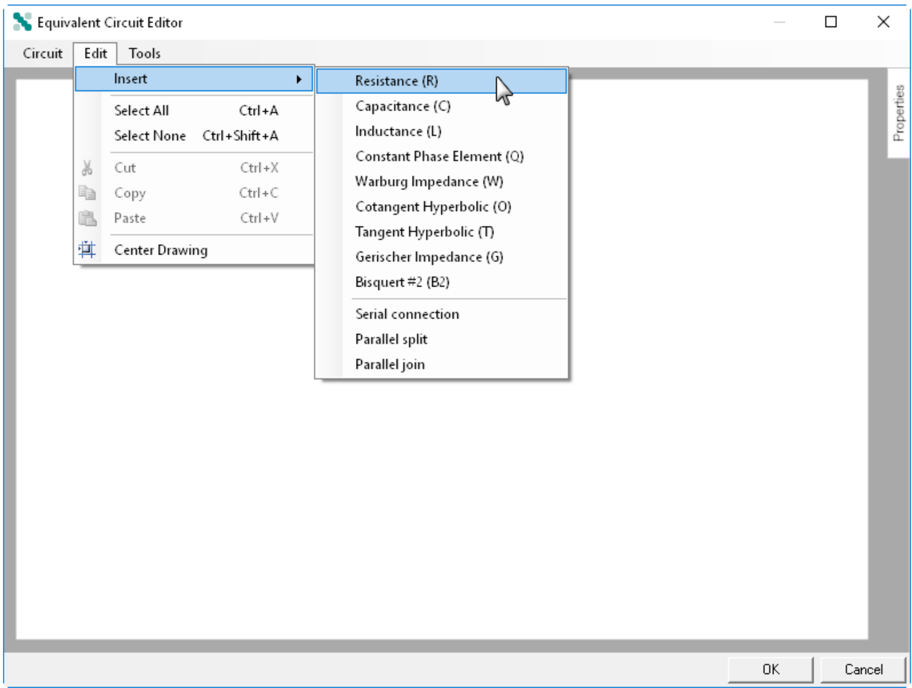


Figure 471 Adding a circuit element from the Edit menu.

It is also possible to add a circuit element by right-clicking the editor window and using the context menu *(see Figure 472, page 415)*.

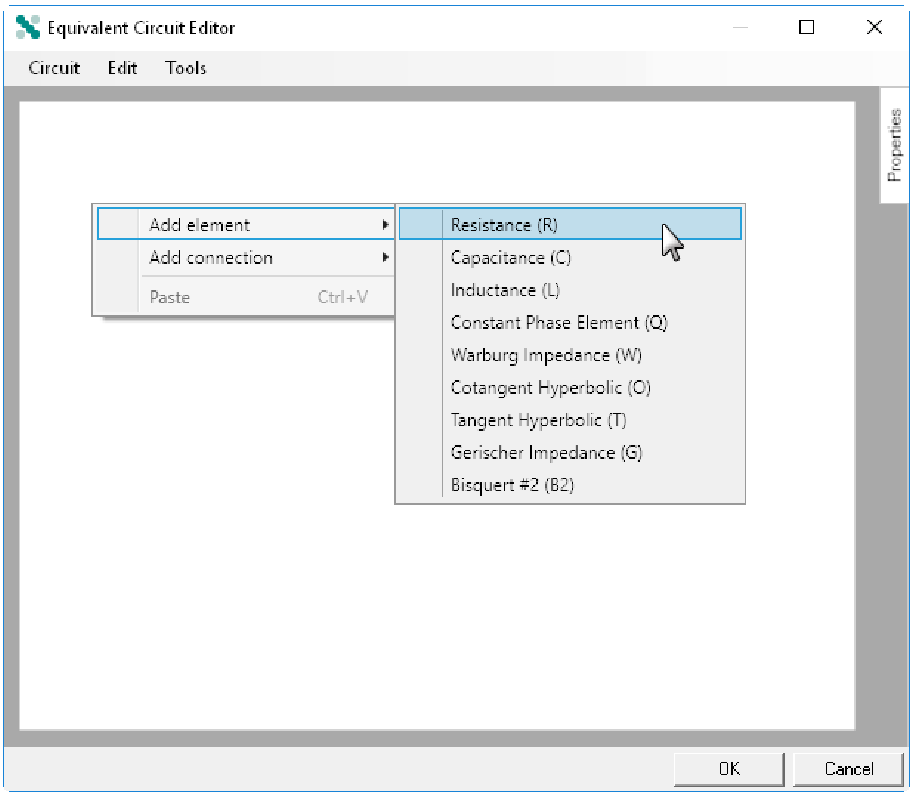


Figure 472 Adding a circuit element from the right-click menu

The selected circuit element will be added to the Equivalent circuit editor window *(see Figure 473, page 416)*.

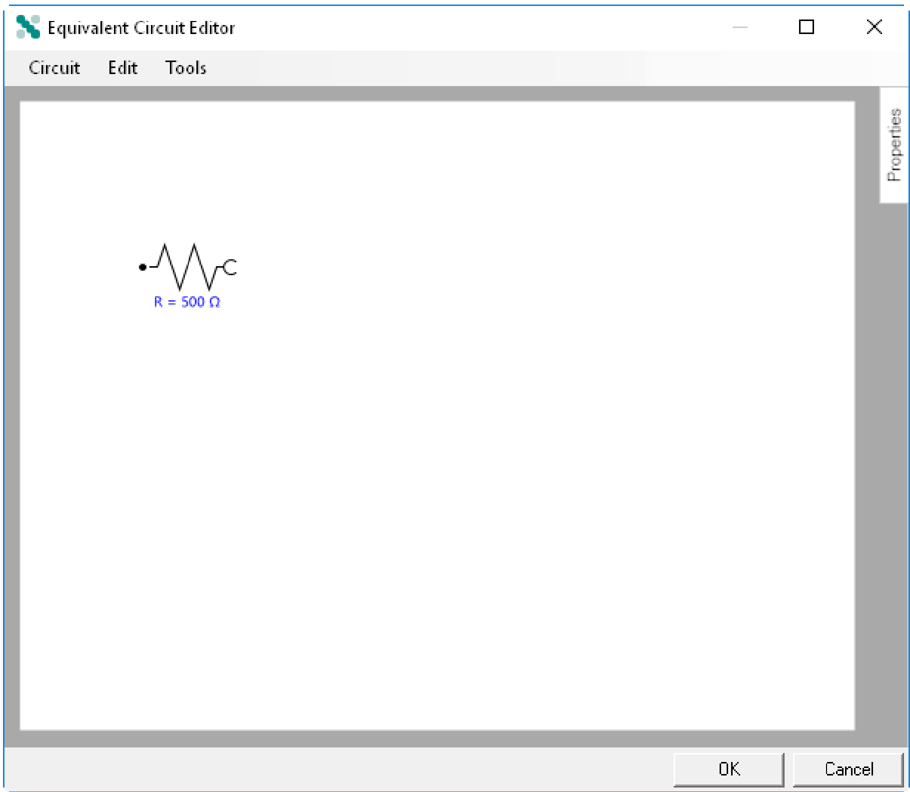


Figure 473 The circuit element is added to the editor

Once two or more circuit elements or connectors are added to the Equivalent circuit editor, they can be linked to one another.

Each circuit element is fitted with one input connection ( ) and one output connection ( ). The connectors can have one or more input connections and one or more output connections *(see Chapter 7.9.2.1, page 395)*.

The following rules are used when creating custom equivalent circuits:

- It is only possible to connect an output connection of one element or connector to the input connection of an adjacent element or connector.
- A valid equivalent circuit can only have one free input connection and one free output connection.
- A valid link between an input connection and an output connection is represented by a closed loop symbol .

To create a link between two items, click one connection, and while holding the mouse button, drag this connection close to the connection of the next item *(see Figure 474, page 417)*.

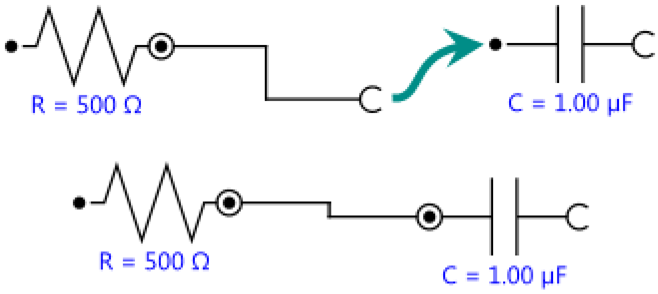


Figure 474 Linking two items in the Equivalent circuit editor

When the two ends are close enough in the editor, the software will automatically create a link.

Using this method, any equivalent circuit respecting the rules detailed above can be created *(see Figure 475, page 417)*.

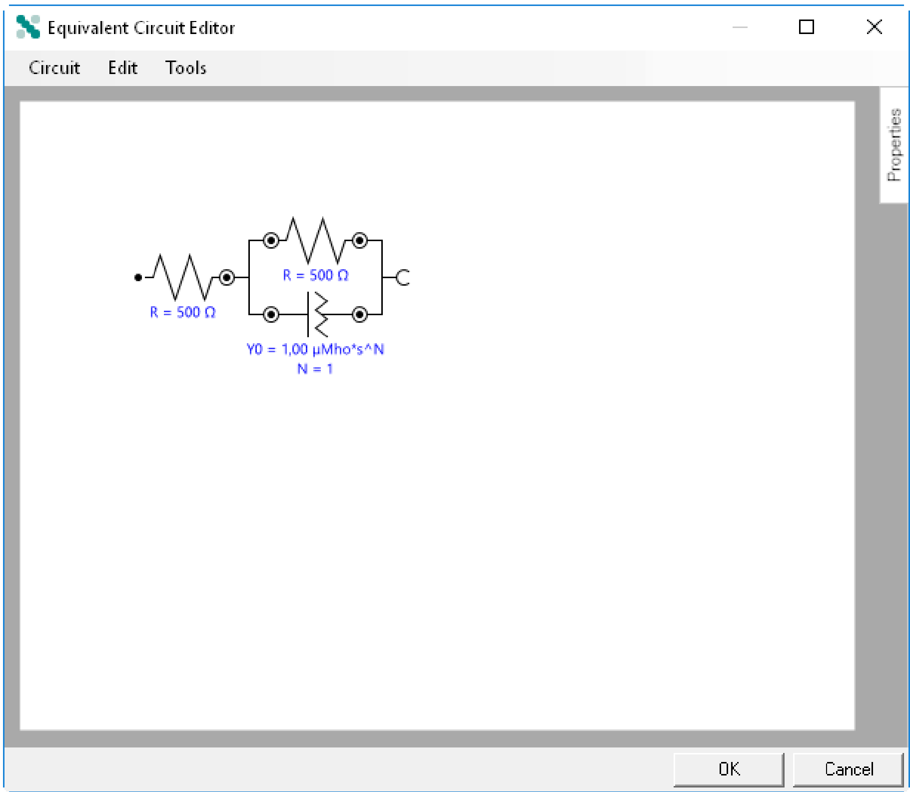


Figure 475 The custom made equivalent circuit

When the circuit is ready, it is possible to verify if there are errors by selecting the *Generate CDC from circuit* option from the Tools menu *(see Chapter 7.9.2.2, page 414)*.

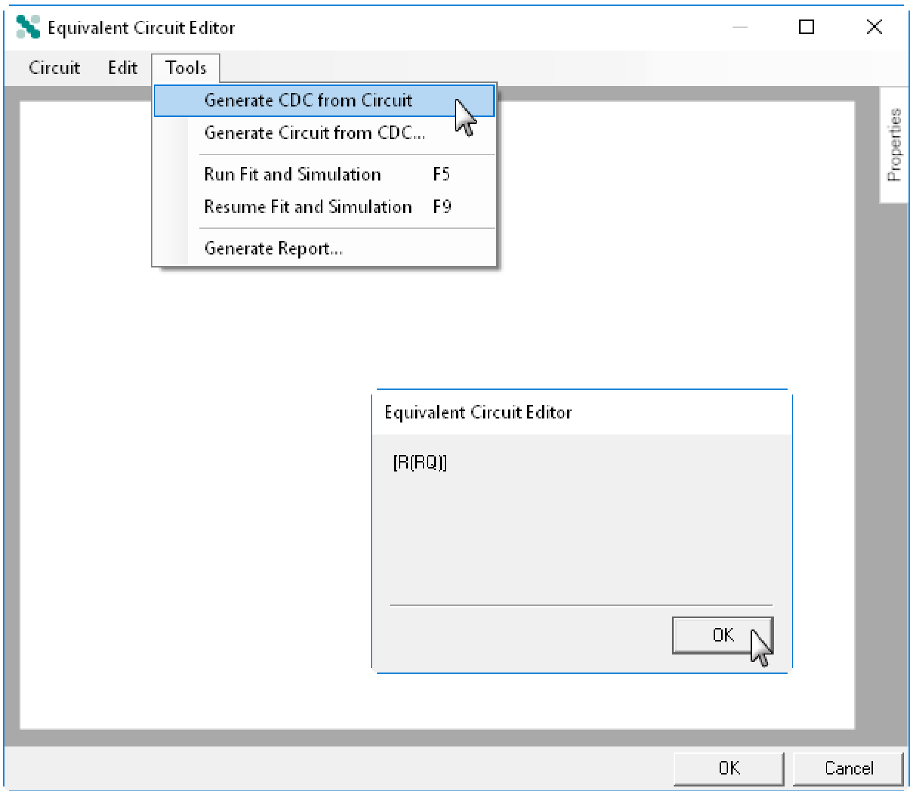


Figure 476 Generating a CDC string from the equivalent circuit

If no errors are detected in the equivalent circuit, a valid CDC string (Circuit Description Code) will be displayed. If errors are detected, an error message will be shown *(see Figure 477, page 418)*.

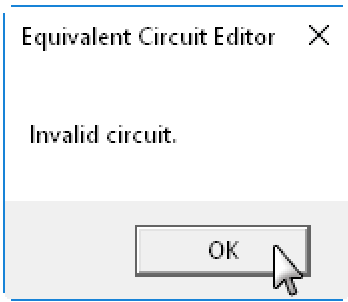


Figure 477 An error message is shown when the circuit is invalid

7.9.2.3 Generate an equivalent circuit from a CDC string

To define the equivalent circuit from a CDC string (Circuit Description Code), select the *Generate Circuit from CDC* option from Tools menu *(see Figure 478, page 419)*.

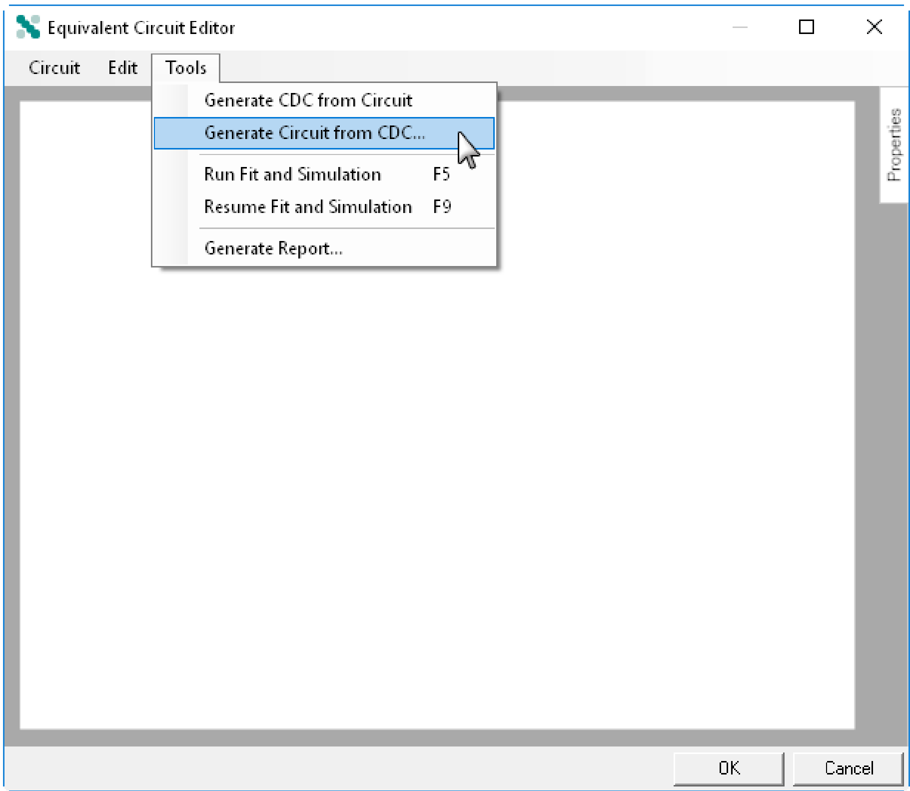


Figure 478 Select the Generate Circuit from CDC option to manually enter a CDC string

A new window that can be used to input the CDC string will be displayed *(see Figure 479, page 419)*.

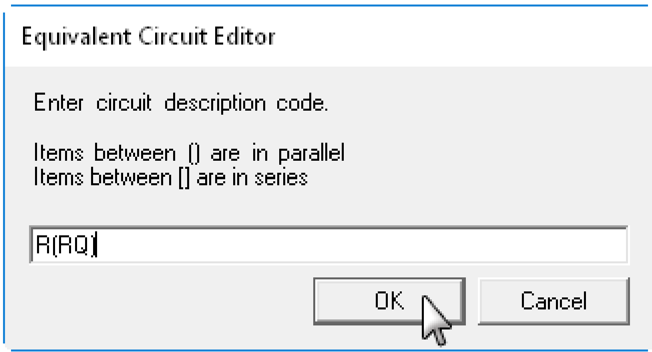


Figure 479 The CDC string can be entered using the proper formatting

To define the CDC string, the following syntax rules must be followed:

- Any of the nine element symbols defined in *Table 12* can be used.
- Element placed in parallel must be written between **( )**.
- Element placed in series must be written between **[ ]**.

Once the CDC string is defined, click the OK button to create the circuit. The equivalent circuit will be drawn in the Equivalent Circuit Editor window, displaying the default initial values of the circuit elements *(see Figure 480, page 420)*.

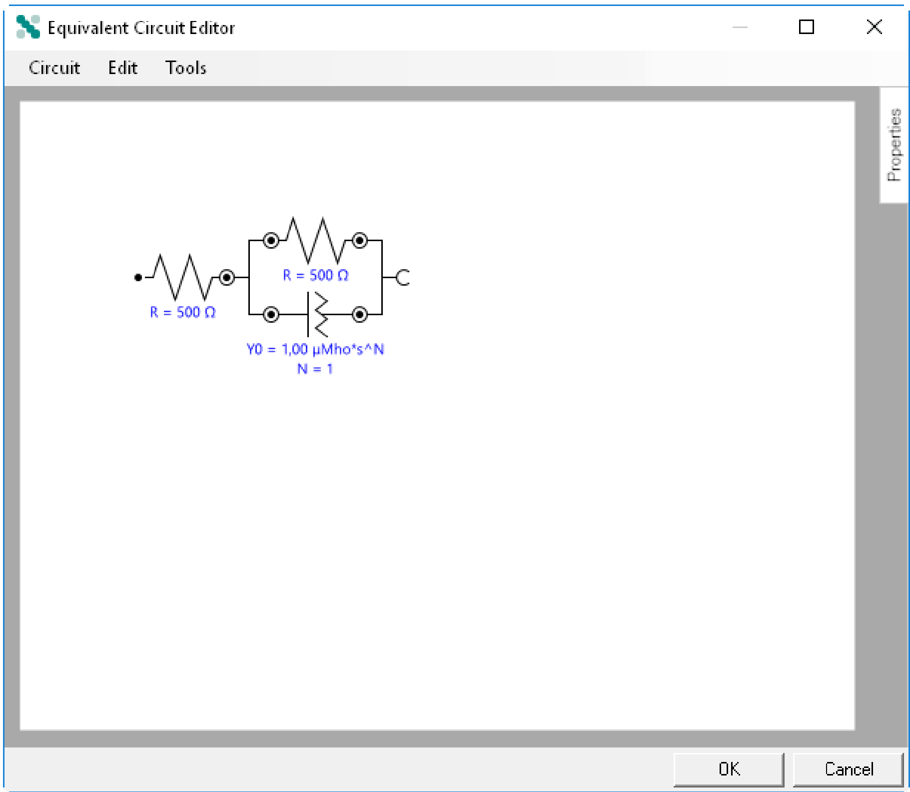


Figure 480 The equivalent circuit is generated from the CDC string

If the CDC string is invalid, an error message will be displayed *(see Figure 481, page 420)*.

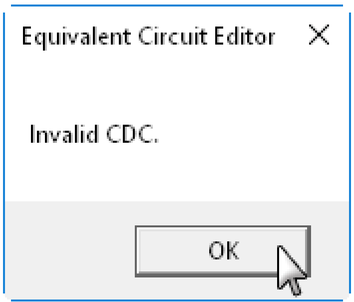


Figure 481 An error message is displayed if the CDC string is invalid

7.9.2.4 Load pre-defined circuit from a list

It is possible to choose an equivalent circuit from a pre-defined list of typical or user-defined circuits. To do this, select the *Open Circuit* option from the Circuit menu *(see Figure 482, page 421)*.

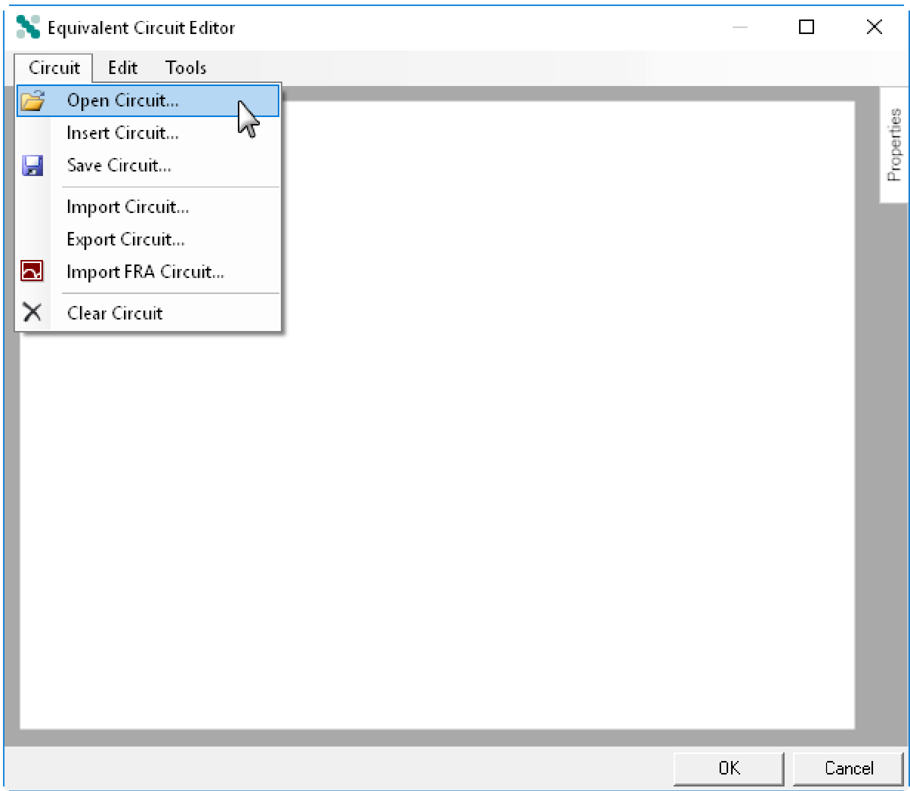


Figure 482 Opening the Circuit library

A new window will be displayed, showing two tabs *(see Figure 483, page 421)*.

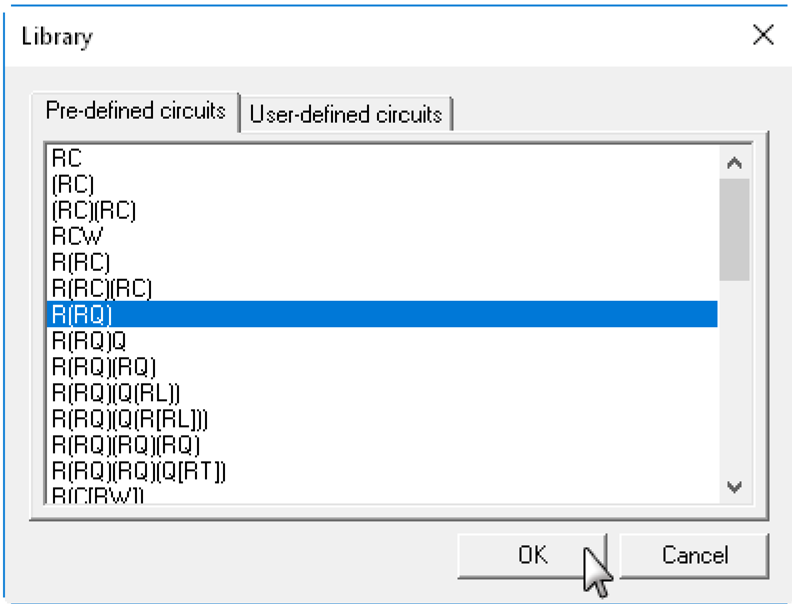


Figure 483 The library provides two lists of pre-defined equivalent circuits

- **Pre-defined circuits:** this list contains a number of typical equivalent circuits.
- **User-defined circuits:** this list contains user-defined circuits.

Select the required equivalent circuit from either list and click the OK button. The selected equivalent circuit will be drawn in the Equivalent Circuit Editor window. The default or user-defined initial values will be displayed in blue *(see Figure 484, page 422)*.

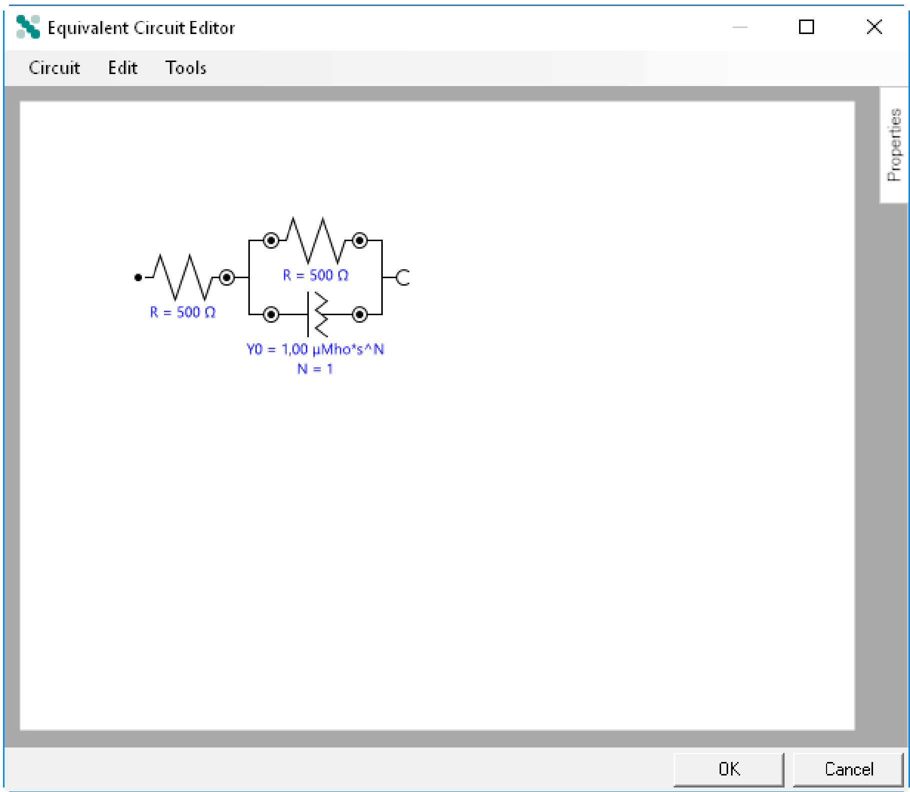


Figure 484 The equivalent circuit is loaded from the circuit library

NOTE

If the *Insert Circuit* option is selected instead of the *Open circuit* option, the selected circuit will be added to the Equivalent Circuit Editor without clearing the editor first.

7.9.2.5 Importing and Exporting equivalent circuits

It is possible to export equivalent circuits or to import equivalent circuits using the **File** menu *(see Figure 485, page 423)*.

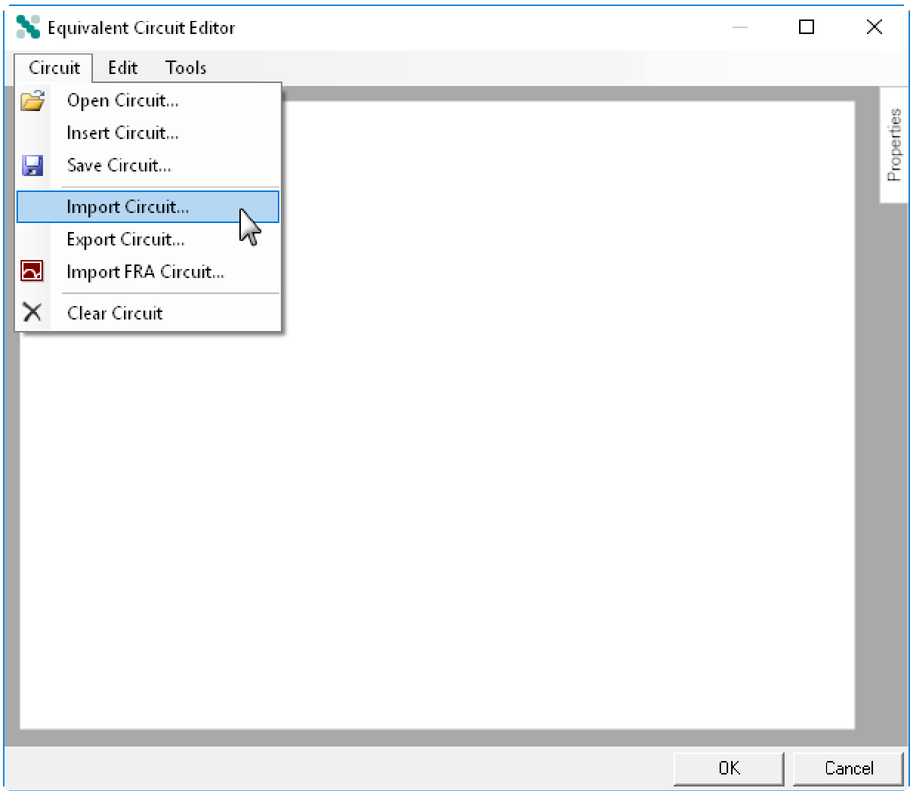


Figure 485 It is possible to import and export circuits from the File menu

The following options are provided in the **File** menu:

- **Import Circuit:** this option can be used to import an equivalent circuit stored as an .ece file, created using NOVA.
- **Export Circuit:** this option can be used to export the active equivalent circuit to an .ece file.
- **Import FRA Circuit:** this option can be used to import an equivalent circuit stored as an .ecc file, created the Autolab **FRA** software.

NOTE

The **E** equivalent circuit element used in the **FRA** equivalent circuits is converted to a **Q** element in NOVA.

7.9.2.6 Advanced editing

Additional tools are provided in the equivalent circuit editor. These can be used at any time to further edit the equivalent circuit:

- Copy/Cut and Paste element(s): select one or more elements in the equivalent editor by dragging a box around the circuit element and selecting the *Copy* or *Cut* option from the **Edit** menu (or the right-click menu or the **[CTRL]** + **[C]** and **[CTRL]** + **[X]** keyboard shortcuts) to copy them to the clipboard *(see Figure 486, page 424)*. The copied elements can then be pasted into the equivalent circuit editor, using the *Paste* option from the **Edit** menu (or the right-click menu or the **[CTRL]** + **[V]** keyboard shortcut), as shown in *Figure 7.9.2.6*.

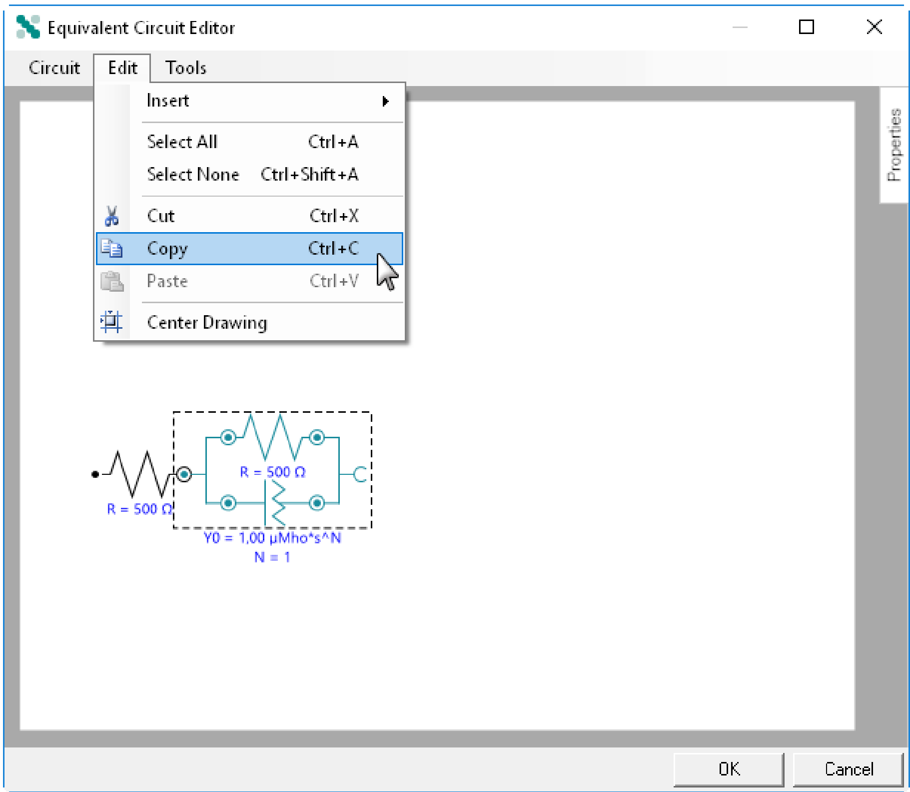


Figure 486 Selected circuit elements can be copied/pasted directly in the editor

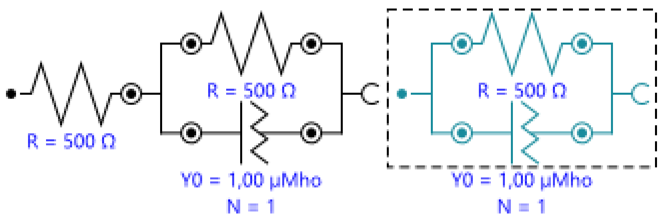


Figure 487 Elements pasted into the equivalent circuit editor have the same parameter values as the source elements

NOTE

The parameter values of the selected items are also copied to the clipboard.

- **Change element type:** right-clicking a circuit element displays a context menu which can be used to change the equivalent circuit element from one type to another *(see Figure 488, page 425)*.

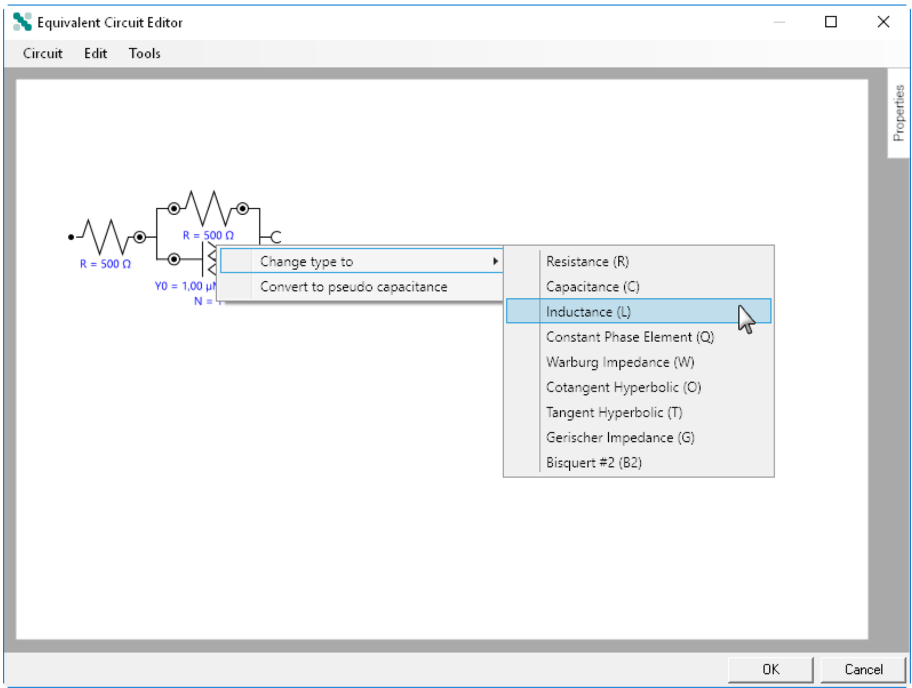


Figure 488 Changing a circuit element

- **Convert Q element to pseudo capacitance:** this option can be used to convert a constant phase element **Q** element placed in parallel with a resistance **R** element to be converted to a pseudocapacitance, **C**. The conversion is performed according to:

  $$C_{pseudo} = Y_0^{\frac{1}{n}} \cdot R^{\left(\frac{1}{n}-1\right)}$$

  Where C_{pseudo} is the resulting pseudo capacitance, in F, Y_0 is the admittance value of the constant phase element, R is the resistance value and n is the exponent of the constant phase element.
  To use this conversion tool, right click a **Q** element in parallel with a **R** element and select the *Convert to pseudo capacitance option* from the context menu as shown in *Figure 489*. The Q element will be converted to an equivalent capacitance value *(see Figure 490, page 426)*.

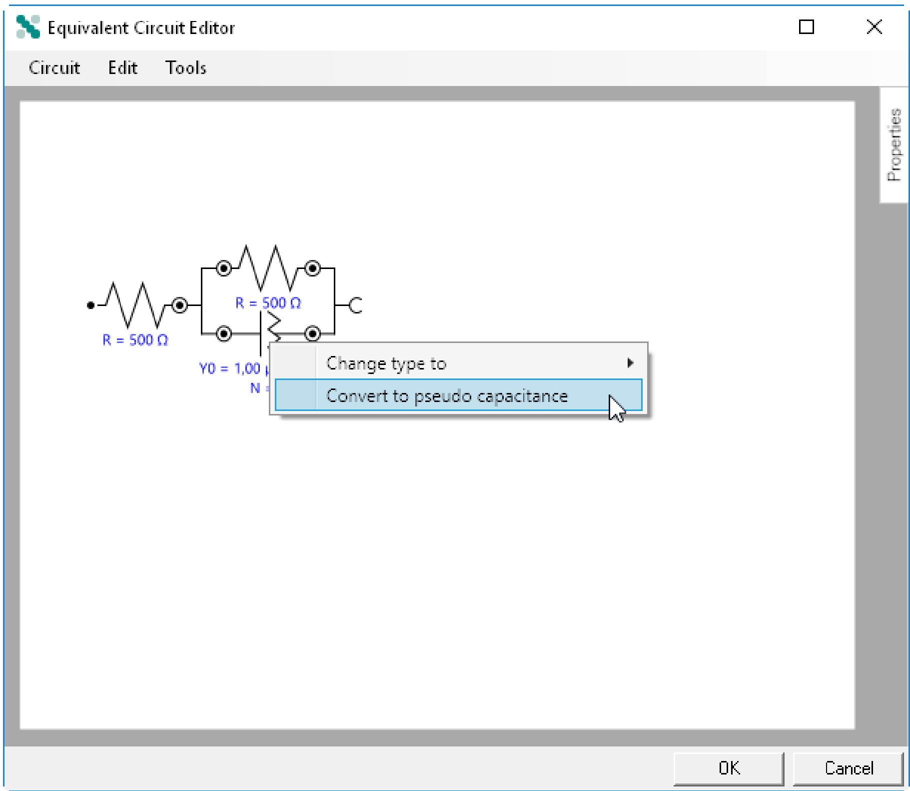


Figure 489 Converting a Q element to a pseudo capacitance

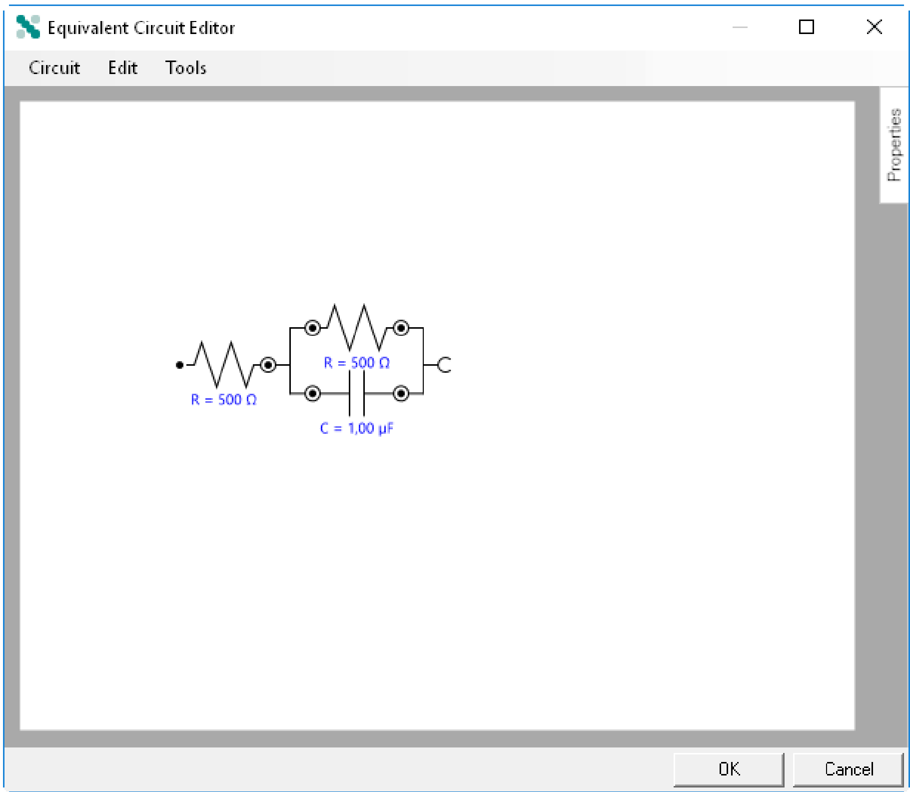


Figure 490 The Q element is converted to a C

7.9.2.7 Editing equivalent circuit properties

When the equivalent circuit is ready, it is possible to edit the properties of each of the circuit elements. To edit the properties of one of the element, click the element to select it. The selected element will be highlighted, as shown in *Figure 491*.

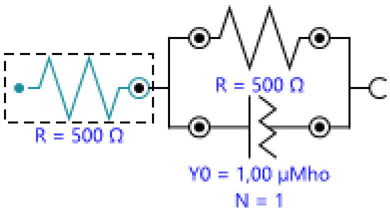


Figure 491 Selecting the equivalent circuit element

With the element selected, move the mouse pointer over the **Properties** tab on the right-hand side. The properties panel will be expanded, revealing the properties of the selected element *(see Figure 492, page 427)*.

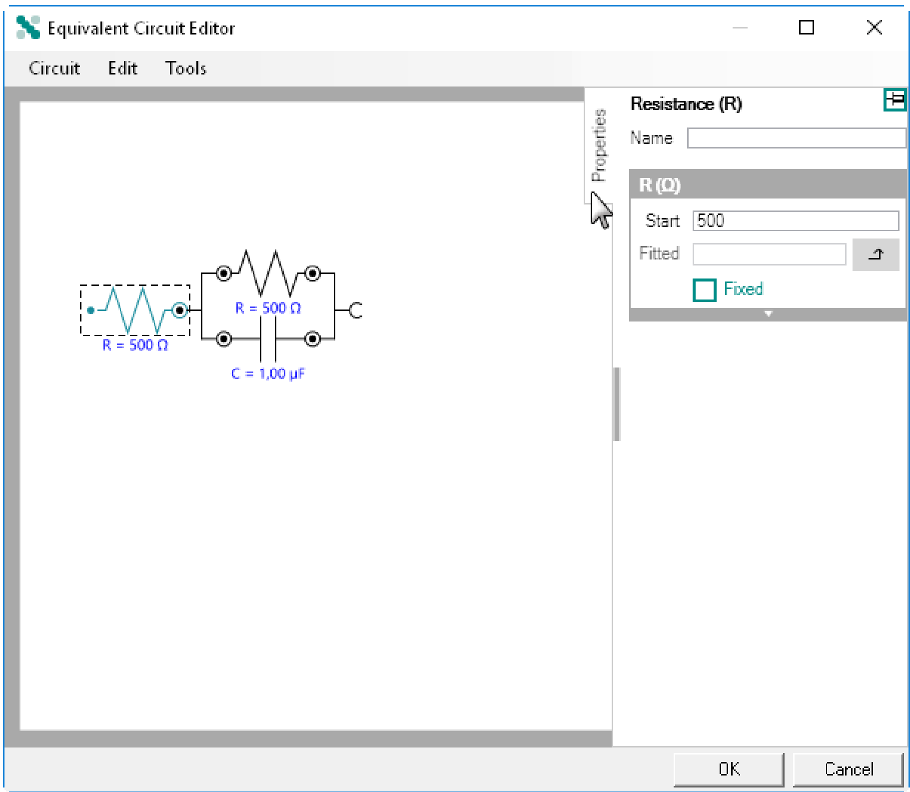


Figure 492 Displaying the properties panel

The properties panel shows one or more containers for each element, which can be expanded or collapsed to reveal or to hide advanced variables *(see Figure 493, page 428)*.

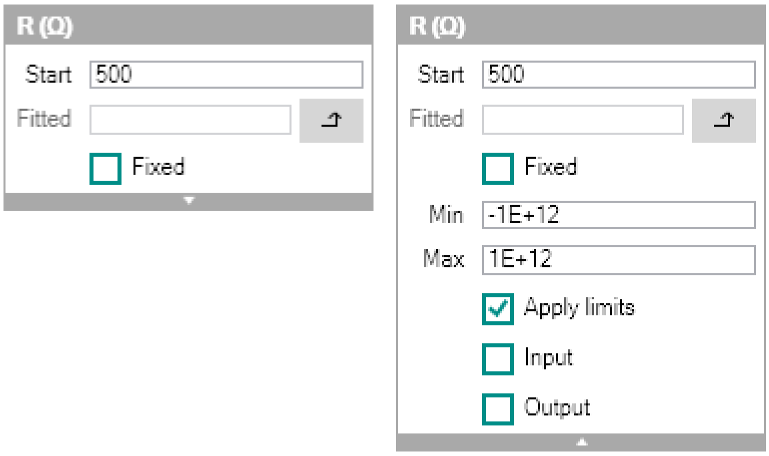


Figure 493 The basic and advanced properties

NOTE

For a description of the circuit element properties, please refer to *Chapter 7.9.2.1.1* to *Chapter 7.9.2.1.9*.

NOTE

To keep the properties tab expanded, click the pushpin button, . When the pushpin button is pressed, the properties tab will remain expanded even if no element is selected in the editor.

NOTE

For each circuit element, a unique *Name* can be specified in the **Properties** panel.

7.9.2.8 Linkable properties

If needed, it is possible to make the properties of one or more of the circuit elements *linkable*. This in turn allows the **Fit and simulation** command to be linked to other command properties.

To make a circuit element property linkable, it is necessary to first assign a *unique* name to the circuit element, by selecting it in the **Equivalent Circuit Editor** window and specifying the name in the **Name** field of the **Properties** panel, as shown in *Figure 494*.

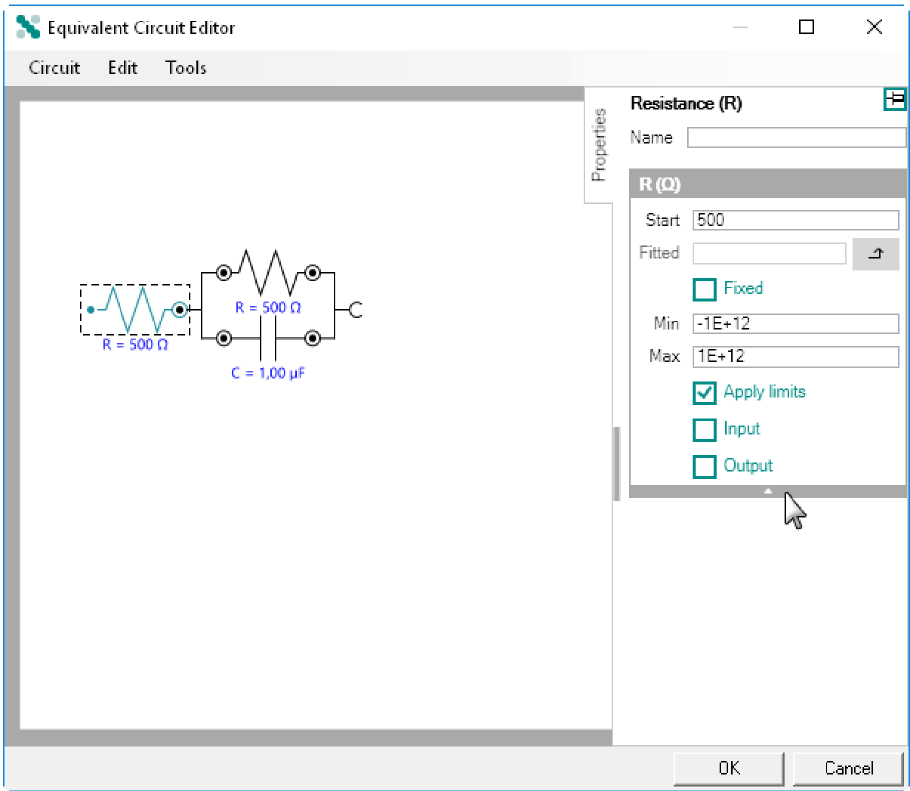


Figure 494 Specifying the name of the element

NOTE

The element name **must** be unique!

Once the name is specified, it is possible to check the **Input** and/or **Output** checkboxes in the **Properties** panel *(see Figure 495, page 430)*.

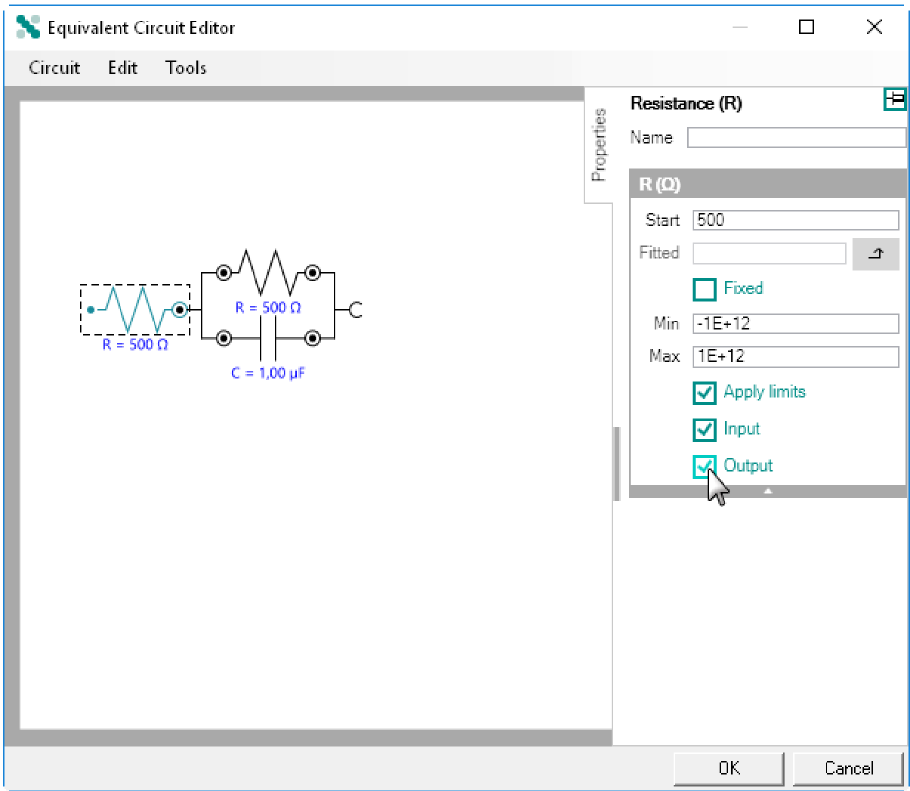


Figure 495 Specifying the linking behavior for the circuit element

This will create an input and output anchoring point for the element property, allowing it to be linked to another command properties *(see Figure 496, page 430)*.

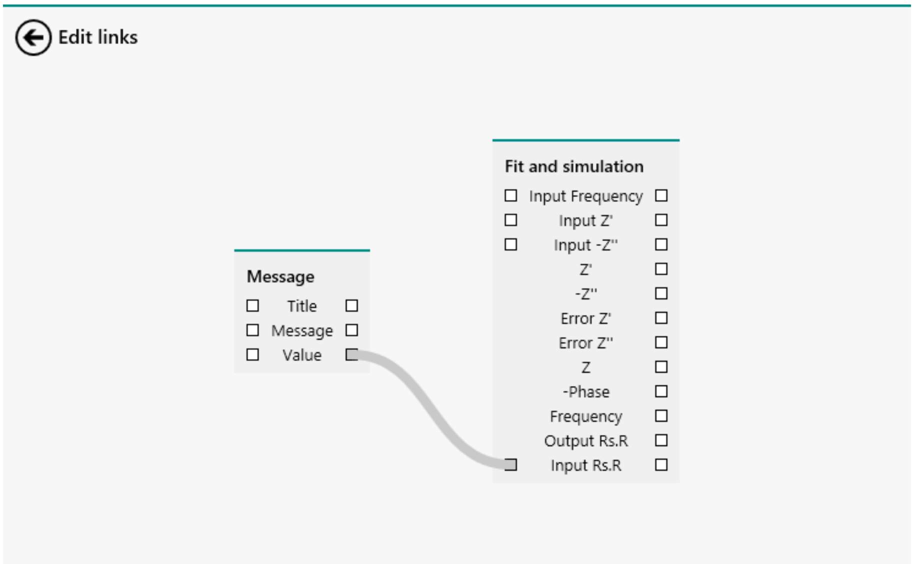


Figure 496 The element property can now be linked

7.9.2.9 Saving equivalent circuits to the Library

Any equivalent circuit can be saved to the **Library**. Saved equivalent circuits will become available to the user from the *Open/Insert Circuit* option as described in *Chapter 7.9.2.4*.

To save the circuit to the Library, select the *Save Circuit* option from the **Circuit** menu *(see Figure 497, page 431)*.

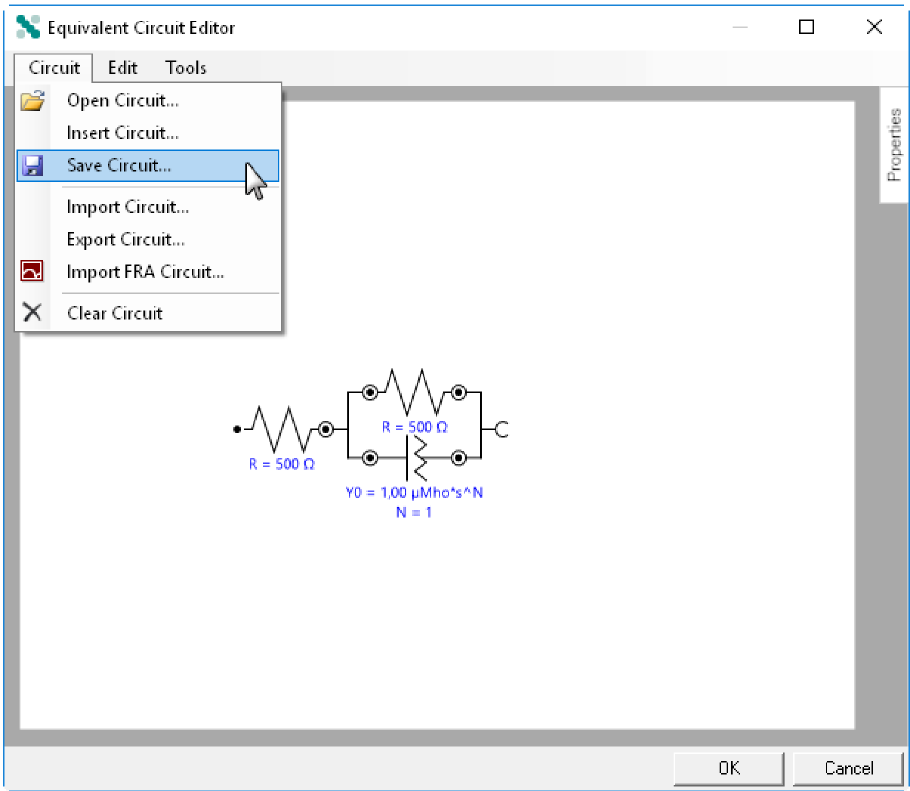


Figure 497 Using the Save circuit option

A new window will be displayed, prompting for the name of the equivalent circuit *(see Figure 498, page 431)*.

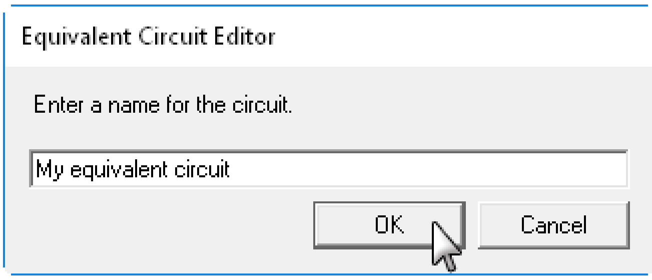


Figure 498 Saving the equivalent circuit

Specify a name for the circuit and press the OK button to save it in the database.

NOTE

The circuit description and the values of the parameters of each element of the circuit are stored in the **Library**.

Once the circuit has been saved, it will be available in the circuit library, on the User-defined circuits tab. It can be opened or inserted into the Equivalent circuit editor window *(see Figure 499, page 432)*.

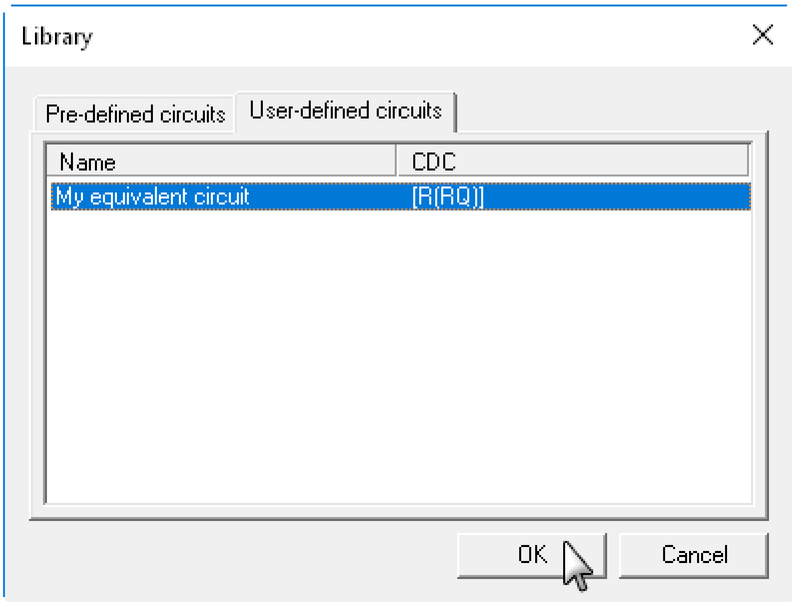


Figure 499 The saved circuits are available under the User-defined circuits tab

NOTE

The equivalent circuits are saved to My Document\NOVA 2.0\Circuits by default.

It is possible to right-click a saved equivalent circuit to rename the circuit or delete it. It is also possible to quickly locate the file on the computer by selecting the Show in Windows Explorer option *(see Figure 500, page 433)*.

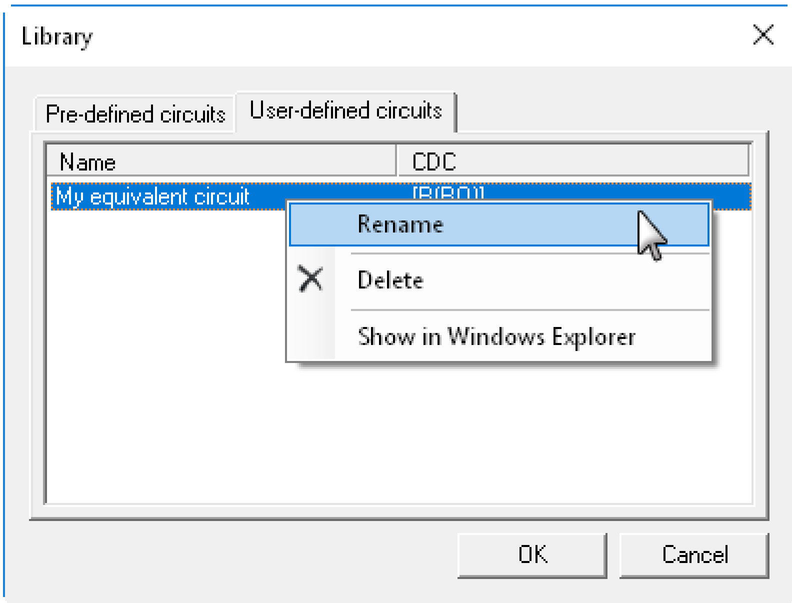
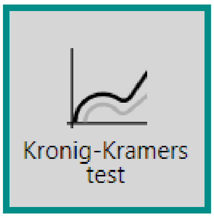


Figure 500 Right-clicking the saved circuit allows renaming, deleting or quick access to the file location

7.9.3 Kronig-Kramers test

	This command allows allows testing a set of measured data using the Kronig-Kramers equations. This test provides an estimation of the 'goodness to fit' of the data set.

The details of the properties of the **Kronig-Kramers test** command are shown in *Figure 501*:

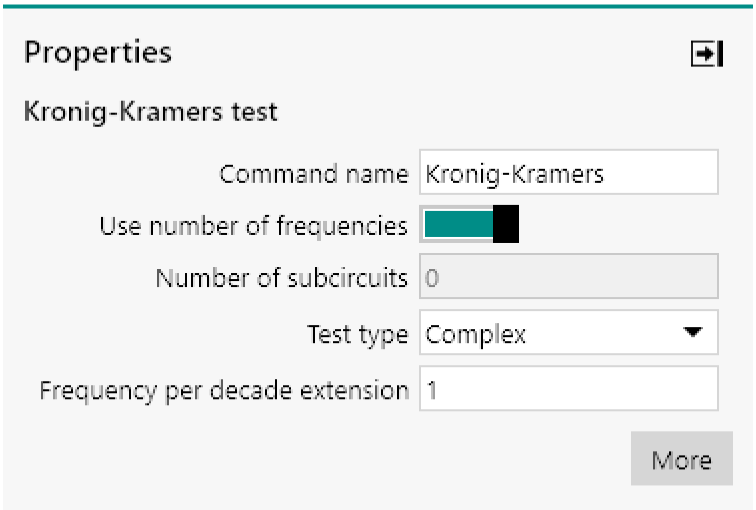


Figure 501 The properties of the Kronig-Kramers test command

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Use number of frequencies:** specifies if the number of (RC) subcircuits is equal to the number of frequencies in the data set, using the provided toggle.
- **Number of subcircuits:** specifies the number of (RC) subcircuits to use in the Kronig-Kramers test. This number must be smaller or equal to the number of data points. It is possible to define this number if the *Use number of frequencies* properties is set to off.
- **Test type:** specifies which part of the data set should be fitted using the distributed equivalent circuit, using the provided drop-down list (Complex, Real, Imaginary).
- **Frequency per decade extension:** defines the Tau-factor used in the calculation.

The Kronig-Kramers relations are mathematical properties which connect the real and imaginary parts of any complex function. These relations are often used to relate the real and imaginary parts of a complex transfer function (like electrochemical impedance, Z). This test can be used to check whether the measured data comply with the assumptions of Kronig-Kramers transformation. These assumptions are:

1. **Linearity:** the response is linear and the perturbation is small.
2. **Stability:** the system does not change with time.
3. **Causality:** the response is only related to the excitation signal.

Additionally, it is also assumed that:

- The system is finite for all values of ω, including zero and infinity.

If the investigated system changes with time due to e.g. aging, temperature change, non-equilibrium initial state etc., the test fails. Failure of Kronig-Kramers test usually means that no good fit can be obtained using the equivalent circuit method. This analysis tool is based on the work of Dr. B.A. Boukamp as published in J. Electrochem. Soc., Vol 142, 6 (June 1995) and coded in the program RCNTRANS by the same author.

The Kramers-Kronig test can be used to check whether the measured system is stable in time and linear. Stability and linearity are a prerequisite for fitting equivalent circuits. If the system changes in time, the data points measured on the beginning of the experiment do not agree with those measured at the end of the experiment. Since stability problems are most likely to be observed in low frequency range, the implementation of electrochemical impedance spectroscopy usually involves scanning from high to low frequency.

During the Kramers-Kronig test, the experimental data points are fitted using a special model circuit which always satisfies the Kramers-Kronig relations. If the measured data set can be represented with this circuit, then the data set should also satisfy Kramers-Kronig assumptions. The special circuit used in the test is a series of RC circuits (for impedance representation). This circuit are shown *Figure 502*.

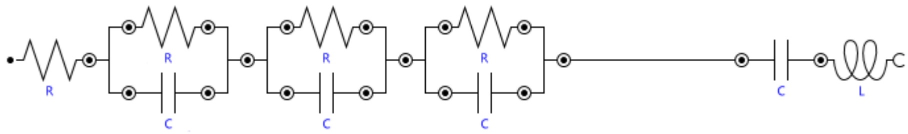


Figure 502 Circuit used in for Kramers-Kronig test on impedance presentation

By default, the number of (RC) subcircuits is equal to the number of data points. If there is a chance that the measured signal was very noisy, the number of subcircuits may be reduced to avoid over-fitting and, consequently, including the noise in the model.

The result of the test is the value of pseudo, χ^2_{ps}, the sum of squares of the relative residuals. In each case the χ^2 for the real and the imaginary part is reported (overall χ^2 is a sum of real and imaginary χ^2). Large χ^2 values indicate that the data quality is low. A small value, on the other hand, usually indicates a good fit.

The equations used in the Kramers-Kronig test are provided below:

$$\chi^2_{ps} = \sum_{i=1}^{n} \frac{\left[Z_{re,i} - Z_{re}(\omega_i)\right]^2 + \left[Z_{im,i} - Z_{im}(\omega_i)\right]^2}{\left|Z(\omega_i)\right|}$$

$$\chi^2_{re} = \sum_{i=1}^{n} \frac{\left[Z_{re,i} - Z_{re}(\omega_i)\right]^2}{\left|Z(\omega_i)\right|}$$

$$\chi^2_{im} = \sum_{i=1}^{n} \frac{\left[Z_{im,i} - Z_{im}(\omega_i)\right]^2}{\left|Z(\omega_i)\right|}$$

What is actually large and small depends on the number and the value of data points. As a rule of thumb, values lower than 10^{-6} usually means an excellent fit, reasonable between 10^{-5} and 10^{-6}, marginal between 10^{-4} and 10^{-5} and bad for even higher values. Moreover, the residuals should be small and randomly distributed around zero.

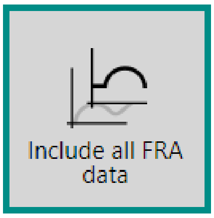
The test can be carried out on real part, imaginary part or both part of admittance/impedance (complex fit). In the case of fit on one part only, the second part of the measured data set is generated using Kramers-Kronig transformation (using the assumption that the system obeys Kramers-Kronig criteria) and then χ^2 for the second part is computed.

In addition to χ^2, the serial or parallel (depending on representation) R, L and C values are computed. These values do not have any special meaning and they simply belong to the set of results of Kramers-Kronig test. In particular, they should not be associated with any serial or parallel elements present in the system or its equivalent circuit representation.

NOTE

The detailed discussion of the Kramers-Kronig test, the theory underlying the choice of properties, and a refined interpretation of the outcomes can be found in B.A. Boukamp, J. Electrochem. Soc. 142, 1885 (1995). It is advised to read this article before this command is used.

7.9.4 Include all FRA data

	This command automatically calculates the admittance data based on measured impedance data.

The details of the properties of the **Include all FRA data** command are shown in *Figure 503*:

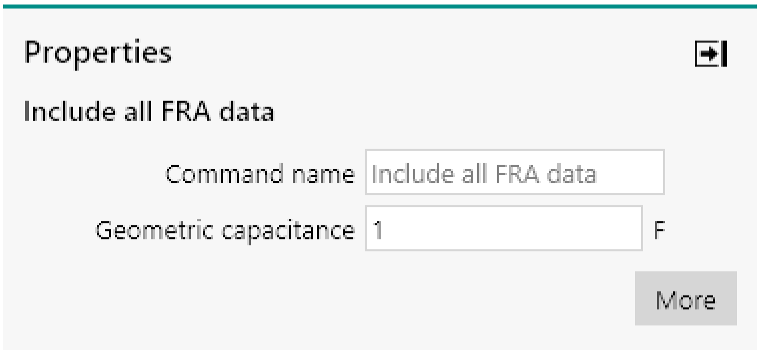


Figure 503 The properties of the Include all FRA data command

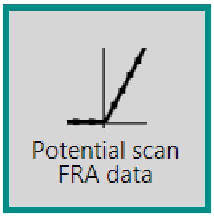
The following properties are available:

- **Command name:** a user-defined name for the command.
- **Geometric capacitance:** specifies the value of the geometric capacitance, ε, in F (default: 1). This value is used in the calculation of the permittivity.

The **Include all FRA data** command can be used to automatically calculate and display additional information that can be derived mathematically from impedance data. This command calculates the following additional values:

- **Real admittance, Y':** the real part of the admittance. This value is calculated according to:
$$Y' = \frac{Z'}{|Z|^2}$$
- **Imaginary admittance, -Y":** the imaginary part of the admittance. This value is calculated according to:
$$-Y" = \frac{-Z"}{|Z|^2}$$
- **Angular frequency, ω:** the angular frequency, in rad/s. This value is calculated according to:
$$\omega = 2\pi f$$
- **Real permittivity, Yε':** the real part of the permittivity. This value is calculated according to:
$$Y\varepsilon' = \Re\left(\frac{Y' - jY"}{j\omega\varepsilon}\right)$$
- **Imaginary permittivity, -Yε":** the imaginary part of the permittivity. This value is calculated according to:
$$-Y\varepsilon" = \Im\left(\frac{Y' - jY"}{j\omega\varepsilon}\right)$$
- **Series capacitance, C_s:** the series capacitance. This value is calculated according to:
$$C_s = -\frac{1}{\omega Z"}$$

7.9.5 Potential scan FRA data

	This command automatically calculates the values required to create a Mott-Schottky plot.

The details of the properties of the **Potential scan FRA data** command are shown in *Figure 504*:

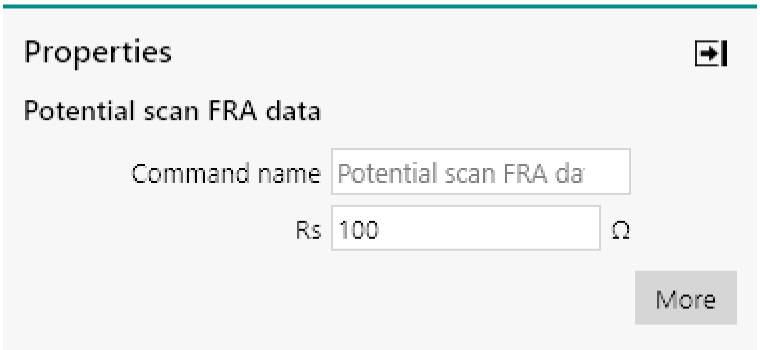


Figure 504 The properties of the Potential scan FRA data command

The following properties are available:

- **Command name:** a user-defined name for the command.
- **Rs:** specifies the value of the serial resistance, Rs, in Ω (default: 100). This value is used in the calculation of the capacitance.

The **Potential scan FRA data** command can be used to calculates several useful values from the impedance data obtained at the different DC potential values. These values are required to create the Mott-Schottky plots.

This command calculates the following additional values:

- **Angular frequency, ω:** the angular frequency, in rad/s. This value is calculated according to:
  $\omega = 2\pi f$
- $\frac{1}{C_S^2}$**:** the inverted squared value of the capacitance determined based on a serial equivalent (R_S-C_S) circuit. This value is calculated according to:
  $$\frac{1}{C_S^2} = (-\omega Z'')^2$$
- $\frac{1}{C_P^2}$**:** the inverted squared value of the capacitance determined based on a parallel equivalent (R_S-R_P/C_P) circuit. This value is calculated according to:
  $$\frac{1}{C_P^2} = \left(\frac{\left(-Z''^2 + (Z' - R_S)^2\right)\omega}{-Z''}\right)^2$$

The calculated values can then be plotted against the applied DC potential in order to build a Mott-Schottky plot.

7.10 Metrohm devices commands

Commands located in the **Metrohm devices** group can be used to control Metrohm devices connected to the computer.

The available commands are represented by a shortcut icon *(see Figure 505, page 439)*.

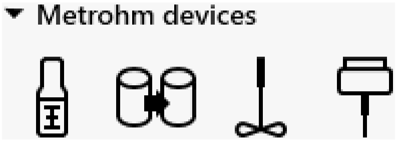


Figure 505 The Metrohm devices commands

The following commands are available:

- **Dosino:** a command that can be used to control a Metrohm 800 Dosino connected to the host computer *(see Chapter 7.10.1, page 439)*.
- **Sample Processor:** a command that can be used to control a Metrohm 814, 815 or 858 Sample Processor connected to the host computer *(see Chapter 7.10.2, page 445)*
- **Stirrer:** a command that can be used to control a Metrohm 801 Magnetic Stirrer or a Metrohm 802 Rod Stirrer or Metrohm 741 Magnetic Stirrer connected to a 804 Titration Stand connected to the host computer *(see Chapter 7.10.3, page 452)*.
- **Remote I/O:** a command that can be used to control a Metrohm 6.2148.010 Remote Box connected to the host computer *(see Chapter 7.10.4, page 453)*.

7.10.1 Dosino

	This command can be used to control the **Metrohm 800 Dosino** connected to the computer.

The **Dosino** command can be used in six different modes, which can be selected using the provided drop-down list *(see Figure 506, page 440)*:

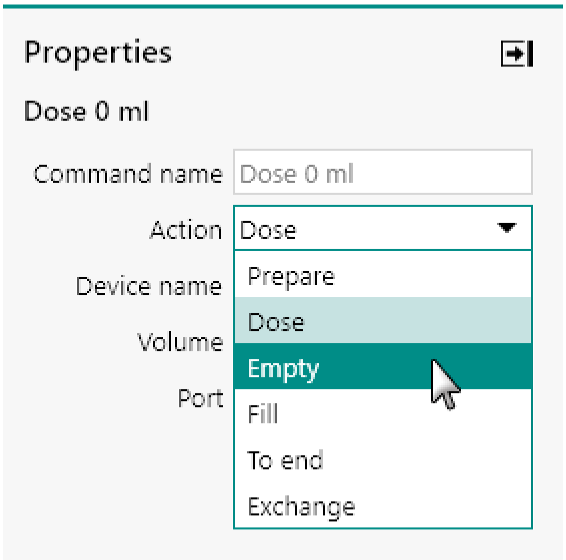


Figure 506 Six modes are provided by the Dosino command

1. Prepare
2. Dose (default mode)
3. Empty
4. Fill
5. To end
6. Exchange

NOTE

The **Dosino** command description in the procedure editor is dynamically adjusted in function of the specified mode.

7.10.1.1 Dose

The *Dose* mode of the **Dosino** command can be used to deliver a user-defined volume through the specified port. If the specified volume exceeds the volume of the dosing cylinder used, the Dosino will be refilled and the dosing will resume until the specified volume is delivered.

NOTE

The Fill port, defined in the Dosino hardware setup, is used to refill the Dosino when dosing the required volume, if applicable. See *Chapter 5.5.1.2* for more information.

If a negative volume is specified, the Dosino will aspirate the required volume through the specified port.

The following properties are available when the **Dosino** command is used in the *Dose* mode *(see Figure 507, page 441)*:

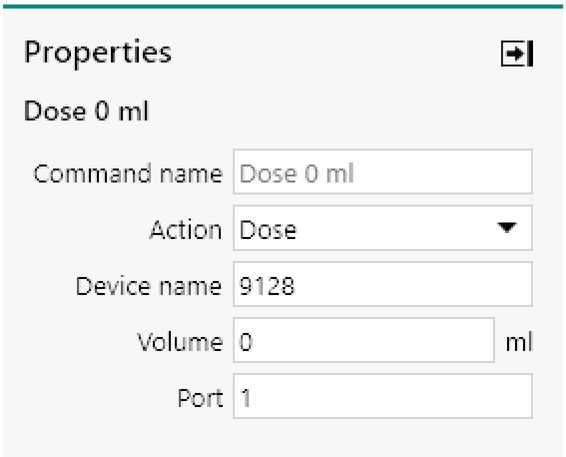


Figure 507 Dose mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying name of the Dosino.
- **Volume:** the volume to dose or aspirate, in ml.
- **Port:** the port used to dose or aspirate the specified volume.

7.10.1.2 Prepare

The *Prepare* mode of the **Dosino** command can be used to prepare the Dosino by rinsing and filling the connected tubes and the dosing cylinder. The tubes of the Dosino should be freed from air bubbles at least once a day by carrying out a full prepare cycle. This process will take time depending on the length of the tubes.

During the preparation process, the dosing cylinder as well as the connected tubings are completely filled. The volume required to fill the tubings is determined based on the parameters specified in the hardware setup of the Dosino.

NOTE

The Fill port, defined in the Dosino hardware setup, is used to refill the Dosino during the preparation process. See *Chapter 5.5.1.2* for more information.

NOTE

Ports that are set to *inactive* in the Dosino hardware setup are not used in the preparation process. See *Chapter 5.5.1.2* for more information.

The following properties are available when the **Dosino** command is used in the *Prepare* mode *(see Figure 508, page 442)*:

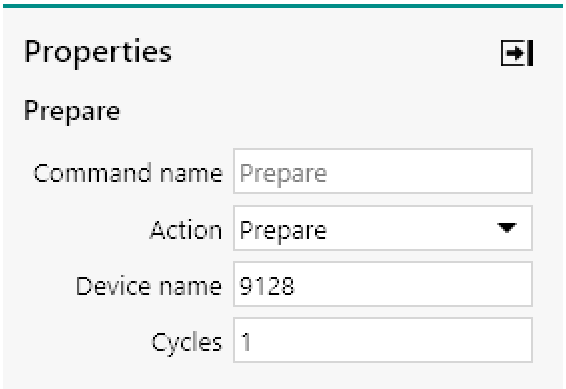


Figure 508 Prepare mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying name of the Dosino.
- **Cycles:** the number of cycles used to prepare the Dosino.

NOTE

It is recommended to use the *Prepare* mode at the beginning of any procedure using a Dosino.

7.10.1.3 Empty

The *Empty* mode of the **Dosino** command can be used to completely empty the dosing cylinder and the tubes connected to the Dosino. The air required to displaced the liquid in the tubes is aspirated via the vent.

NOTE

The liquid in the dosing cylinder is ejected through the Dosing port specified in the Dosino hardware setup. See *Chapter 5.5.1.2* for more information.

NOTE

Ports that are set to *inactive* in the Dosino hardware setup are not used in the emptying process.

The following properties are available when the **Dosino** command is used in the *Empty* mode *(see Figure 509, page 443)*:

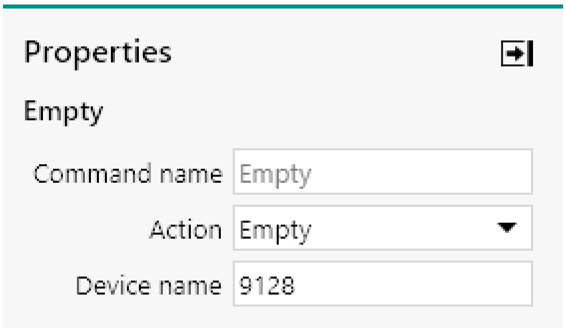


Figure 509 Empty mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying name of the Dosino.

7.10.1.4 Fill

The *Fill* mode of the **Dosino** command can be used to completely refill the dosing cylinder of the specified Dosino. The liquid is aspirated through the Fill port defined in the Dosino hardware setup *(see Chapter 5.5.1.2, page 141)*.

The following properties are available when the **Dosino** command is used in the *Fill* mode *(see Figure 510, page 443)*:

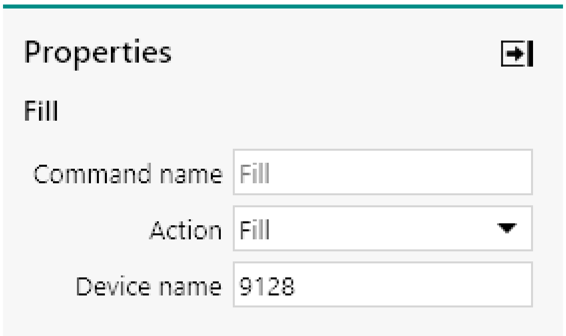


Figure 510 Fill mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying name of the Dosino.

7.10.1.5 To end

The *To end* mode of the **Dosino** command can be used to eject the contents of the dosing cylinder through the specified port. The piston stops at the specified end volume. This is useful for pipetting functions or for removing air bubbles from the dosing cylinder.

The following properties are available when the **Dosino** command is used in the *To end* mode *(see Figure 511, page 444)*:

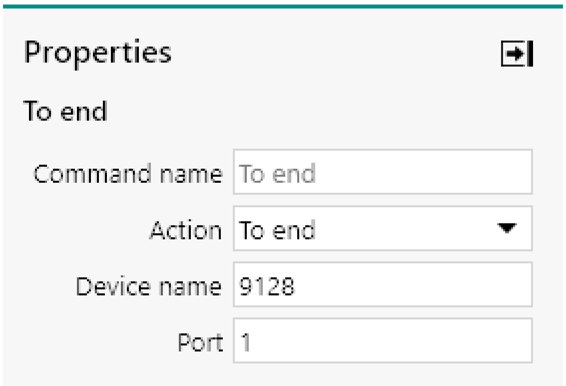


Figure 511 Exchange mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying name of the Dosino.
- **Port:** the port used to perform the to end action.

7.10.1.6 Exchange

The *Exchange* mode of the **Dosino** command can be used to prepare a dosing cylinder for exchange. The dosing cylinder is filled and the stopcock is moved to the exchange position. The cylinder is filled by aspirating the necessary volume via the Fill port specified in the Dosino hardware setup *(see Chapter 5.5.1.2, page 141)*.

The following properties are available when the **Dosino** command is used in the *Exchange* mode *(see Figure 512, page 444)*:

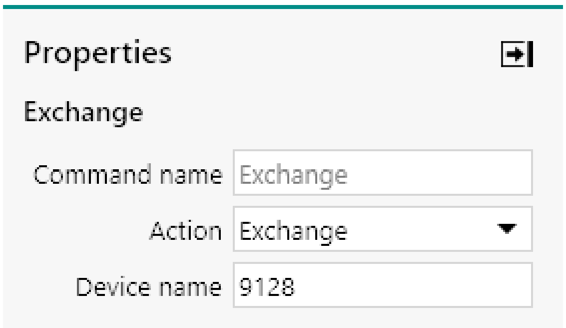
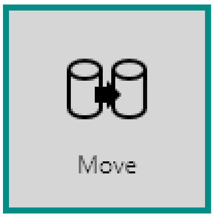


Figure 512 Exchange mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying name of the Dosino.

7.10.2 Sample Processor

	This command can be used to control the **Metrohm 814**, **815** or **858** Sample Processor connected to the computer.

The **Sample Processor** command can be used in eight different modes, which can be selected using the provided drop-down list *(see Figure 513, page 445)*:

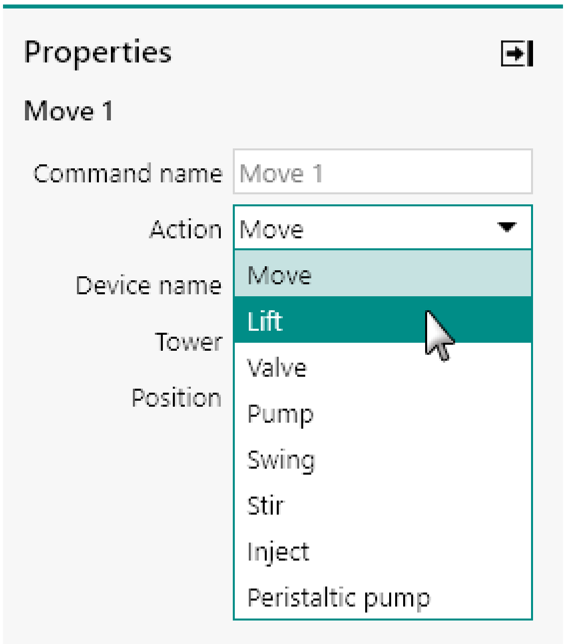


Figure 513 Eight modes are provided by the Sample Processor command

1. Move (default mode)
2. Lift
3. Valve
4. Pump
5. Swing
6. Stirrer
7. Inject
8. Peristaltic pump

> **NOTE**
>
> The last two modes are only available when using the **Metrohm 858 Professional Sample Processor**.

> **NOTE**
>
> The **Sample Processor** command description in the procedure editor is dynamically adjusted in function of the specified mode.

7.10.2.1 Move

The *Move* mode of the **Sample Processor** command can be used to change the position of the sample rack, relative to the Sample Processor tower, to the required position.

The following properties are available when the **Sample Processor** command is used in the *Move* mode *(see Figure 514, page 446)*:

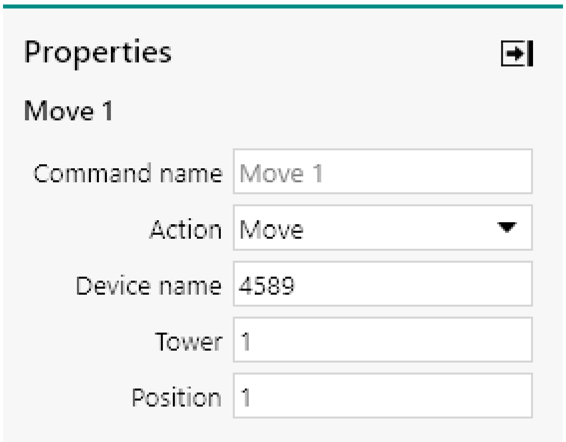


Figure 514 Move mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying name of the Sample Processor.
- **Tower:** specifies which tower is used by the command.
- **Position:** specifies the position of the rack. This value can be be specified between 1 and the maximum number of positions available on the rack. The maximum number of position depends on the type of sample rack defined in the Sample Processor hardware setup *(see Chapter 5.5.2.2, page 148)*.

NOTE

When the sample rack is fitted with several rows of samples and the Sample Processor is fitted with a swing arm, the swing arm will be operated while the **Sample Processor** command is executed in the *Move* mode, if required.

NOTE

When the Sample Processor lift is not in the shift position (0 mm, top of the tower), the lift will be first moved to the shift position before the rack is moved.

7.10.2.2 Lift

The *Lift* mode of the **Sample Processor** command can be used to set the position of the lift on the specified Sample Processor tower. The position of the lift can be specified between 0 mm (top of the tower) and the maximum position defined in the Sample Processor hardware setup *(see Chapter 5.5.2.2, page 148)*.

The following properties are available when the **Sample Processor** command is used in the *Lift* mode *(see Figure 515, page 447)*:

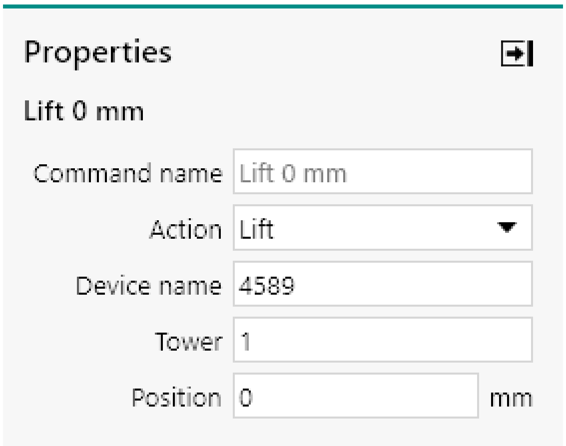


Figure 515 Lift mode properties

- **Device name:** the identifying name of the Sample Processor.
- **Tower:** specifies which tower is used by the command.
- **Position:** specifies the position of the lift, in mm, with respect to the top of the tower.

7.10.2.3 Valve

The *Valve* mode of the **Sample Processor** command can be used to activate or deactivate valves mounted on the back plane of a tower.

NOTE

Valves remain on or off until modified by the procedure or through the Sample processor manual control panel *(see Chapter 5.5.2, page 145)*.

The following properties are available when the **Sample Processor** command is used in the *Valve* mode *(see Figure 516, page 448)*:

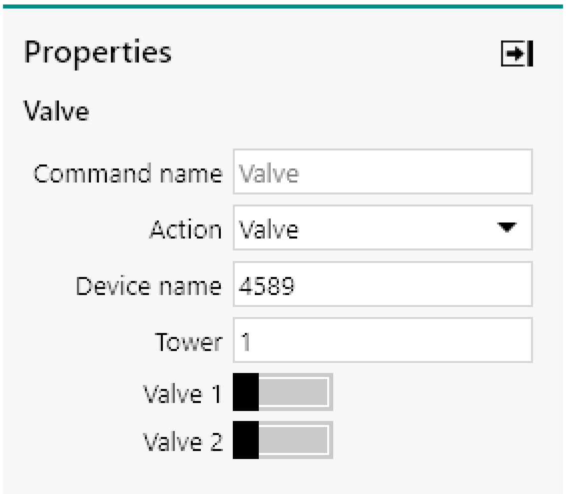


Figure 516 Valve mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying name of the Sample Processor.
- **Tower:** specifies which tower is used by the command.
- **Valve 1:** specifies the state of valve 1 through a dedicated toggle.
- **Valve 2:** specifies the state of valve 2 through a dedicated toggle.

7.10.2.4 Pump

The *Pump* mode of the **Sample Processor** command can be used to activate or deactivate pumps mounted on the back plane of a tower or connected to the tower.

NOTE

Pumps remain on or off until modified by the procedure or the Sample processor manual control panel *(see Chapter 5.5.2, page 145)*.

The following properties are available when the **Sample Processor** command is used in the *Pump* mode *(see Figure 517, page 449)*:

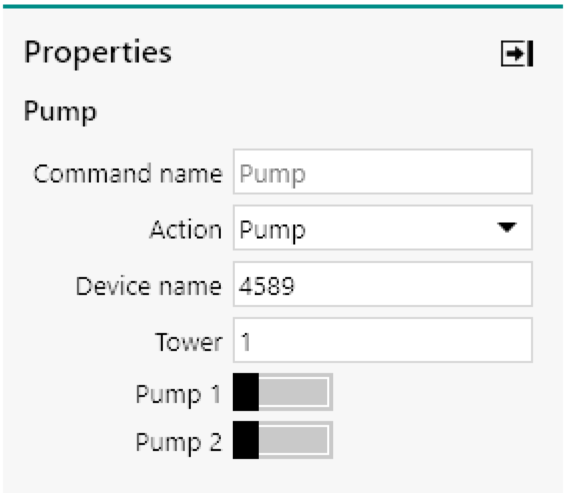


Figure 517 Pump mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying name of the Sample Processor.
- **Tower:** specifies which tower is used by the command.
- **Pump 1:** specifies the state of pump 1 through a dedicated toggle.
- **Pump 2:** specifies the state of pump 2 through a dedicated toggle.

7.10.2.5 Swing

The *Swing* mode of the **Sample Processor** command can be used to change the position of the swing head installed on the specified Sample Processor tower.

The following properties are available when the **Sample Processor** command is used in the *Swing* mode *(see Figure 518, page 450)*:

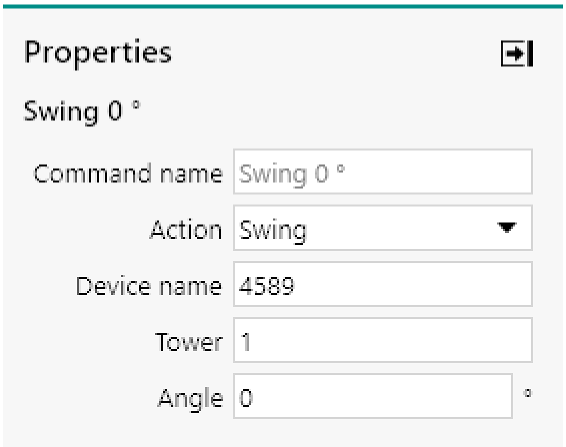


Figure 518 Swing mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying name of the Sample Processor.
- **Tower:** specifies which tower is used by the command.
- **Angle:** specifies the angle of the swing arm with respect to the tower, in °. The range of value depends on the type of swing head mounted on the swing arm.

7.10.2.6 Stir

The *Stirrer* mode of the **Sample Processor** command can be used to control the rotation rate of a **Metrohm 802 Rod Stirrer** or **Metrohm 741 Magnetic Stirrer** connected to the Sample Processor tower.

The following properties are available when the **Sample Processor** command is used in the *Stirrer* mode *(see Figure 519, page 450)*:

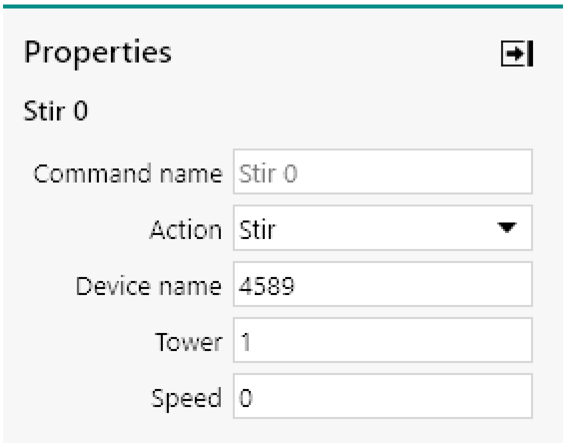


Figure 519 Stirrer mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying name of the Sample Processor.
- **Tower:** specifies which tower is used by the command.
- **Rotation rate:** the rotation rate, specified between -15 and 15. A value of 0 will stop the stirrer.

7.10.2.7 Inject

The *Inject* mode of the **Sample Processor** command can be used to set the position of the injection valve. The connections to the injection valve can be toggled between the **Fill** position and the **Inject** position *(see Figure 520, page 451)*.

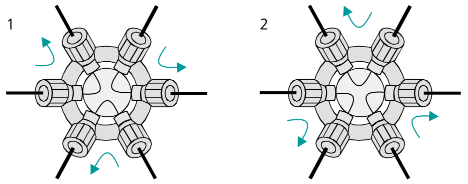


Figure 520 The positions of the injection valve

1	**Fill position**	**2**	**Inject position**

NOTE

This mode can only be used in combination with the **Metrohm 858 Professional Sample Processor** fitted with the injection valve.

The following properties are available when the **Sample Processor** command is used in the *Inject valve* mode *(see Figure 520, page 451)*:

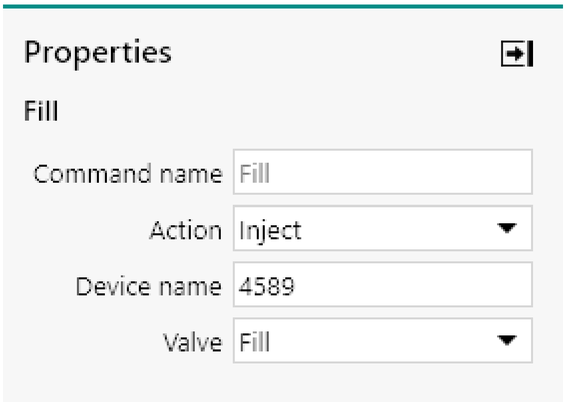


Figure 521 Inject valve mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying name of the Sample Processor.
- **Position:** specifies the position of the inject valve, using the provided drop-down list.

7.10.2.8 Peristaltic pump

The *Peristaltic pump* mode of the **Sample Processor** command can be used to control the peristaltic pump installed on the Sample Processor.

> **NOTE**
>
> This mode can only be used in combination with the **Metrohm 858 Professional Sample Processor** fitted with the peristaltic pump.

The following properties are available when the **Sample Processor** command is used in the *Peristaltic pump* mode *(see Figure 522, page 452)*:

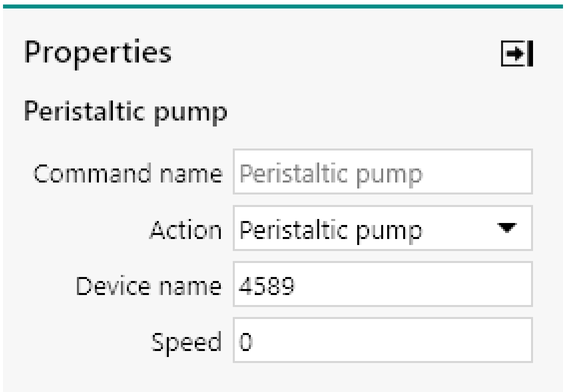
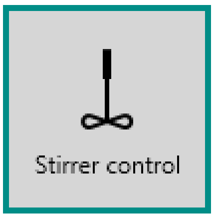


Figure 522 Peristaltic pump mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying name of the Sample Processor.
- **Rotation rate:** the rotation rate, specified between -15 and 15. A value of 0 will stop the pump.

7.10.3 Stirrer

	This command can be used to control the **Metrohm 801 Magnetic Stirrer** or **Metrohm 804 Titration Stand** in combination with the **Metrohm 802 Rod Stirrer** connected to the computer.

The details of the command properties of the **Stirrer** command are shown in *Figure 523*:

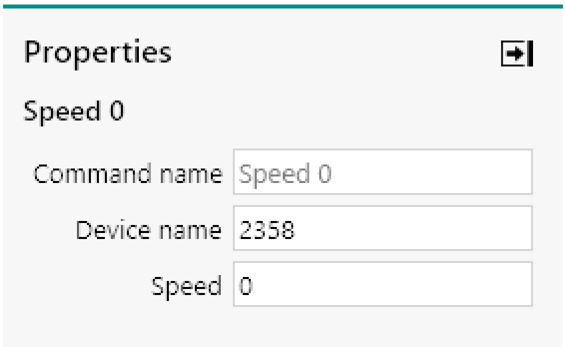


Figure 523 The properties of the Stirrer command

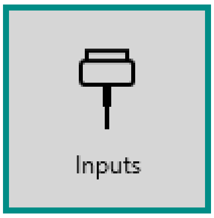
The following properties are available:

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying name of the Stirrer.
- **Rotation rate:** the rotation rate, specified between -15 and 15. A value of 0 will stop the stirrer.

NOTE

The **Stirrer** command description in the procedure editor is dynamically adjusted in function of the specified value.

7.10.4 Remote

	This command can be used to control the **Metrohm 6.2148.010 Remote Box** connected to the computer.

The **Remote** command can be used in two different modes, which can be selected using the provided drop-down list *(see Figure 524, page 454)*:

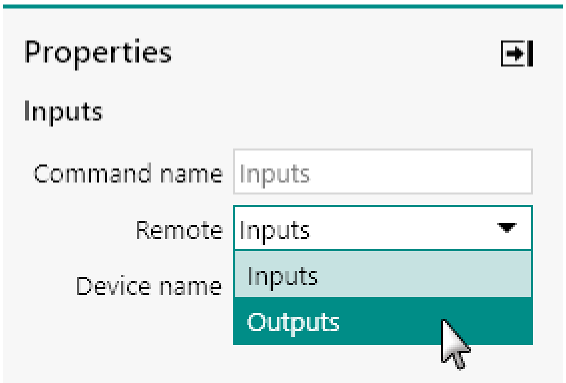


Figure 524 Two modes are provided by the Remote command

1. Remote inputs (default mode)
2. Remote outputs

NOTE

The **Remote** command description in the procedure editor is dynamically adjusted in function of the specified mode.

CAUTION

The **Metrohm 6.2148.010 Remote Box** can also be used in combination with the **Wait** command *(see Chapter 7.2.4, page 225)*. When the Remote Box is connected to the computer, the **Wait** command provides one additional mode, *Wait for Remote Inputs*, which uses the eight input lines provided by the Remote Box.

7.10.4.1 Remote inputs

The *Remote inputs* mode of the **Remote** command can be used to read the state of the 8 input lines (numbered IN7 to IN0). The state of each input line can be either 'low' or 'high' state, represented by a 0 or a 1, respectively.

The following properties are available when the **Remote** command is used in the *Remote inputs* mode *(see Figure 525, page 455)*:

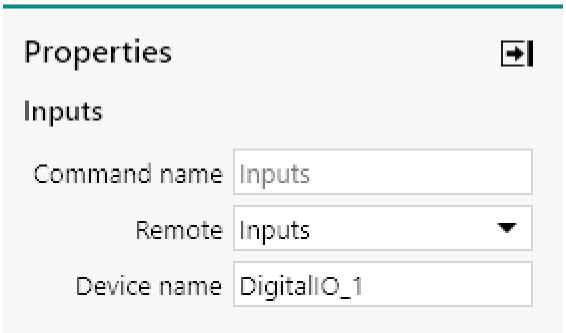


Figure 525 Remote inputs mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying name of the Remote Box.
- **Inputs:** specifies the state of the 8 input lines, during the execution of the procedure. The state is returned as a string of 8 characters, consisting of '0' and '1', representing the state of the input lines, from IN7 to IN0.

NOTE

The state of the 8 input lines of the Remote Box is determined when the command is executed.

7.10.4.2 Remote outputs

The *Remote outputs* mode of the **Remote** command can be used to set the state of the 14 output lines (numbered OUT13 to OUT0). The state of each output line can set to either 'low' or 'high' state, represented by a 0 or a 1, respectively.

The following properties are available when the **Remote** command is used in the *Remote outputs* mode *(see Figure 526, page 455)*:

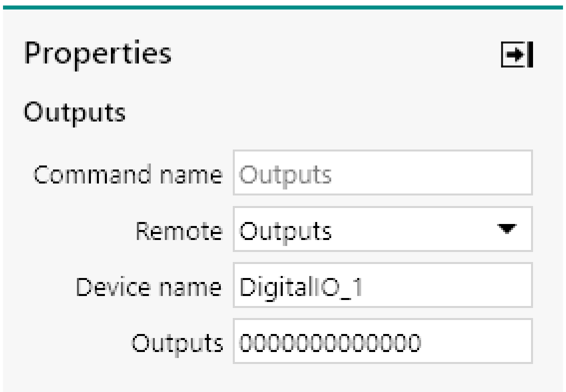


Figure 526 Remote outputs mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying name of the Remote Box.

- **Outputs:** specifies the state of the 14 output lines is specified as a 14 character string, consisting of '0' and '1', representing the state of the output lines, from OUT13 to OUT0.

> **NOTE**
>
> The state of the 14 output lines of the Remote Box is persistent until changed or until the Remote Box is powered down.

7.11 External devices commands

Commands located in the **External devices** group can be used to communicate with supported external devices connected to the computer.

The available commands are represented by a shortcut icon *(see Figure 527, page 456)*.

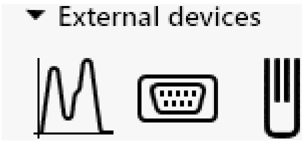
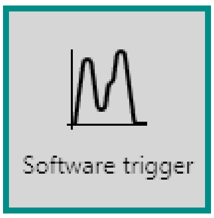


Figure 527 The External devices commands

The following commands are available:

- **Spectroscopy:** a command which can be used to control Autolab (or Avantes) spectrophotometers connected to the computer through a USB connection *(see Chapter 7.11.1, page 456)*.
- **RS232:** a command which can be used to control an external device through the RS-232 protocol *(see Chapter 7.11.2, page 463)*.
- **RHD:** a command which can be used to control Autolab RHD Microcell HC controllers connected to the computer through a RS232 connection *(see Chapter 7.11.3, page 467)*.

7.11.1 Spectroscopy

	This command can be used to interface to an external Autolab (or Avantes) spectrophotometer connected to the computer through a USB connection.

The **Spectroscopy** command can be used in two different modes, which can be selected using the provided drop-down list *(see Figure 528, page 457)*:

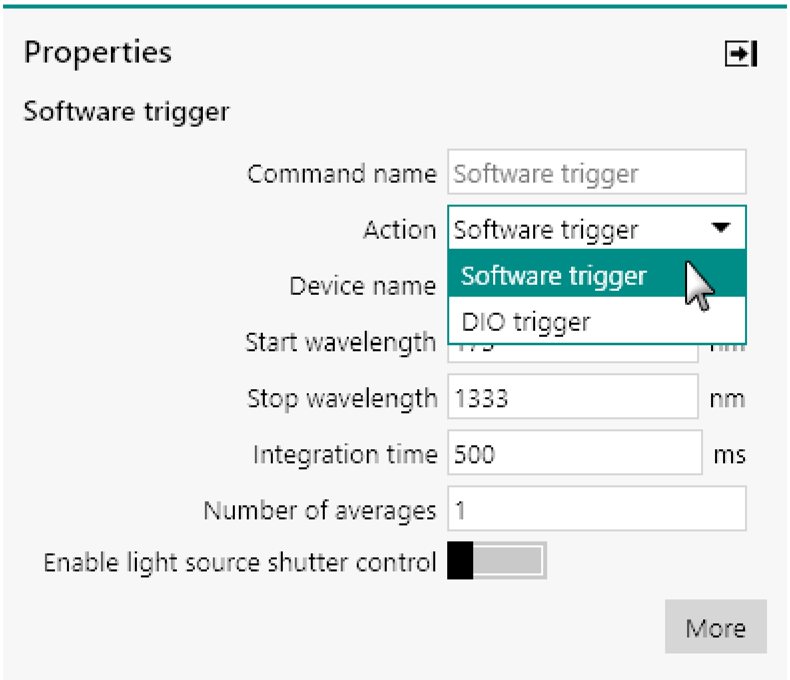


Figure 528 Two modes are provided by the Spectroscopy command

1. Software trigger (default mode)
2. DIO trigger

NOTE

The **Spectroscopy** command description in the procedure editor is dynamically adjusted in function of the specified mode.

7.11.1.1 Software trigger

The following properties are available when the command is used in the *Software trigger* mode *(see Figure 529, page 458)*:

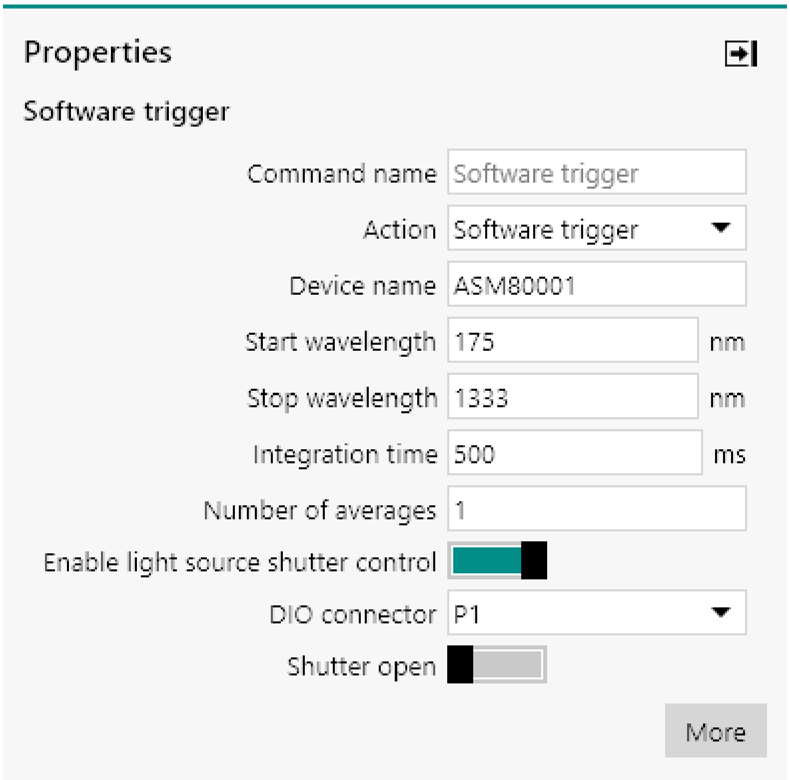


Figure 529 Software trigger mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** specifies the name of the spectrophotometer used in the measurement.
- **Start wavelength:** specifies the start wavelength used by the spectrophotometer, in nm.
- **Stop wavelength:** specifies the start wavelength used by the spectrophotometer, in nm.
- **Integration time:** specifies the integration time used by the spectrophotometer, in ms.
- **Number of averages:** specifies the number of averages used by the spectrophotometer.
- **Enable light source shutter control:** specifies if the command should control the light source shutter position using the provided  toggle. This requires a physical connection between the Autolab potentiostat/galvanostat and the spectrophotometer. It is also necessary to set the shutter control to TTL mode on the connected light source.
- **DIO connector:** specifies which DIO connector is used to interface to the light source, using the provided drop-down list. This property is only visible if the *Enable light source shutter control* property is set to on. For the PGSTAT101, M101, PGSTAT204 and M204 instruments, this property is not shown.

- **Shutter open:** specifies the state of the light source shutter, using the provided toggle. This property is only visible if the *Enable light source shutter control* property is set to on.

NOTE

The light source shutter will remains in the specified state until changed.

CAUTION

This mode requires a physical connection between the light source and the Autolab DIO connector if the *Enable light source shutter control* property is set to on. Please refer to the Spectrophotometer User Manual for more information.

7.11.1.2 DIO trigger

The following properties are available when the command is used in the *DIO trigger* mode *(see Figure 530, page 459)*:

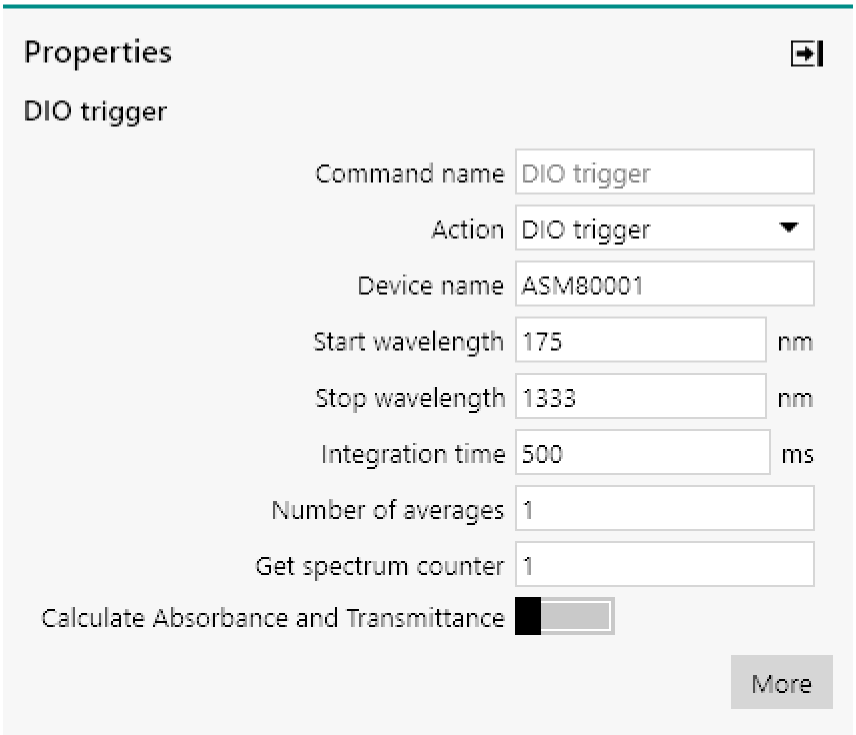


Figure 530 DIO trigger mode properties

- **Command name:** a user-defined name for the command.

- **Device name:** specifies the name of the spectrophotometer used in the measurement.
- **Start wavelength:** specifies the start wavelength used by the spectrophotometer, in nm.
- **Stop wavelength:** specifies the start wavelength used by the spectrophotometer, in nm.
- **Integration time:** specifies the integration time used by the spectrophotometer, in ms.
- **Number of averages:** specify the number of averages used by the spectrophotometer.
- **Get spectrum counter:** the counter value used by the triggering command.
- **Calculate Absorbance and Transmittance:** specifies if the measured values should be converted to absorbance and transmittance using values of a dark spectrum and reference spectrum using the provided toggle.

If the **Calculate Absorbance and Transmittance** property is on, it is necessary to link two single spectra to the **Spectroscopy** command. Two input anchoring points will be added to the command *(see Figure 531, page 461)*.

Figure 531 Dark and reference spectra can be linked to the Spectroscopy command

Using these two anchoring points, a dark spectrum and a reference spectrum can be linked to the **Spectroscopy** command in order to convert the measured values to absorbance and transmittance.

These values are calculated using the measured values ($S_{Measured}$), the linked *Dark* spectrum values(S_{Dark}) and the linked *Reference* spectrum values ($S_{Reference}$) according to:

- **Absorbance:**

$$A = -\log\left(\frac{S_{Measured} - S_{Dark}}{S_{Reference} - S_{Dark}}\right)$$

- **Transmittance:**

$$T = 100 \cdot \left(\frac{S_{Measured} - S_{Dark}}{S_{Reference} - S_{Dark}}\right)$$

CAUTION

The linked dark and reference spectra must be measured in the same conditions as those of the **Spectroscopy** command they are linked to.

CAUTION

This mode requires a physical connection between the spectrophotometer and the Autolab DIO connector. Please refer to the Spectrophotometer User Manual for more information.

To use the **Spectroscopy** command in *DIO trigger* mode in a NOVA procedure, the command needs to be stacked onto the electrochemical measurement command that it is used with. *Figure 532* provides an example.

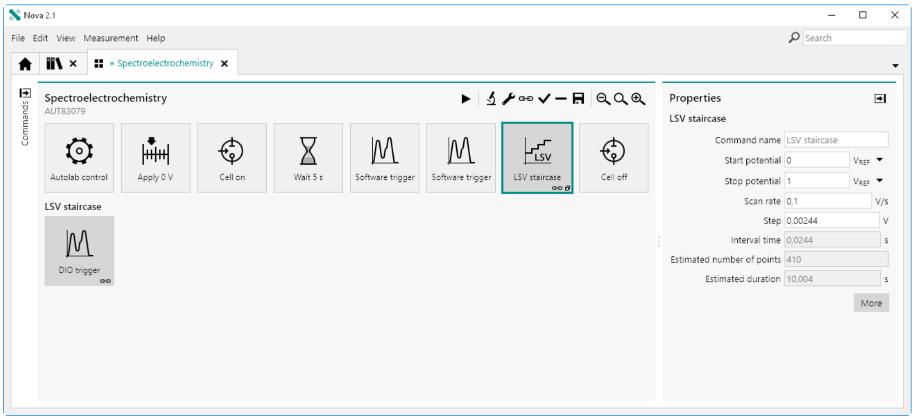


Figure 532 Stacking the spectroscopy command on a measurement command

Using this configuration, the **Spectroscopy** command used in *DIO trigger* mode will be executed whenever the parent measurement command (**LSV staircase** in *Figure 532*) will send a DIO trigger to the spectrophotometer.

NOTE

More information on the stacking of commands can be found in *Chapter 10.12*.

At the end of a measurement, the electrochemical data will be provided by the parent measurement command and the spectroelectrochemical data will be provided by the **Spectroscopy** command *(see Figure 533, page 463)*.

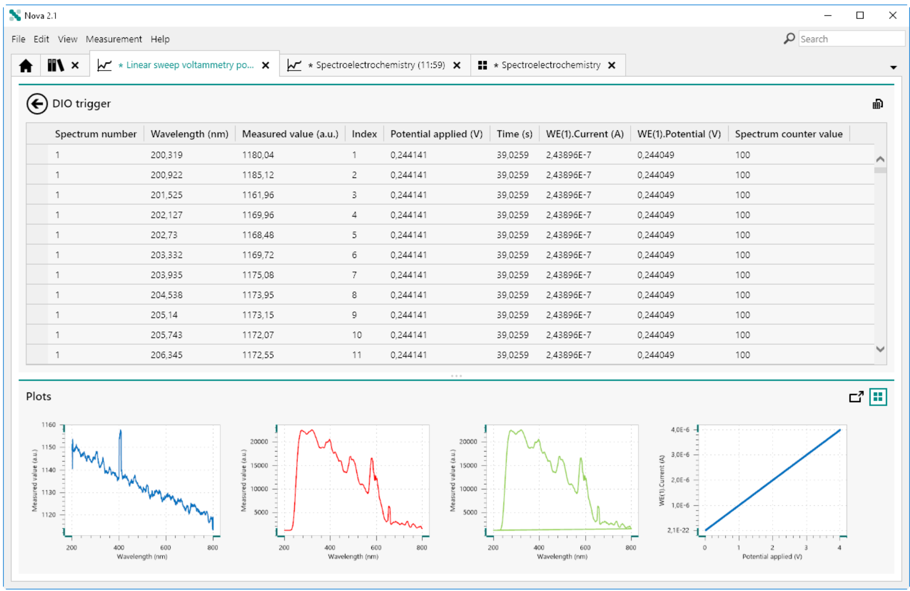


Figure 533 The spectroscopy and electrochemistry data is available in the Spectroscopy command

7.11.2 External device control

	This command can be used to interface to external instruments connected through the RS-232 standard.

The RS-232 standard describes a communication method where information is sent bit by bit on a physical channel. The information must be broken up in data words. The length of a data word is variable (usually between 5 and 8 bits). For proper transfer additional bits are added for synchronization and error checking purposes.

NOTE

Interfacing to external devices through the RS-232 standard requires a properly configured COM port on the computer.

The **External device control** command can be used in four different modes, which can be selected using the provided drop-down list *(see Figure 534, page 464)*:

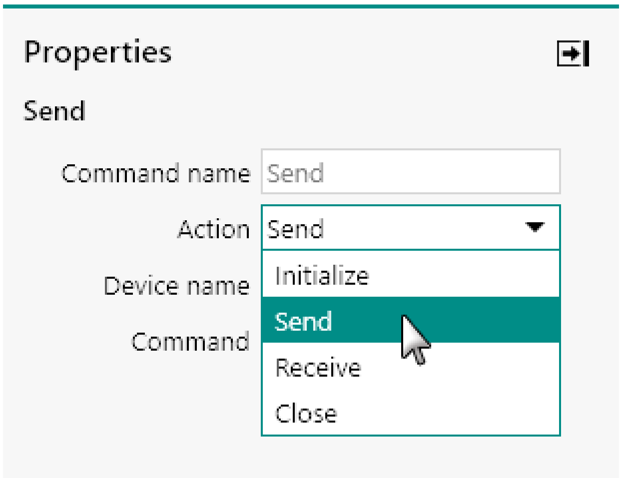


Figure 534 Four modes are provided by the External device control command

1. Initialize
2. Send (default mode)
3. Receive
4. Close

NOTE

The **External device control** command description in the procedure editor is dynamically adjusted in function of the specified mode.

7.11.2.1 Initialize

The following properties are available when the command is used in the *Initialize* mode *(see Figure 535, page 465)*:

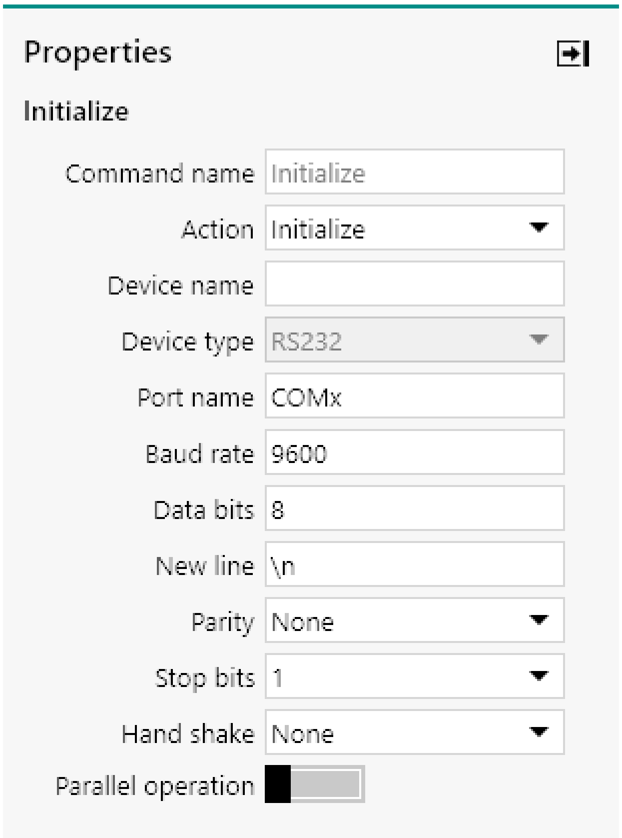


Figure 535 Initialize mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** specifies the name of the device to initialize at the beginning of the measurement. The device name must be unique and will be used to identify the connected device in NOVA.
- **Port name:** specifies the COM port used to control the external device (replace *x* with the COM port number).
- **Baud rate:** specifies the baud rate used to communicate with the external device.
- **Data bits:** specifies the number of data bits (8 by default).
- **New line:** specify the character used to create a new line (*n* by default).
- **Parity:** specifies the parity, using the provided drop-down list (None, Odd, Even, Mark, Space).
- **Stop bits:** specifies the number of stop bits, using the provided drop down list (0, 1, 2, 1.5).
- **Handshake:** specifies the handshaking mode for the communication, using the provided drop down list (None, X on X off, Request to send, Request to send X on X off)

CAUTION

Before an external instrument can be used, the communication with the instrument must be initialized properly, using the *Initialize* mode.

7.11.2.2 Send

The following properties are available when the command is used in the *Send* mode *(see Figure 536, page 466)*:

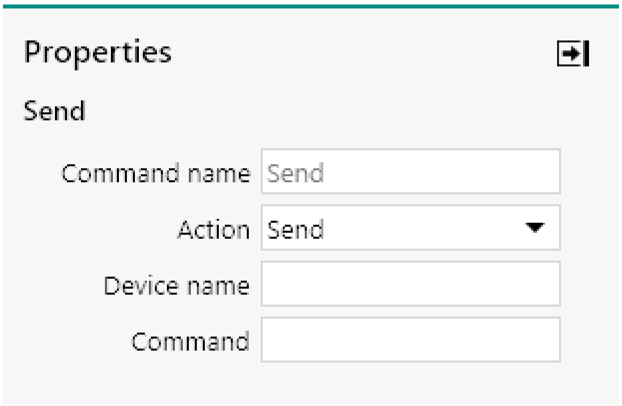


Figure 536 Send mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the name of the device to which a data string is sent.

7.11.2.3 Receive

The following properties are available when the command is used in the *Receive* mode *(see Figure 537, page 466)*:

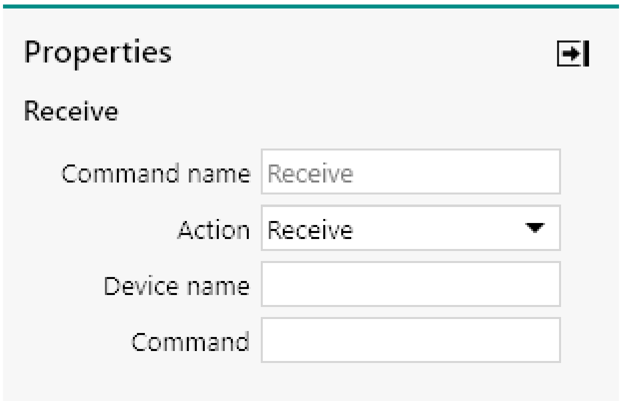


Figure 537 Receive mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the name of the device from which a data string is received.

- **Command:** a string defining the format of the expected string from the external device with placeholders ({0}, {1}, ...) for (variable) parameters in the string.

NOTE

To receive a data string from the external device, the *Send* mode is first used to send a specific data string to the external device. The *Receive* is then added to the procedure to read the reply string from the external device.

7.11.2.4 Close

The following properties are available when the command is used in the *Close* mode *(see Figure 538, page 467)*:

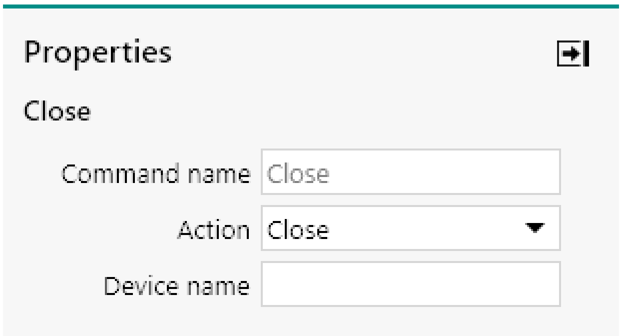


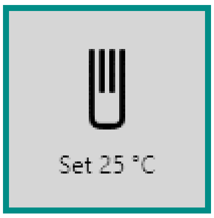
Figure 538 Close mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the name of the device to close at the end of the measurement.

CAUTION

Always use the *Close* mode to close the communication to an external device at the end of each measurement.

7.11.3 RHD control

	This command can be used to control the **Autolab RHD Microcell HC** connected to the computer.

The **RHD control** command can be used in two different modes, which can be selected using the provided drop-down list *(see Figure 539, page 468)*:

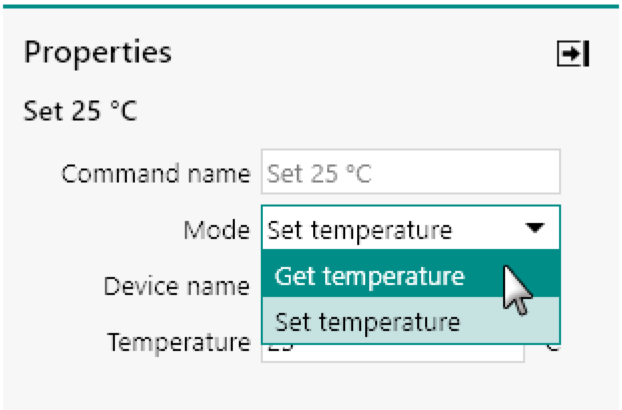


Figure 539 Two modes are provided by the RHD control command

1. Get temperature
2. Set temperature (default mode)

NOTE

The **RHD control** command description in the procedure editor is dynamically adjusted in function of the specified mode.

7.11.3.1 Get temperature

The following properties are available when the **RHD control** command is used in the *Get temperature* mode *(see Figure 540, page 468)*:

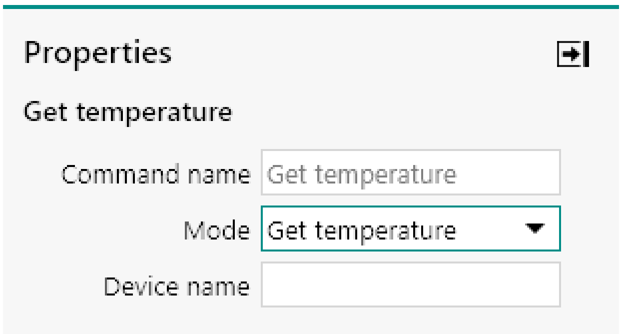


Figure 540 Get temperature mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying device name of the Autolab RHD Microcell HC controller.

7.11.3.2 Set temperature

The following properties are available when the **RHD control** command is used in the *Set temperature* mode *(see Figure 541, page 469)*:

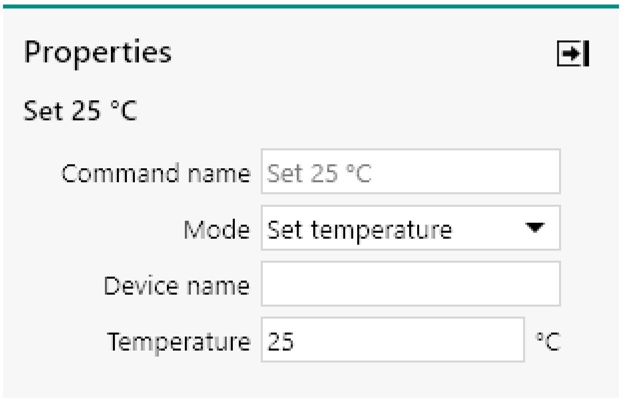


Figure 541 Set temperature mode properties

- **Command name:** a user-defined name for the command.
- **Device name:** the identifying device name of the Autolab RHD Microcell HC controller.
- **Temperature:** the target temperature to set on the Autolab RHD Microcell HC controller.

NOTE

When the **RHD control** command is used in the *Set temperature* mode, the command will be executed and will hold until the temperature stabilization conditions, defined in the hardware setup of the Autolab RHD Microcell HC controller are reached *(see Chapter 5.3, page 123)*. This command cannot be skipped or stopped.

8 Default procedures

NOVA is provided with a number of factory default procedures. These procedures can be accessed through the **Library** and can be used to perform simple measurements or as templates to for user-defined procedures.

The current version of NOVA provides the following default procedures, grouped per technique as explained in *Chapter 6.1*:

- Cyclic voltammetry
  - Cyclic voltammetry potentiostatic
  - Cyclic voltammetry galvanostatic
  - Cyclic voltammetry current integration
  - Cyclic voltammetry linear scan
  - Cyclic voltammetry linear scan high speed
- Linear sweep voltammetry
  - Linear sweep voltammetry potentiostatic
  - Linear sweep voltammetry galvanostatic
  - Linear polarization
  - Hydrodynamic linear sweep
  - Hydrodynamic linear sweep with RRDE
  - Spectroelectrochemical linear sweep voltammetry
- Voltammetric analysis
  - Sampled DC polarography
  - Normal pulse voltammetry
  - Differential pulse voltammetry
  - Differential normal pulse voltammetry
  - Square wave voltammetry
  - AC voltammetry
- Chrono methods
  - Chrono amperometry ($\Delta t > 1$ ms)
  - Chrono coulometry ($\Delta t > 1$ ms)
  - Chrono potentiometry ($\Delta t > 1$ ms)
  - Chrono amperometry fast
  - Chrono coulometry fast
  - Chrono potentiometry fast
  - Chrono amperometry high speed
  - Chrono potentiometry high speed
  - Chrono charge discharge
- Potentiometric stripping analysis
  - Potentiometric stripping analysis
  - Potentiometric stripping analysis (constant current)

- Impedance spectroscopy
  - FRA impedance potentiostatic
  - FRA impedance galvanostatic
  - FRA potential scan
  - FRA current scan
  - FRA time scan potentiostatic
  - FRA time scan galvanostatic

  Electrochemical Frequency Modulation

The rest of this chapter provides a detailed description of each procedure provided as a default in NOVA.

8.1 Cyclic voltammetry

NOVA provides five default procedures for cyclic voltammetry. These procedures can be used to perform a cyclic potential or current scan and record the response of the cell. Some of these procedures require optional hardware extensions.

The following procedures are available:

- Cyclic voltammetry potentiostatic
- Cyclic voltammetry galvanostatic
- Cyclic voltammetry current integration (requires the **FI20** or **on-board integrator**, please refer to *Chapter 16.3.2.11* for more information)
- Cyclic voltammetry linear scan (requires the **SCAN250** or **SCANGEN** module, please refer to *Chapter 16.3.2.19* for more information)
- Cyclic voltammetry linear scan high speed (requires the **SCAN250** or **SCANGEN** module and **ADC10M** or **ADC750** module, please refer to *Chapter 16.3.2.19* and *Chapter 16.3.2.1* for more information).

8.1.1 Cyclic voltammetry potentiostatic

The default **Cyclic voltammetry potentiostatic** procedure provides an example of a typical *staircase* cyclic voltammetry procedure in potentiostatic mode *(see Figure 542, page 471)*.

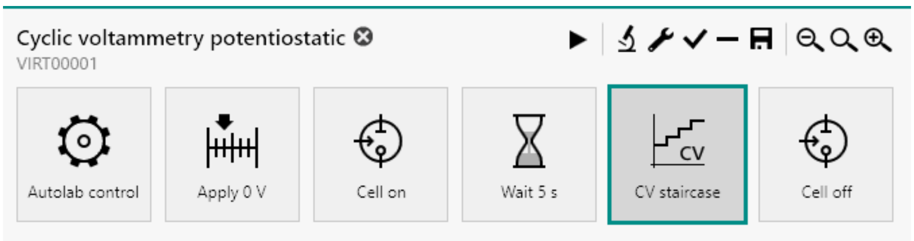


Figure 542 The default Cyclic voltammetry potentiostatic procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure has the following measurement properties, specified for the **CV staircase** command *(see Figure 543, page 472)*:

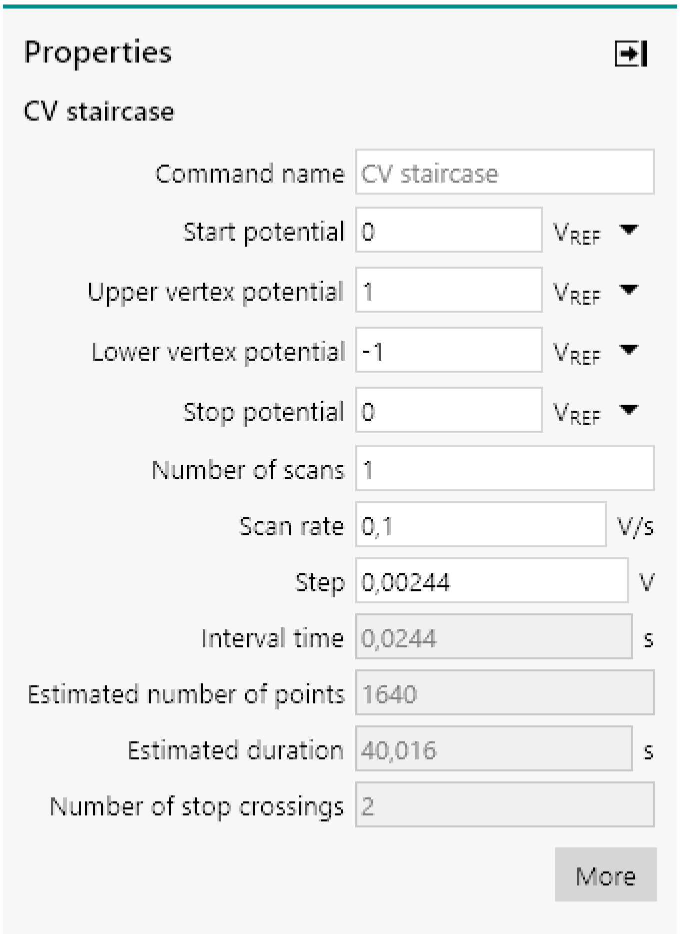


Figure 543 The measurement properties of the CV staircase command

- **CV staircase**
  - Start potential: 0 V, versus reference electrode
  - Upper vertex potential: 1 V, versus reference electrode
  - Lower vertex potential: -1 V, versus reference electrode
  - Stop: 0 V, versus reference electrode
  - Number of scans: 1
  - Step potential: 0.00244 V
  - Scan rate: 100 mV/s

The procedure samples the following signals *(see Figure 544, page 473)*:

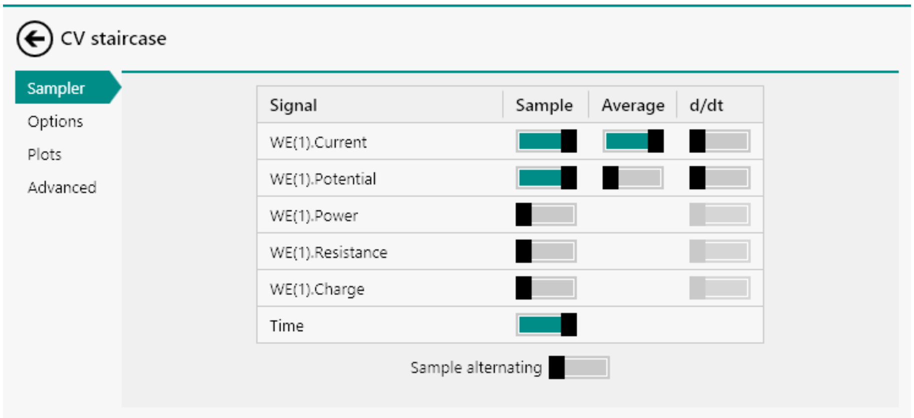


Figure 544 The sampler of the CV staircase command

- WE(1).Current (averaged)
- WE(1).Potential
- Time

The procedure uses the following options *(see Figure 545, page 473)*:

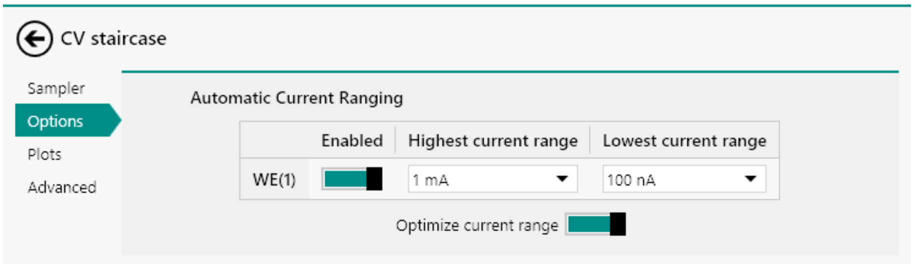


Figure 545 The options of the CV staircase command

- Automatic current ranging
  - Highest current range: 1 mA
  - Lowest current range: 100 nA

The procedure plots the following data *(see Figure 546, page 474)*:

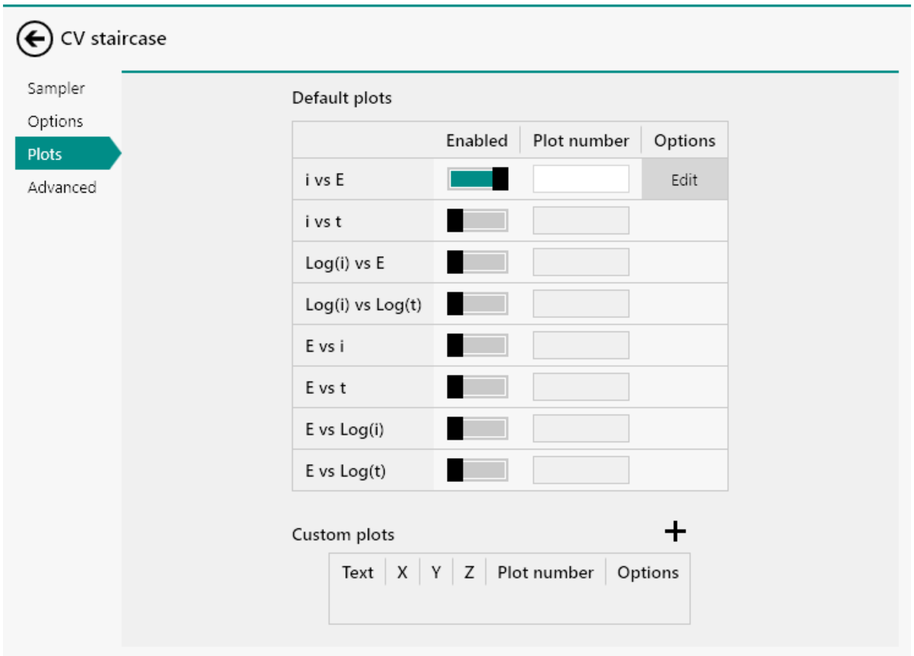


Figure 546 The plots of the CV staircase command

- i vs E: WE(1).Current versus Potential applied

The procedure also has the value of alpha property available in the Advanced section. This value is set to the default value of 1 *(see Figure 547, page 474)*.

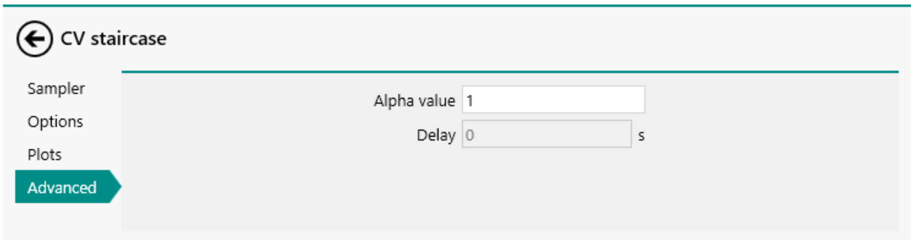


Figure 547 The advanced settings of the CV staircase command

8.1.2 Cyclic voltammetry galvanostatic

The default **Cyclic voltammetry galvanostatic** procedure provides an example of a typical *staircase* cyclic voltammetry procedure in galvanostatic mode *(see Figure 548, page 474)*.

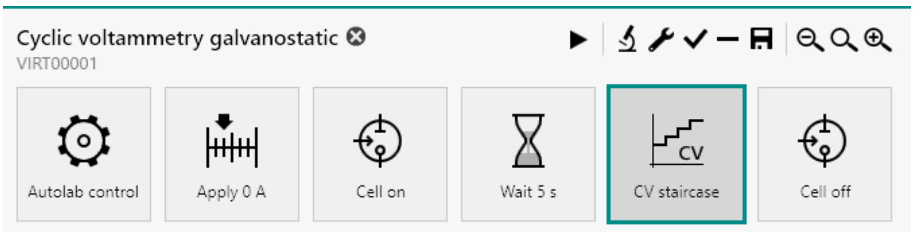


Figure 548 The default Cyclic voltammetry galvanostatic procedure

> **NOTE**
>
> The galvanostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure has the following measurement properties, specified for the **CV staircase** command *(see Figure 549, page 475)*:

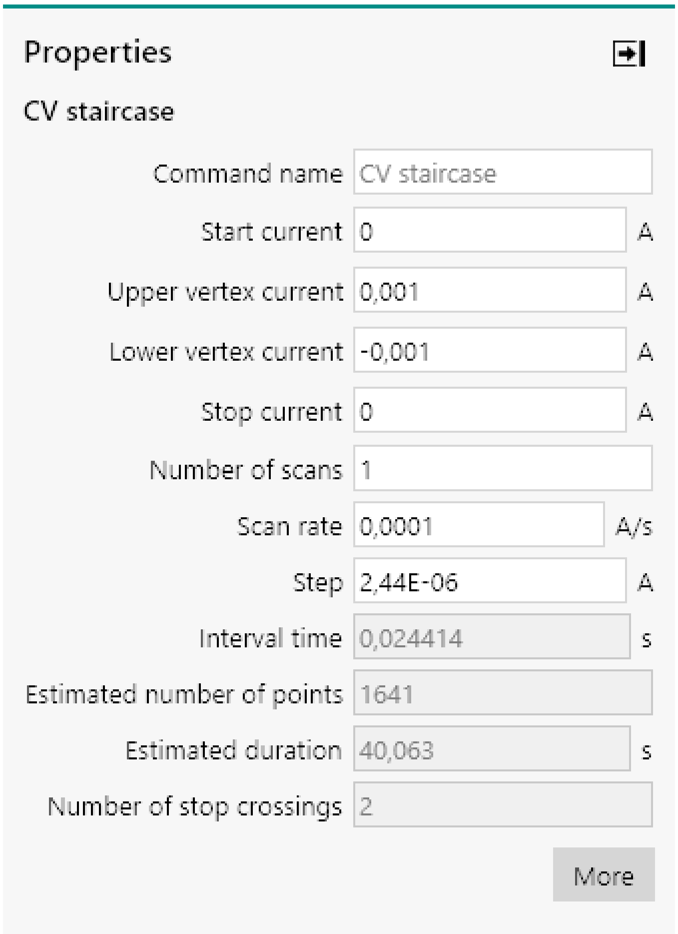


Figure 549 The measurement properties of the CV staircase command

- **CV staircase**
  - Start current: 0 A
  - Upper vertex current: 0.001 A
  - Lower vertex current: -0.001 A
  - Stop current: 0 A
  - Number of scans: 1
  - Step: 2.44 µA
  - Scan rate: 0.0001 A/s

The procedure samples the following signals *(see Figure 550, page 476)*:

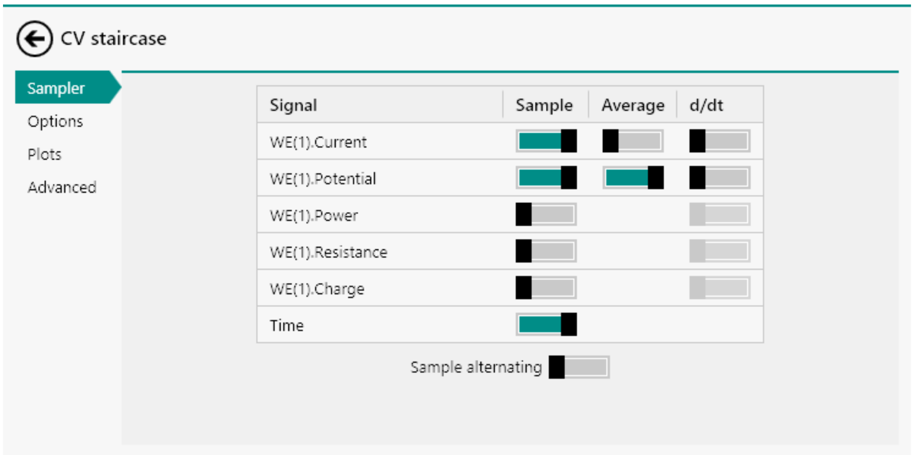


Figure 550 The sampler of the CV staircase command

- WE(1).Current
- WE(1).Potential (averaged)
- Time

The procedure plots the following data *(see Figure 551, page 476)*:

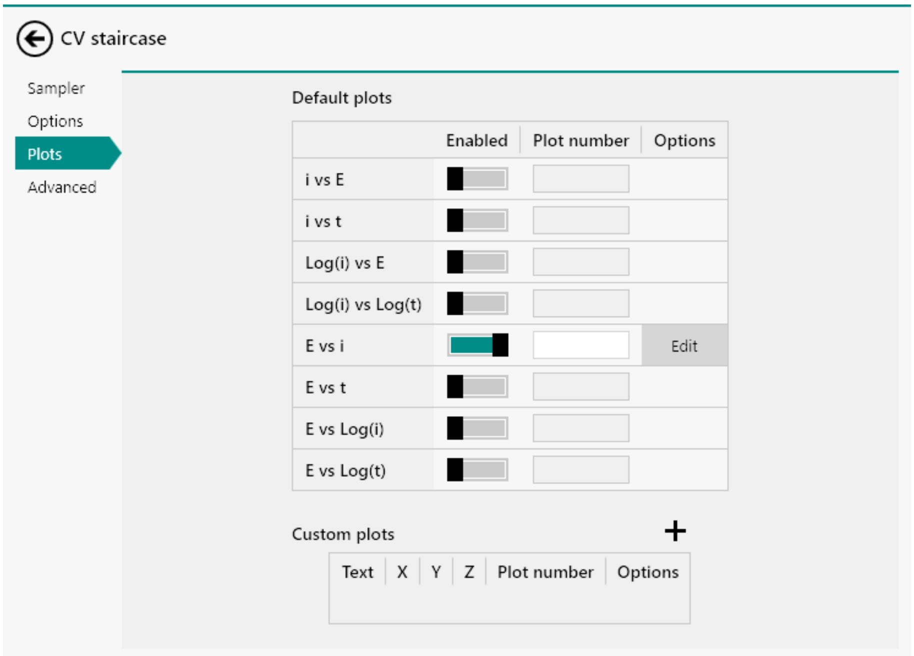


Figure 551 The plots of the CV staircase command

- E vs i: WE(1).Potential versus Current applied

The procedure also has the value of alpha property available in the Advanced section. This value is set to the default value of 1 *(see Figure 552, page 477)*.

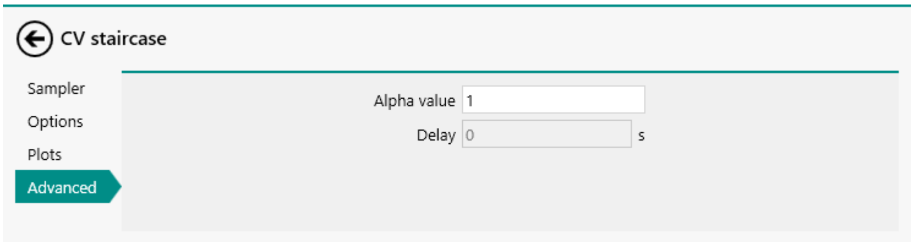


Figure 552 The advanced settings of the CV staircase command

8.1.3 Cyclic voltammetry potentiostatic current integration

CAUTION

This procedure requires the optional **FI20** module or the **on-board integrator** *(see Chapter 16.3.2.11, page 1061)*.

The default **Cyclic voltammetry potentiostatic** current integration procedure provides an example of a typical *staircase* cyclic voltammetry procedure in potentiostatic mode, using the optional **FI20** module or **on-board integrator** *(see Figure 553, page 477)*.

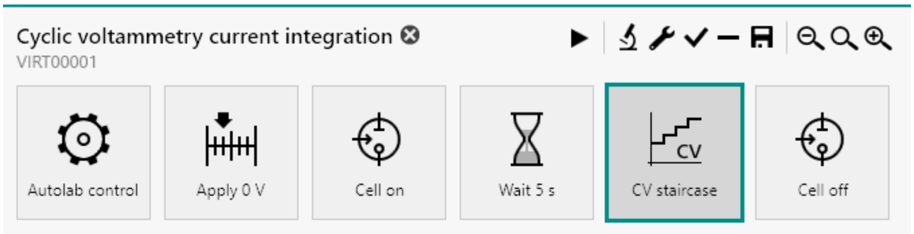


Figure 553 The default Cyclic voltammetry potentiostatic current integration procedure

The charge determined during each step is used to recalculate the total current.

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

NOTE

It is highly recommended to determine and reset the integrator drift before using this procedure. The drift can be determined using the dedicated tool *(see Chapter 5.2.2.6, page 115)*.

The procedure has the following measurement properties, specified for the **CV staircase** command *(see Figure 554, page 478)*:

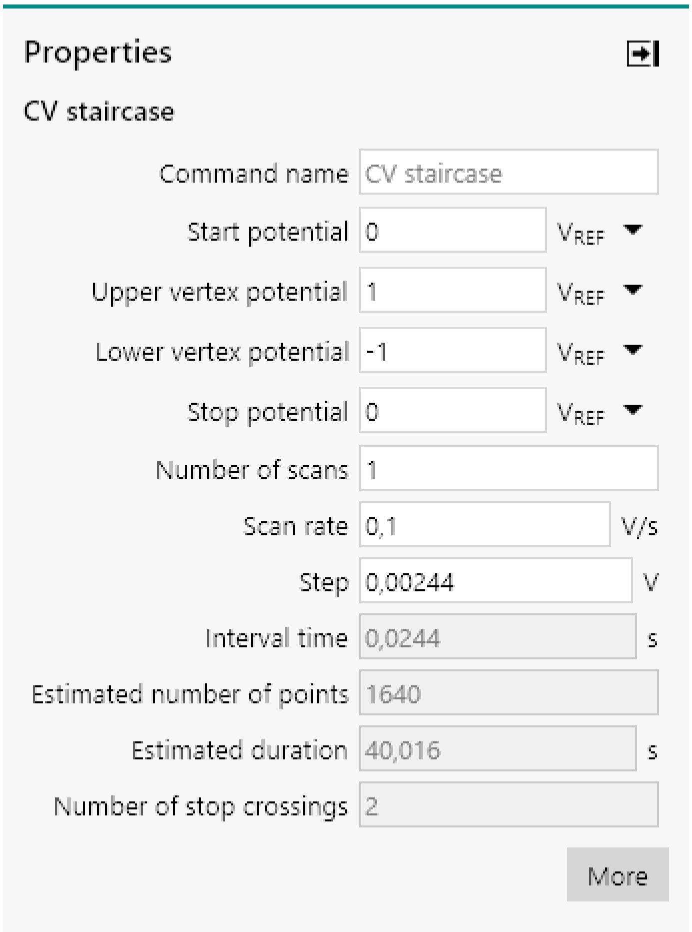


Figure 554 The properties of the CV staircase command

- **CV staircase**
  - Start potential: 0 V, versus reference electrode
  - Upper vertex potential: 1 V, versus reference electrode
  - Lower vertex potential: -1 V, versus reference electrode
  - Stop potential: 0 V, versus reference electrode
  - Number of scans: 1
  - Step: 0.00244 V
  - Scan rate: 100 mV/s

The procedure samples the following signals *(see Figure 555, page 479)*:

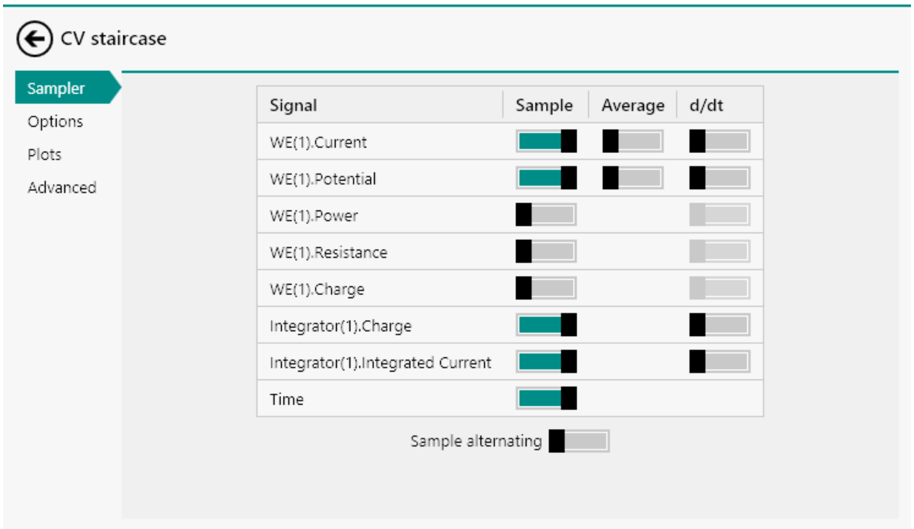


Figure 555 The sampler of the CV staircase command

- WE(1).Current
- WE(1).Potential
- Integrator(1).Integrated Current
- Time

The procedure plots the following data *(see Figure 556, page 479)*:

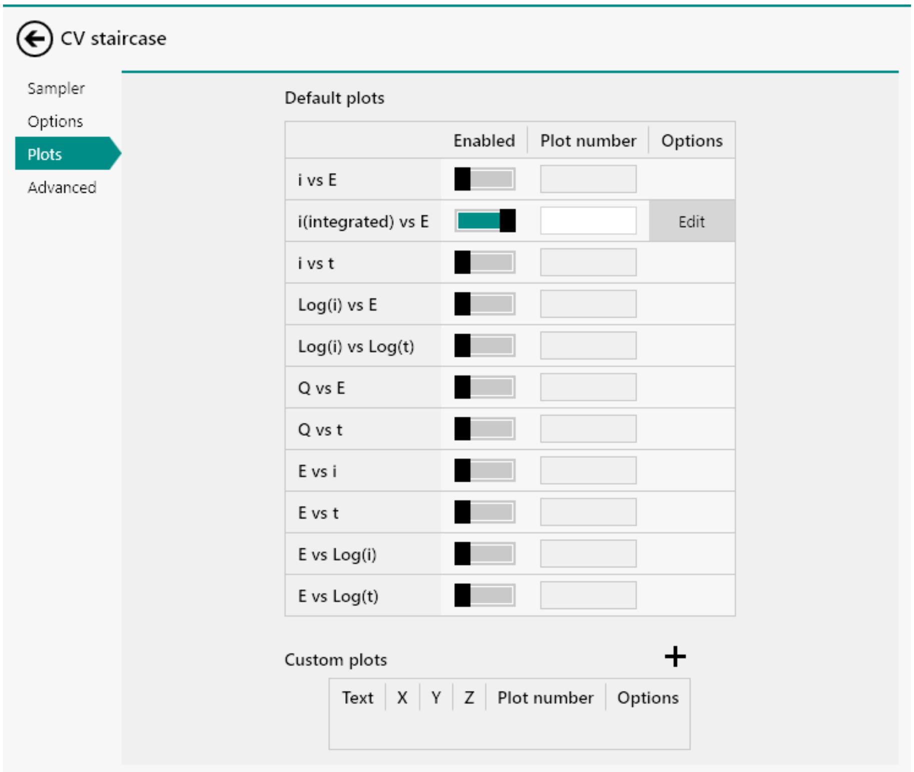


Figure 556 The plots of the CV staircase command

- i vs E: Integrator(1).Integrated Current versus Potential applied

The procedure also has the value of alpha property available in the Advanced section. This value is set to the default value of 1 *(see Figure 557, page 480)*.

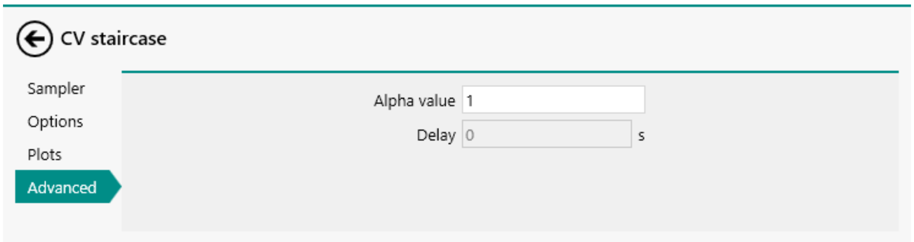


Figure 557 The advanced settings of the CV staircase command

8.1.4 Cyclic voltammetry potentiostatic linear scan

CAUTION

This procedure requires the optional **SCAN250** or **SCANGEN** module *(see Chapter 16.3.2.19, page 1148)*.

The default **Cyclic voltammetry potentiostatic linear scan** procedure provides an example of a typical *linear scan* cyclic voltammetry procedure in potentiostatic mode, using the optional **SCAN250** or **SCANGEN** module *(see Figure 558, page 480)*.

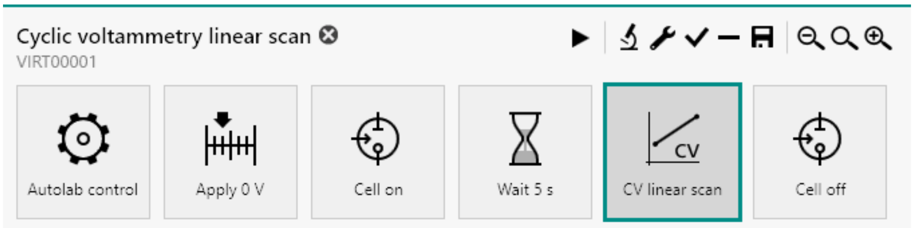


Figure 558 The default Cyclic voltammetry potentiostatic linear scan procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure has the following measurement parameters, specified for the **CV linear scan** command *(see Figure 559, page 481)*:

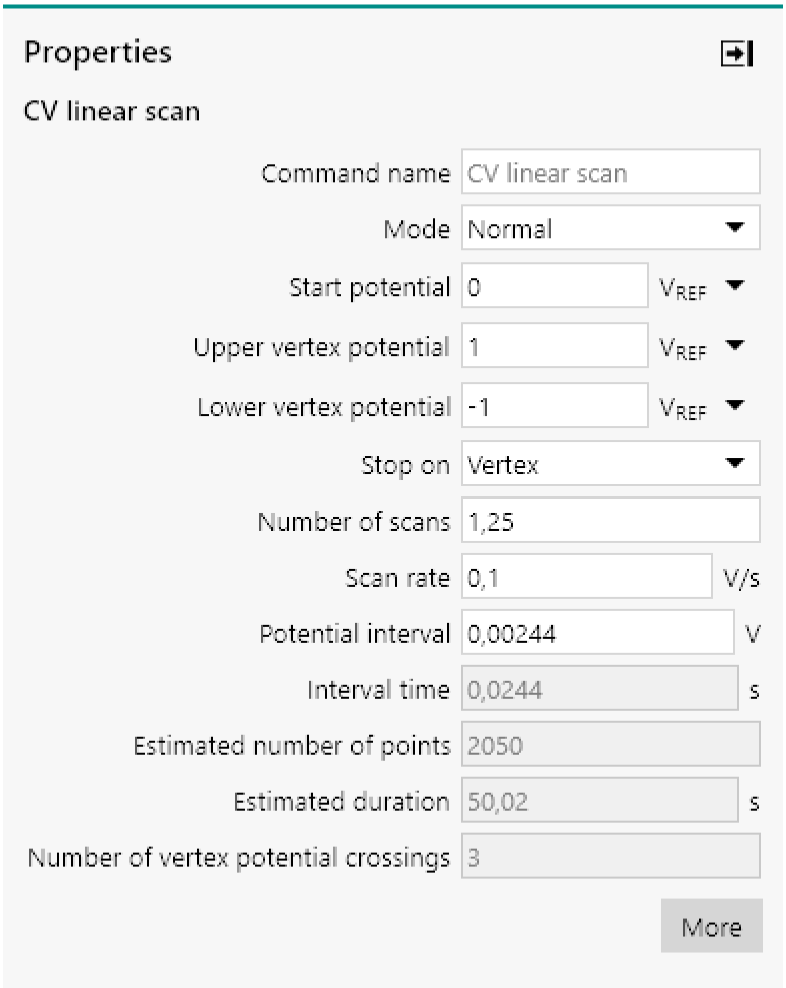


Figure 559 The properties of the CV linear scan command

- **CV linear scan**
  - Start potential: 0 V, versus reference electrode
  - Upper vertex potential: 1 V, versus reference electrode
  - Lower vertex potential: -1 V, versus reference electrode
  - Number of scans: 1,25
  - Potential interval: 0.00244 V
  - Scan rate: 100 mV/s

The procedure samples the following signals *(see Figure 560, page 482)*:

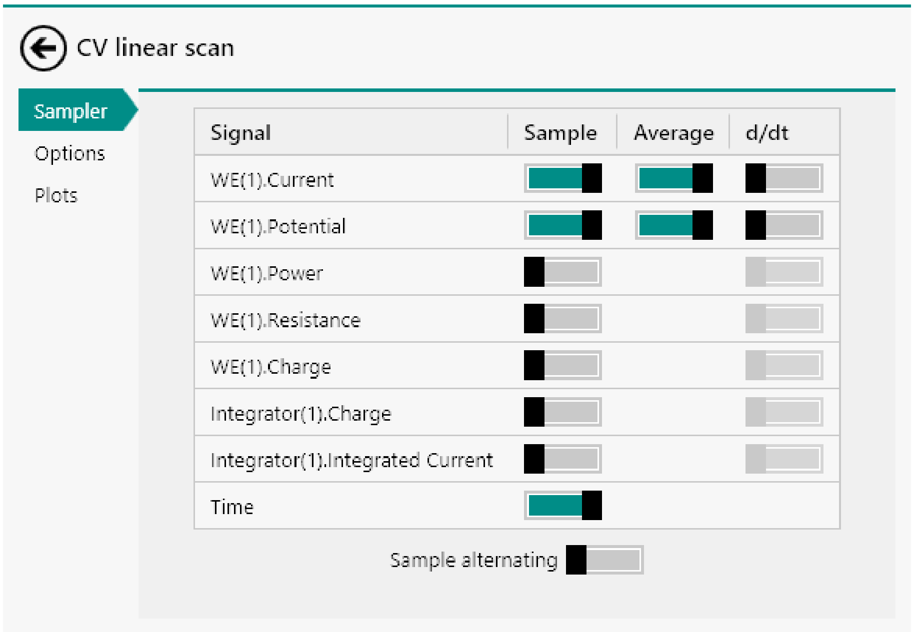


Figure 560 The sampler of the CV linear scan command

- WE(1).Current (averaged)
- WE(1).Potential
- Time

The procedure uses the following options *(see Figure 561, page 482)*:

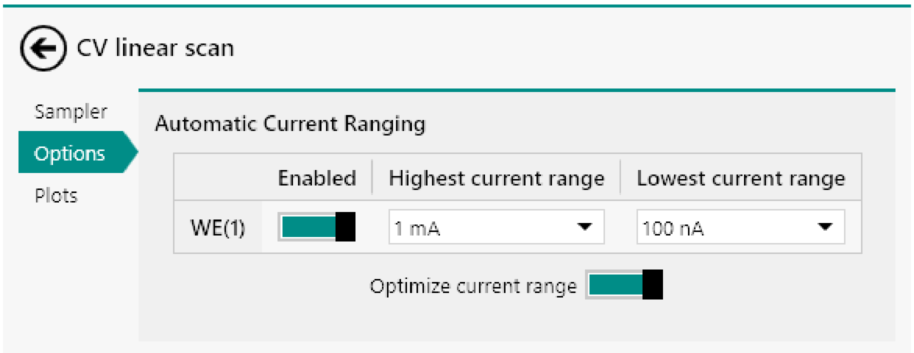


Figure 561 The options of the CV linear scan command

- Automatic current ranging
  - Highest current range: 1 mA
  - Lowest current range: 100 nA

The procedure plots the following data *(see Figure 562, page 483)*:

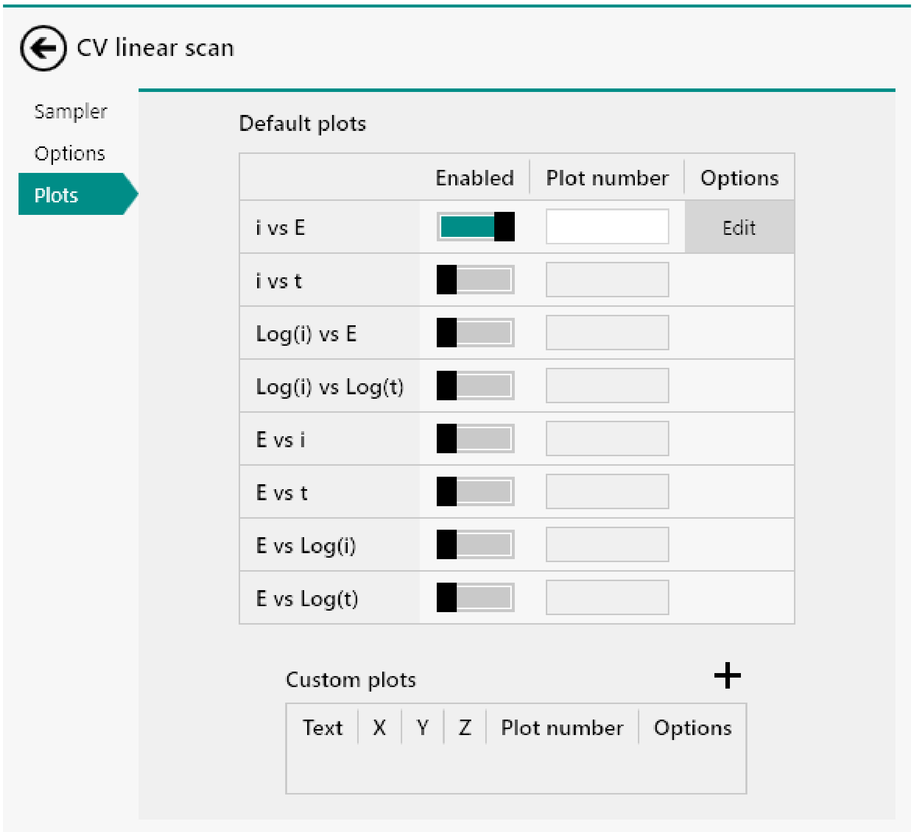


Figure 562 The plots of the CV linear scan command

- i vs E: WE(1).Current versus Potential applied

8.1.5 Cyclic voltammetry potentiostatic linear scan high speed

CAUTION

This procedure requires the optional **SCAN250** or **SCANGEN** module in combination with the optional **ADC10M** or **ADC750** module*(see Chapter 16.3.2.19, page 1148)* and *(see Chapter 16.3.2.1, page 977)*.

The default **Cyclic voltammetry potentiostatic linear scan high speed** procedure provides an example of a typical **linear scan** cyclic voltammetry procedure at very high scan rate in potentiostatic mode, using the optional **SCAN250** or **SCANGEN** module in combination with the optional **ADC10M** or **ADC750** module *(see Figure 563, page 484)*.

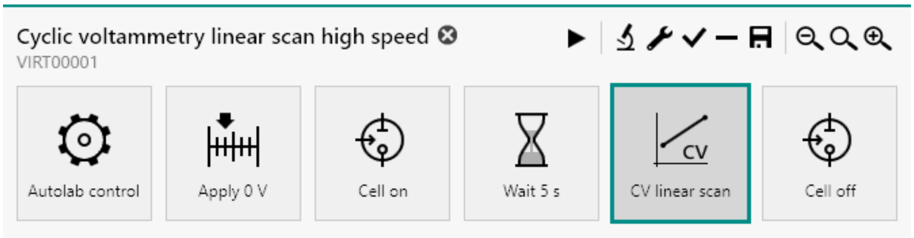


Figure 563 The default Cyclic voltammetry potentiostatic linear scan high speed procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure has the following measurement properties, specified for the **CV linear scan** command *(see Figure 564, page 484)*:

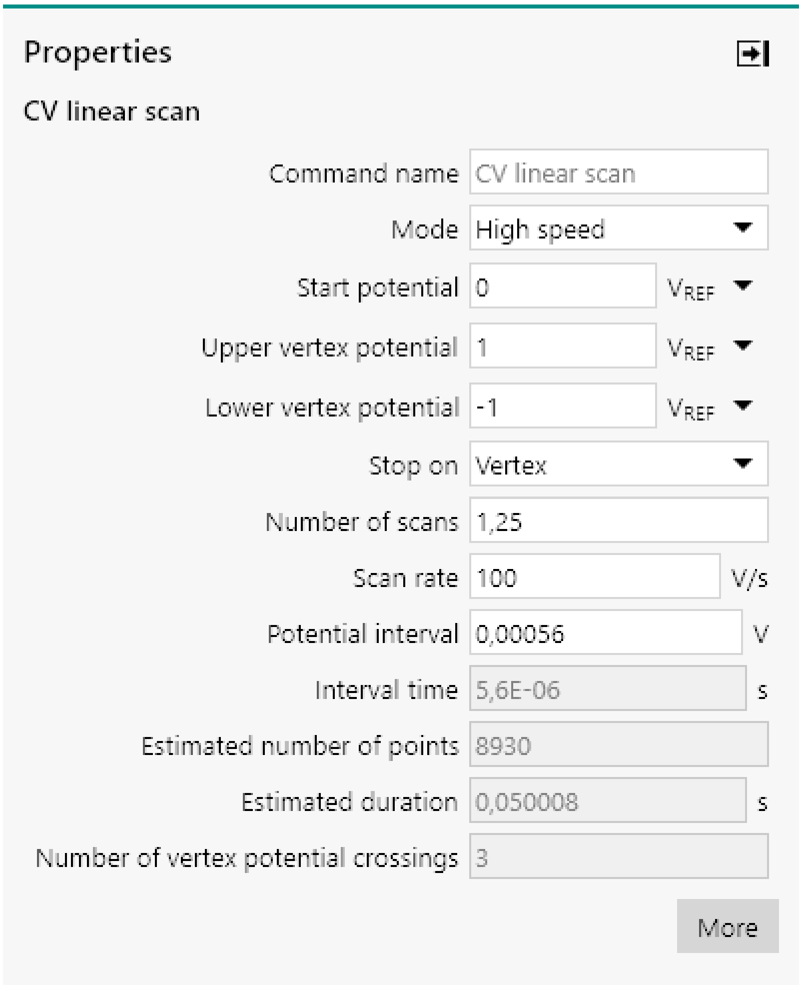


Figure 564 The properties of the CV linear scan command

- **CV linear scan**
  - Start potential: 0 V, versus reference electrode
  - Upper vertex potential: 1 V, versus reference electrode
  - Lower vertex potential: -1 V, versus reference electrode
  - Number of scans: 1,25
  - Potential interval: 0.00056 V
  - Scan rate: 100 V/s

The procedure samples the following **ADC10M** or **ADC750** settings *(see Figure 565, page 485)*:

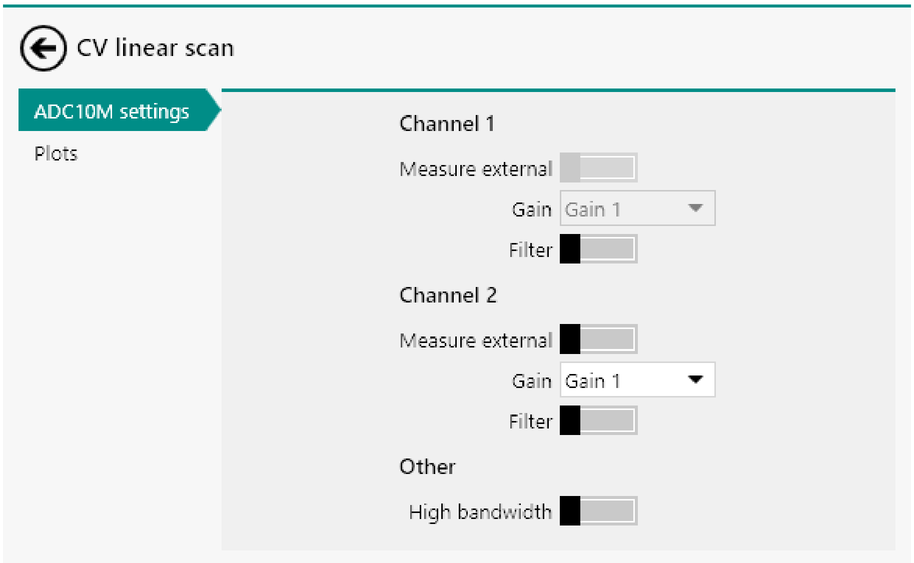


Figure 565 The ADC10M or ADC750 settings of the CV linear scan command

- Channel 1: WE(1).Potential, Gain 1, unfiltered
- Channel 2: WE(1).Current, Gain 1, unfiltered

The procedure plots the following data *(see Figure 566, page 486)*:

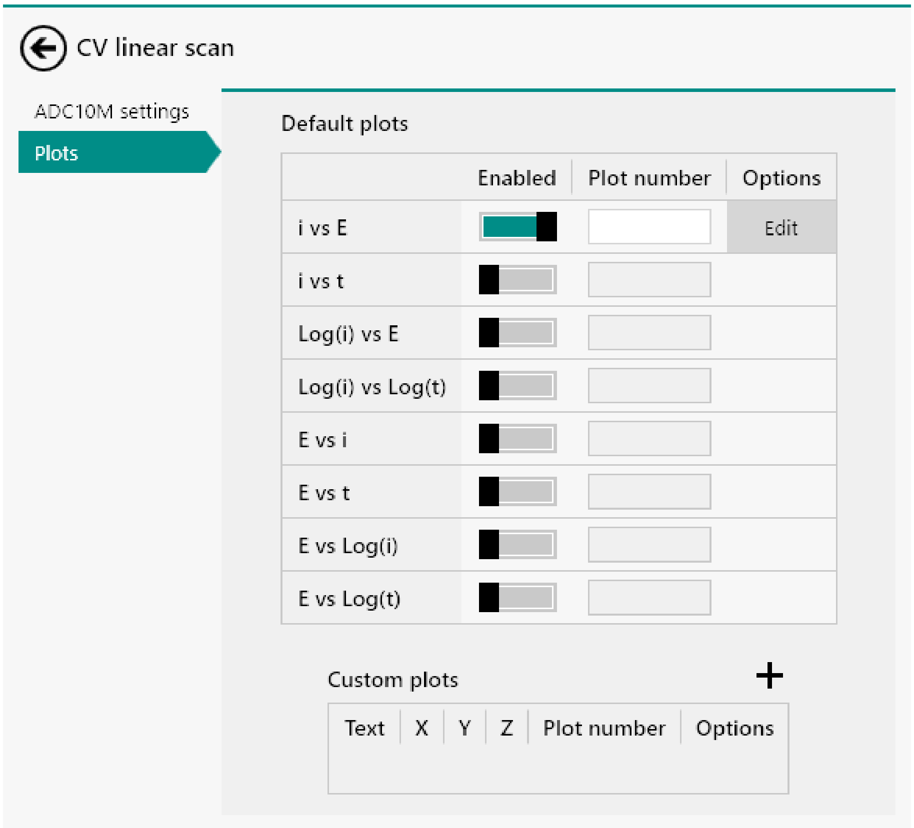


Figure 566 The plots of the CV linear scan command

- i vs E: WE(1).Current versus Potential applied

NOTE

The measured data cannot be displayed in real-time. The data is only available at the end of the measurement.

8.2 Linear sweep voltammetry

NOVA provides four default procedures for linear sweep voltammetry. These procedures can be used to perform a potential or current sweep and record the response of the cell. Some of these procedure require optional hardware extensions.

The following procedures are available:

- Linear sweep voltammetry potentiostatic
- Linear sweep voltammetry galvanostatic
- Linear polarization

- Hydrodynamic linear sweep (requires the Autolab rotating disk electrode (RDE) or Autolab rotating ring disk electrode (RRDE), please refer to the Autolab RDE/RRDE User Manual for more information)
- Hydrodynamic linear sweep with RRDE (requires the Autolab rotating ring disk electrode (RRDE), please refer to the Autolab RDE/RRDE User Manual for more information)
- Spectroelectrochemical linear sweep voltammetry (requires an Autolab or Avantes spectrophotometer)

8.2.1 Linear sweep voltammetry potentiostatic

The default **Linear sweep voltammetry potentiostatic** procedure provides an example of a typical *staircase* Linear sweep voltammetry procedure in potentiostatic mode *(see Figure 567, page 487)*.

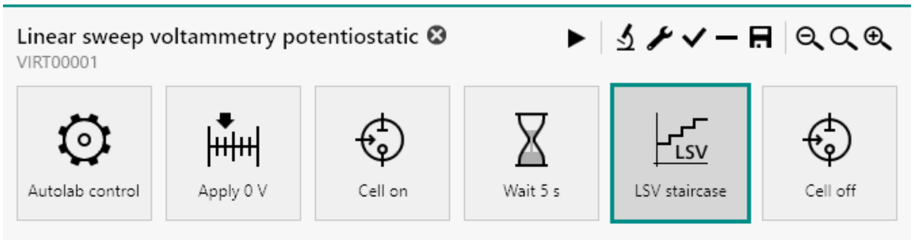


Figure 567 The default Linear sweep voltammetry potentiostatic procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure has the following measurement properties, specified for the **LSV staircase** command *(see Figure 568, page 488)*:

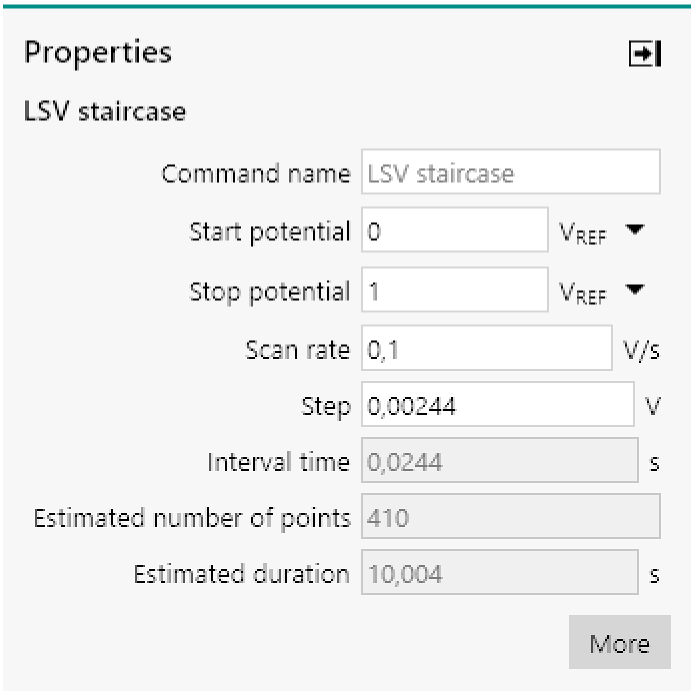


Figure 568 The properties of the LSV staircase command

- **LSV staircase**
  - Start potential: 0 V, versus reference electrode
  - Stop potential: 0 V, versus reference electrode
  - Step: 0.00244 V
  - Scan rate: 100 mV/s

The procedure samples the following signals *(see Figure 569, page 488)*:

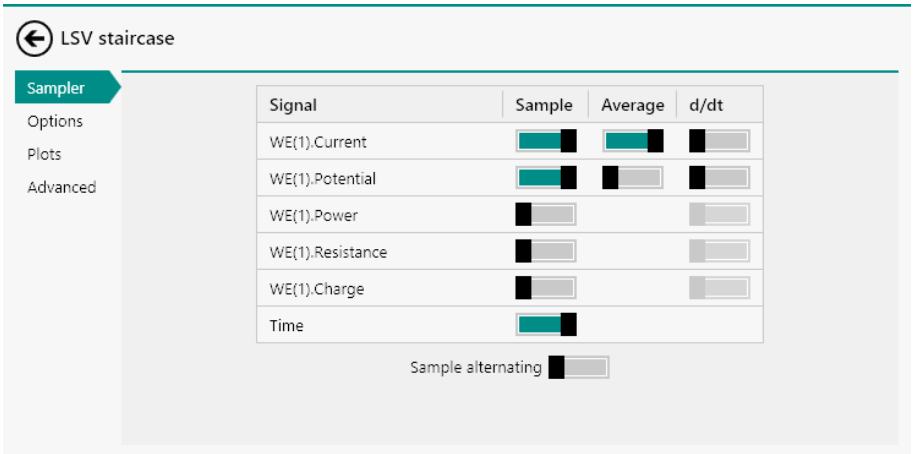


Figure 569 The sampler of the LSV staircase command

- WE(1).Current (averaged)
- WE(1).Potential
- Time

The procedure uses the following options *(see Figure 570, page 489)*:

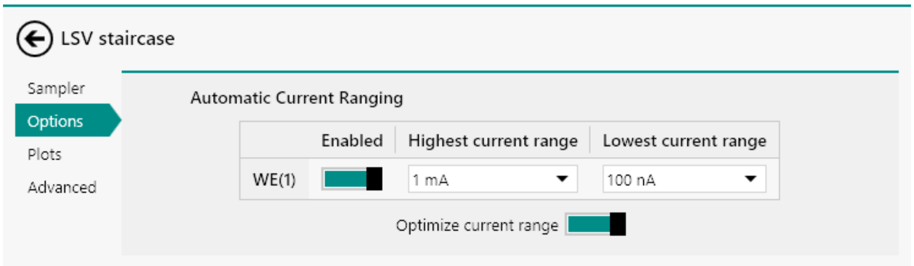


Figure 570 The options of the LSV staircase command

- Automatic current ranging
  - Highest current range: 1 mA
  - Lowest current range: 100 nA

The procedure plots the following data *(see Figure 571, page 489)*:

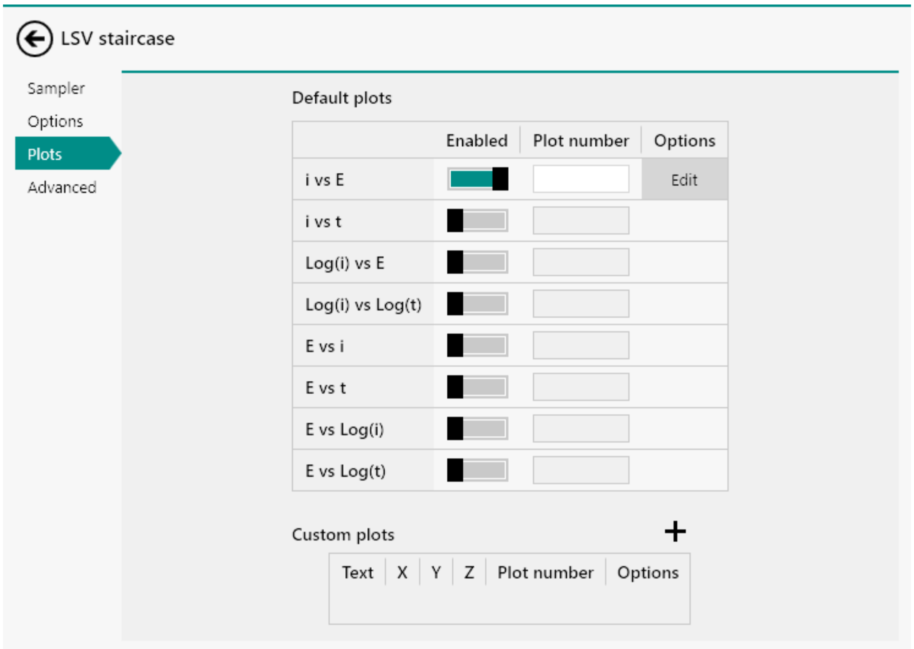


Figure 571 The plots of the LSV staircase command

- i vs E: WE(1).Current versus Potential applied

The procedure also has the value of alpha property available in the Advanced section. This value is set to the default value of 1 *(see Figure 572, page 489)*.

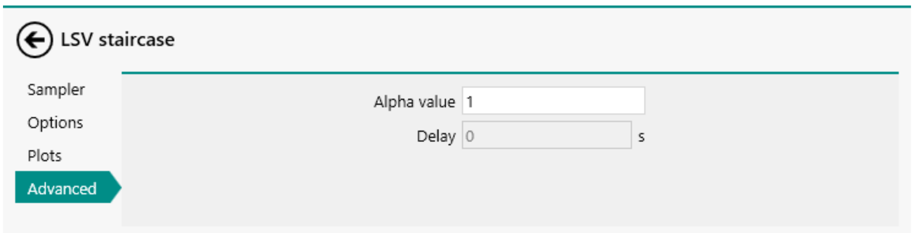


Figure 572 The advanced settings of the LSV staircase command

8.2.2 Linear sweep voltammetry galvanostatic

The default **Linear sweep voltammetry galvanostatic** procedure provides an example of a typical *staircase* Linear sweep voltammetry procedure in galvanostatic mode *(see Figure 573, page 490)*.

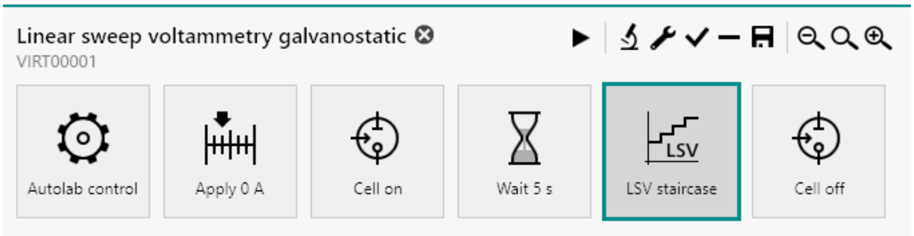


Figure 573 The default Linear sweep voltammetry galvanostatic procedure

NOTE

The galvanostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure has the following measurement properties, specified for the **LSV staircase** command *(see Figure 574, page 490)*:

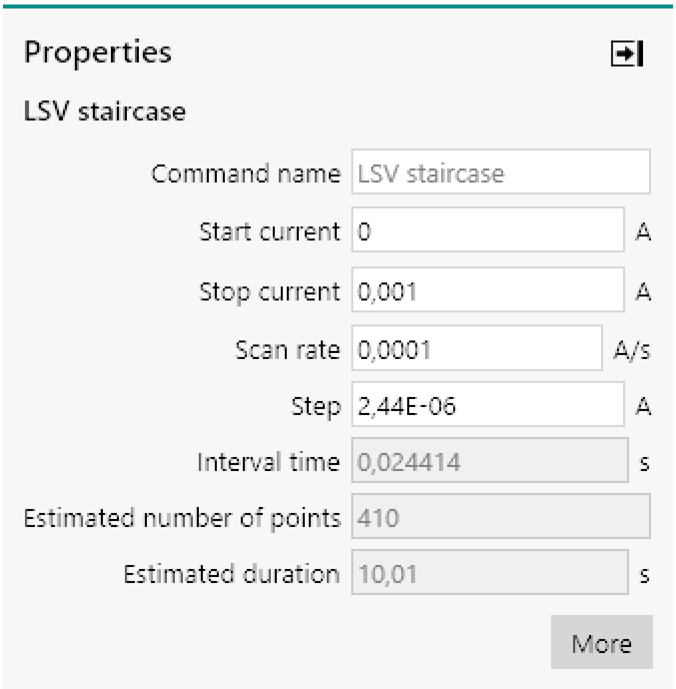


Figure 574 The measurement properties of the LSV staircase command

- **LSV staircase**
  - Start current: 0 A
  - Stop current: 0.001 A
  - Step: 2.44 μA
  - Scan rate: 0.0001 A/s

The procedure samples the following signals *(see Figure 575, page 491)*:

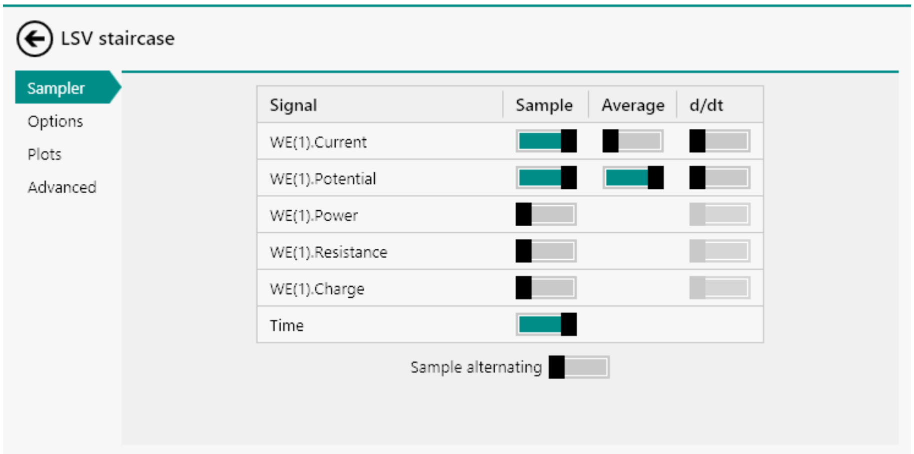


Figure 575 The measurement properties of the LSV staircase command

- WE(1).Current
- WE(1).Potential (averaged)
- Time

The procedure plots the following data *(see Figure 576, page 492)*:

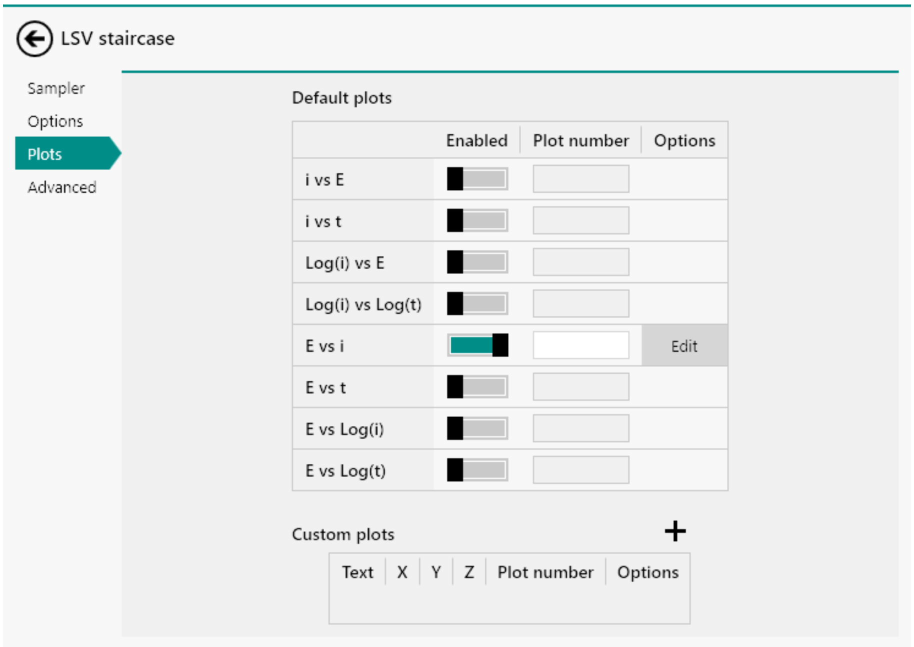


Figure 576 The measurement properties of the LSV staircase command

- E vs i: WE(1).Potential versus Current applied

The procedure also has the value of alpha property available in the Advanced section. This value is set to the default value of 1 *(see Figure 577, page 492)*.

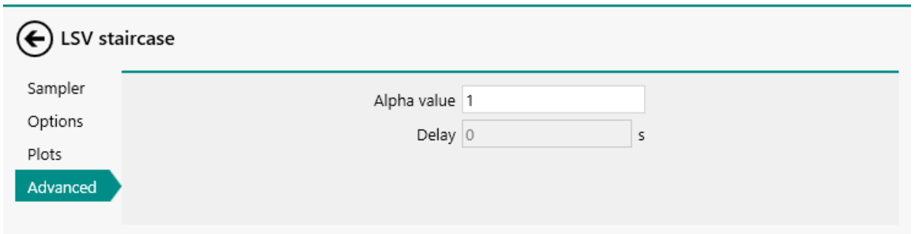


Figure 577 The advanced settings of the LSV staircase command

8.2.3 Linear polarization

The default **Linear polarization** procedure provides an example of a typical *staircase* corrosion measurement according to *ASTM G5-14* in potentiostatic mode *(see Figure 578, page 493)*.

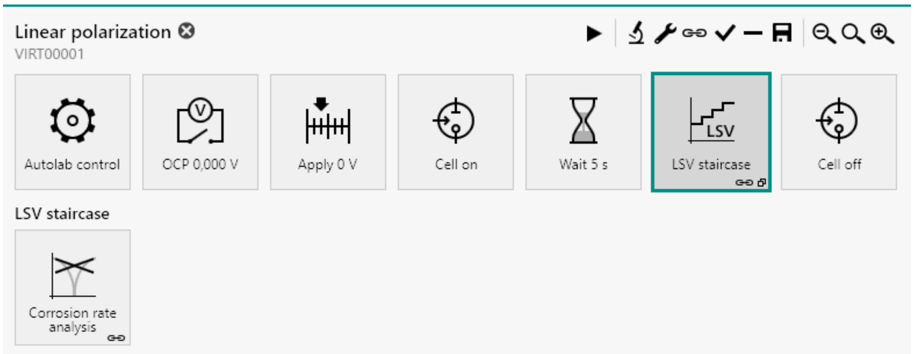


Figure 578 The default Linear polarization procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure has the following measurement properties, specified for the **LSV staircase** command *(see Figure 579, page 493)*:

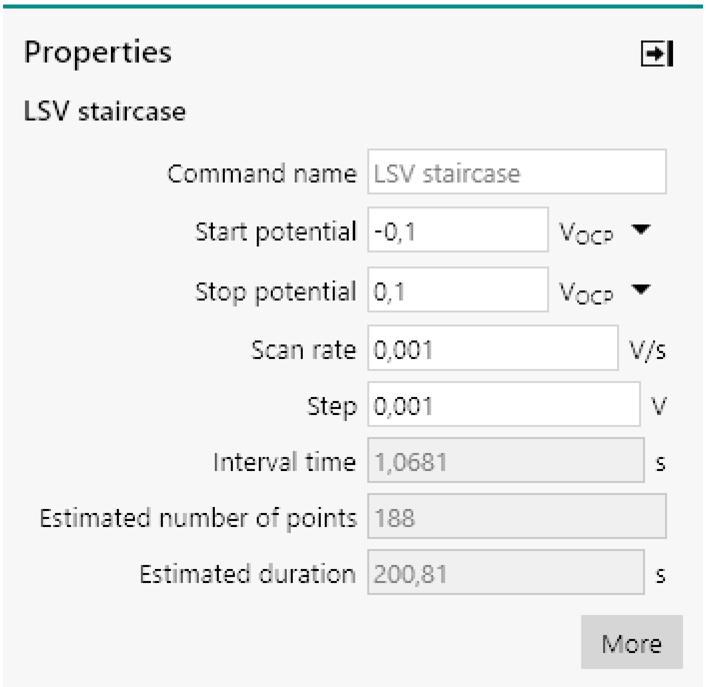


Figure 579 The measurement properties of the LSV staircase command

- **LSV staircase**
  - Start potential: -0.1 V, versus open circuit potential
  - Stop potential: 0.1 V, versus open circuit potential
  - Step: 0.001 V
  - Scan rate: 1 mV/s

The procedure samples the following signals *(see Figure 580, page 494)*:

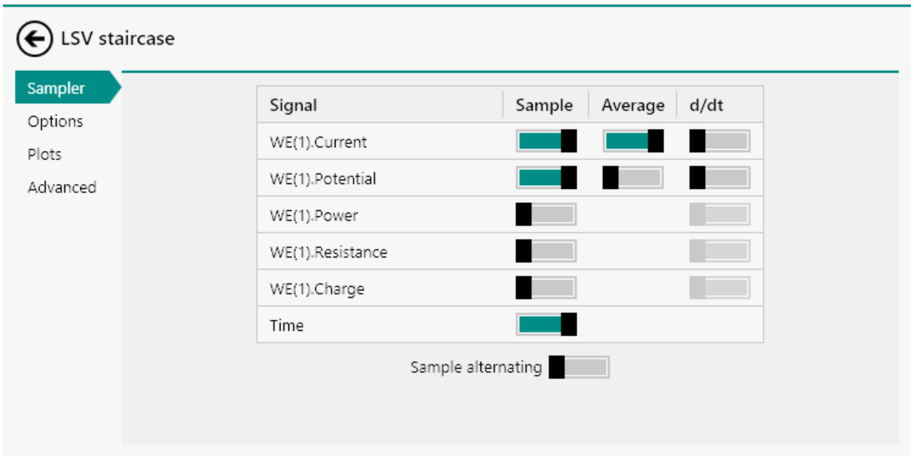


Figure 580 The sampler of the LSV staircase command

- WE(1).Current (averaged)
- WE(1).Potential
- Time

The procedure uses the following options *(see Figure 581, page 494)*:

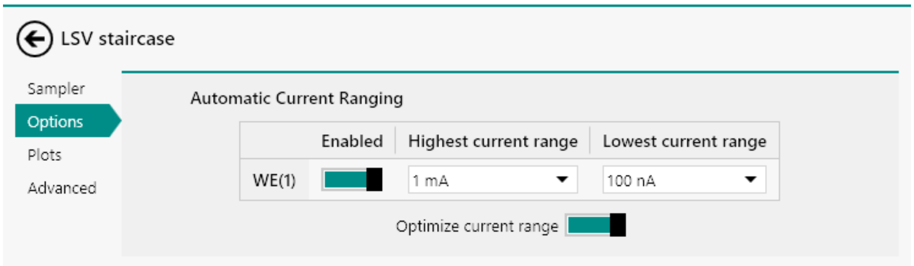


Figure 581 The options of the LSV staircase command

- Automatic current ranging
  - Highest current range: 1 mA
  - Lowest current range: 100 nA

The procedure plots the following data *(see Figure 582, page 495)*:

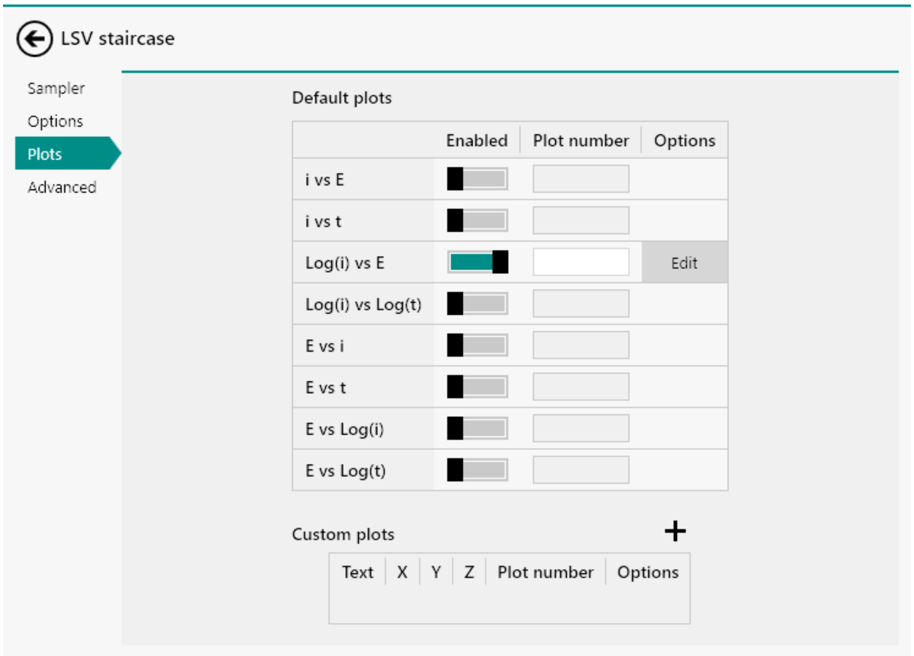


Figure 582 The plots of the LSV staircase command

- Log(i) vs E: Log(WE(1).Current) versus Potential applied

The procedure also has the value of alpha property available in the Advanced section. This value is set to the default value of 1 *(see Figure 583, page 495)*.

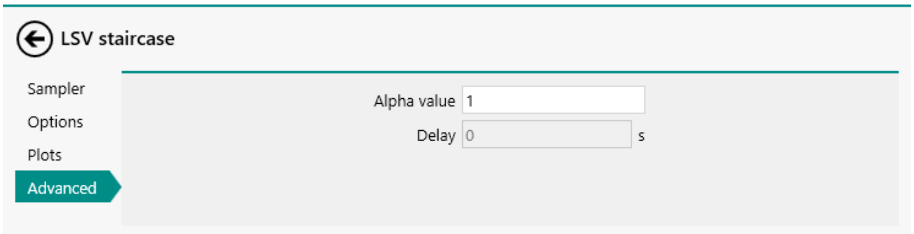


Figure 583 The advanced settings of the LSV staircase command

NOTE

The open circuit potential is measured by the **OCP determination** command located before the **LSV staircase** command. Please refer to *Chapter 7.2.5* for more information.

> **NOTE**
>
> The procedure includes a **Corrosion rate analysis** command to automatically analyze the measured data. Please refer to *Chapter 7.8.14* for more information.

8.2.4 Hydrodynamic linear sweep

> **CAUTION**
>
> This procedure requires the optional **Autolab rotating disk electrode (RDE)** or **Autolab rotating ring disk electrode (RRDE)** connected to the Autolab using the motor controller. The procedure is designed to remotely control the rotation rate. For more information, please refer to the Autolab RDE/RRDE User Manual.

The default **Hydrodynamic linear sweep** procedure provides an example of a typical *staircase* linear sweep voltammetry procedure in potentiostatic mode in combination with the **Autolab rotating disk electrode (RDE)** or **Autolab rotating ring disk electrode (RRDE)** *(see Figure 584, page 496)*.

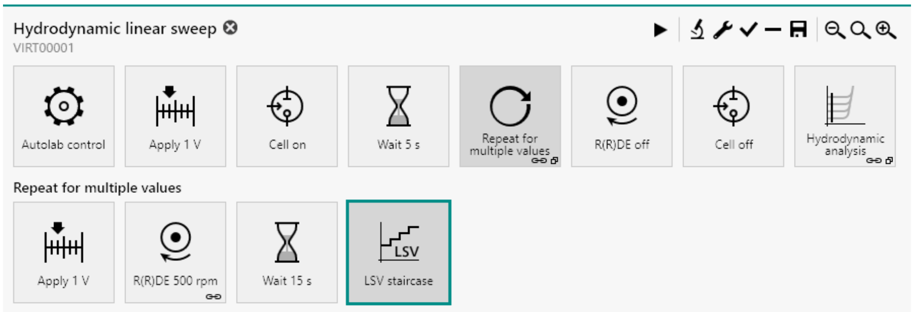


Figure 584 The default Hydrodynamic linear sweep procedure

> **NOTE**
>
> The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The Hydrodynamic linear sweep voltammetry procedure performs a linear sweep voltammetry using the Autolab RDE or Autolab RRDE, with six different rotation rates. The rotation rate of the Autolab RDE or Autolab

RRDE is set using the **R(R)DE** command linked to the values of a **Repeat** command.

The **Repeat** command is used in the *Repeat for multiple values* mode and is preconfigured to cycle through six rotation rates, starting at 500 RPM until 3000 RPM, using a square root distribution *(see Figure 585, page 497)*.

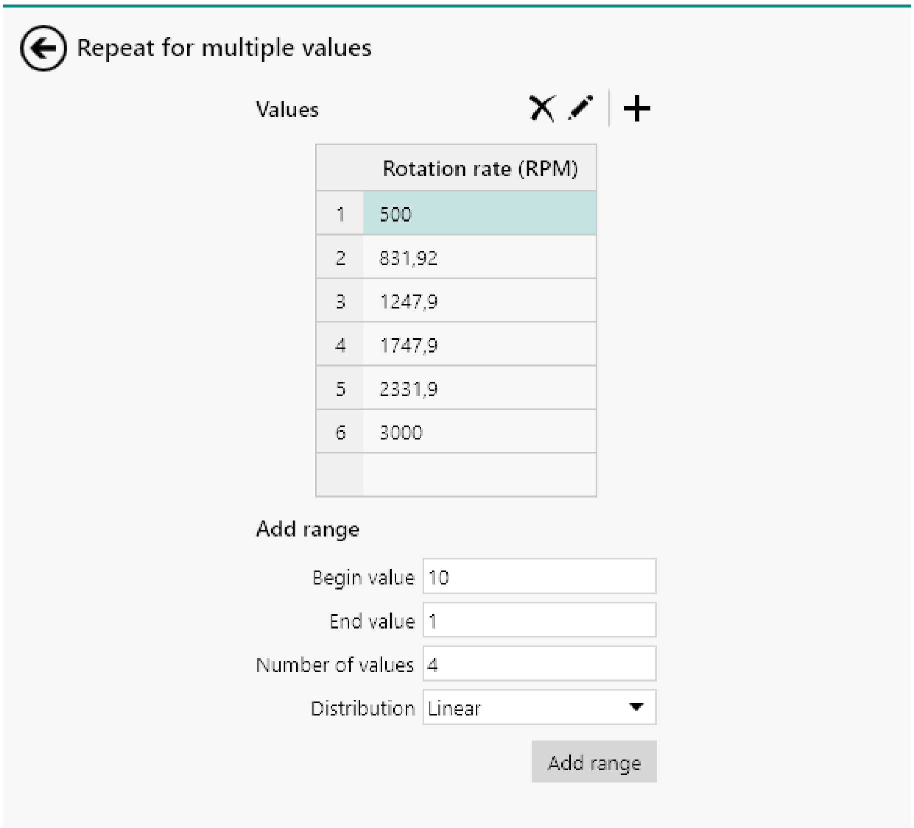


Figure 585 The repeat loop used in the default Hydrodynamic linear sweep procedure

The **Rotation rate** parameter, created by **Repeat** command, is linked to the **R(R)DE** command included in the repeat loop *(see Figure 586, page 498)*.

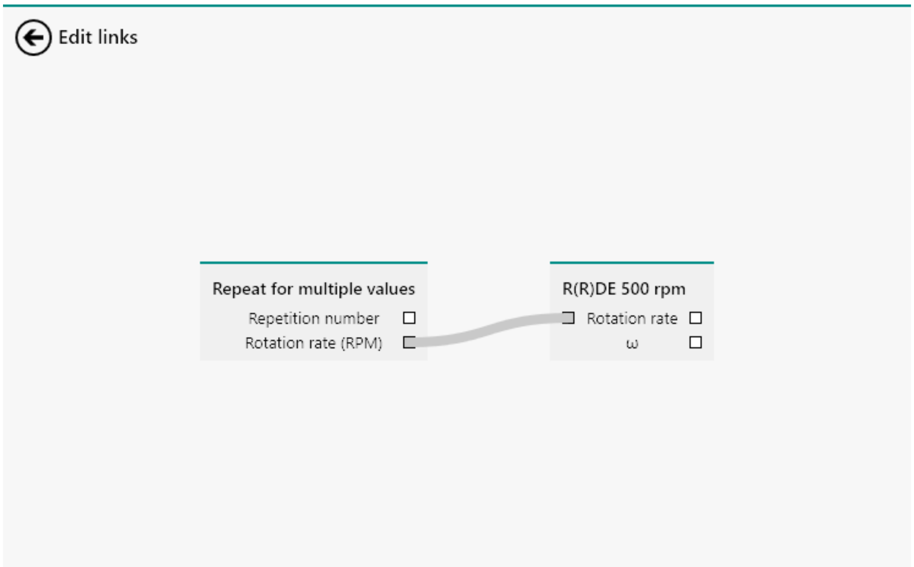


Figure 586 The link used to control the rotation rate of the R(R)DE

This procedure is intended to be used with the **Remote** switch of the Autolab motor controller engaged (on the back plane of the controller) and with a BNC cable connected between the DAC164 ←1 connector (Vout for the μAutolab type II, μAutolab type III, PGSTAT101, M101, PGSTAT204 and M204) and the Remote input plug on the back plane of the Autolab RDE motor controller *(see Figure 587, page 498)*.

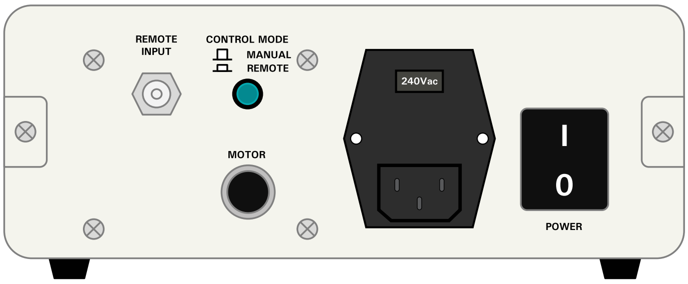


Figure 587 The back plane of the Autolab motor controller

The procedure has the following measurement properties, specified for the **LSV staircase** command *(see Figure 588, page 499)*:

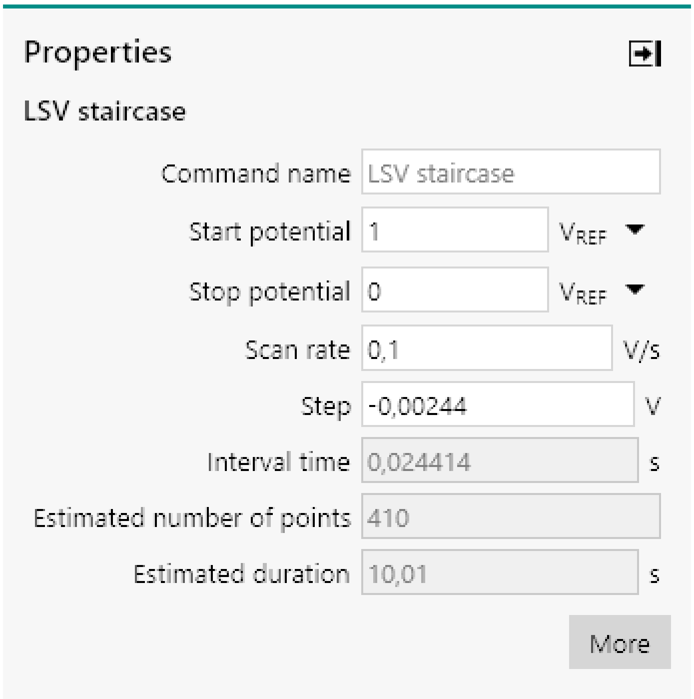


Figure 588 The measurement properties of the LSV staircase command

- **LSV staircase**
  - Start potential: 1 V, versus reference electrode
  - Stop potential: 0 V, versus reference electrode
  - Step: -0.00244 V
  - Scan rate: 100 mV/s

NOTE

The *Step potential* value is negative because the potential scan is performed in the negative going direction.

The procedure samples the following signals *(see Figure 589, page 500)*:

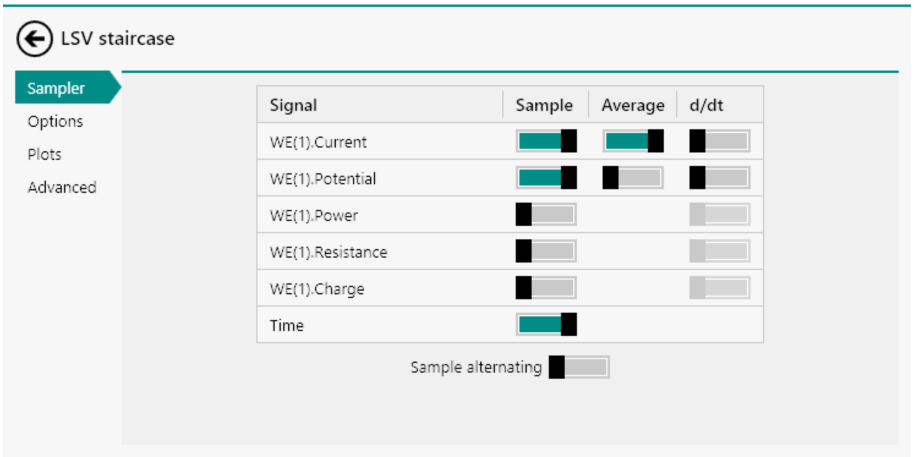


Figure 589 The sampler of the LSV staircase command

- WE(1).Current (averaged)
- WE(1).Potential
- Time

The procedure uses the following options *(see Figure 590, page 500)*:

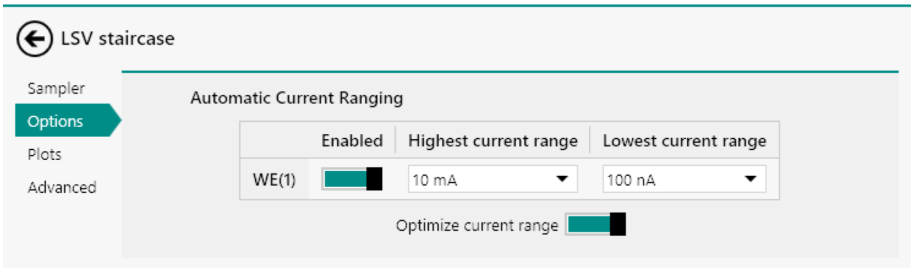


Figure 590 The options of the LSV staircase command

- Automatic current ranging
  - Highest current range: 10 mA
  - Lowest current range: 100 nA

The procedure plots the following data *(see Figure 591, page 501)*:

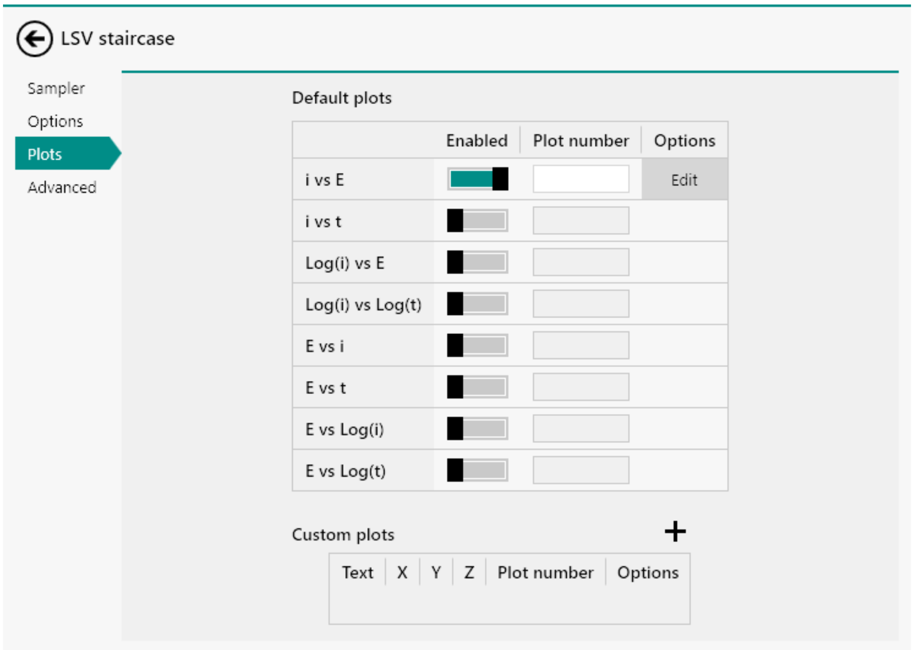


Figure 591 The plots of the LSV staircase command

- i vs E: WE(1).Current versus Potential applied

The procedure also has the value of alpha property available in the Advanced section. This value is set to the default value of 1 *(see Figure 592, page 501)*.

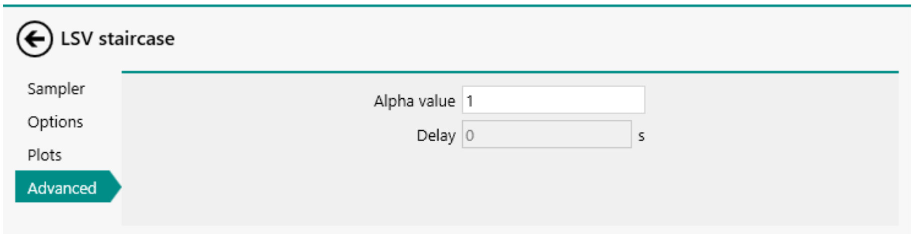


Figure 592 The advanced settings of the LSV staircase command

NOTE

The procedure includes a **Hydrodynamic analysis** command to automatically analyze the measured data. Please refer to *Chapter 7.8.10* for more information.

8.2.5 Hydrodynamic linear sweep with RRDE

CAUTION

This procedure requires the **BA** module *(see Chapter 16.3.2.3, page 990)*.

CAUTION

This procedure requires the optional **Autolab rotating ring disk electrode (RRDE)** connected to the Autolab using the motor controller. The procedure is designed to remotely control the rotation rate. For more information, please refer to the Autolab RDE/RRDE User Manual.

The default **Hydrodynamic linear sweep with RRDE** procedure provides an example of a typical *staircase* linear sweep voltammetry procedure in potentiostatic mode in combination with the **Autolab rotating ring disk electrode (RRDE)** *(see Figure 593, page 502)*.

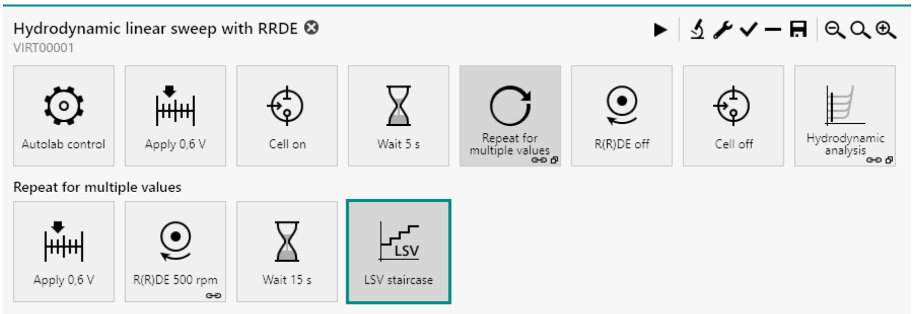


Figure 593 The default Hydrodynamic linear sweep with RRDE procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The Hydrodynamic linear sweep with RRDE procedure performs a linear sweep voltammetry using the Autolab RRDE, with six different rotation rates. The rotation rate of the Autolab RRDE is set using the **R(R)DE** command linked to the values of a **Repeat** command.

The **Repeat** command is used in the *Repeat for multiple values* mode and is preconfigured to cycle through six rotation rates, starting at 500 RPM until 3000 RPM, using a square root distribution *(see Figure 585, page 497)*.

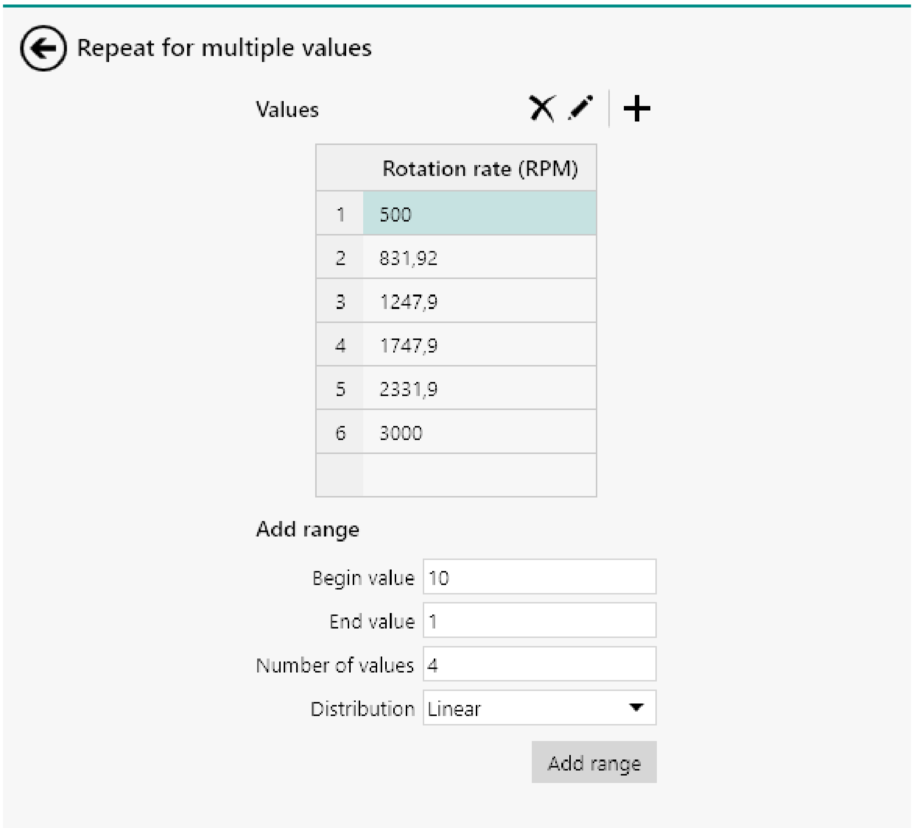


Figure 594 The repeat loop used in the default Hydrodynamic linear sweep procedure

The **Rotation rate** parameter, created by **Repeat** command, is linked to the **R(R)DE** command included in the repeat loop *(see Figure 586, page 498)*.

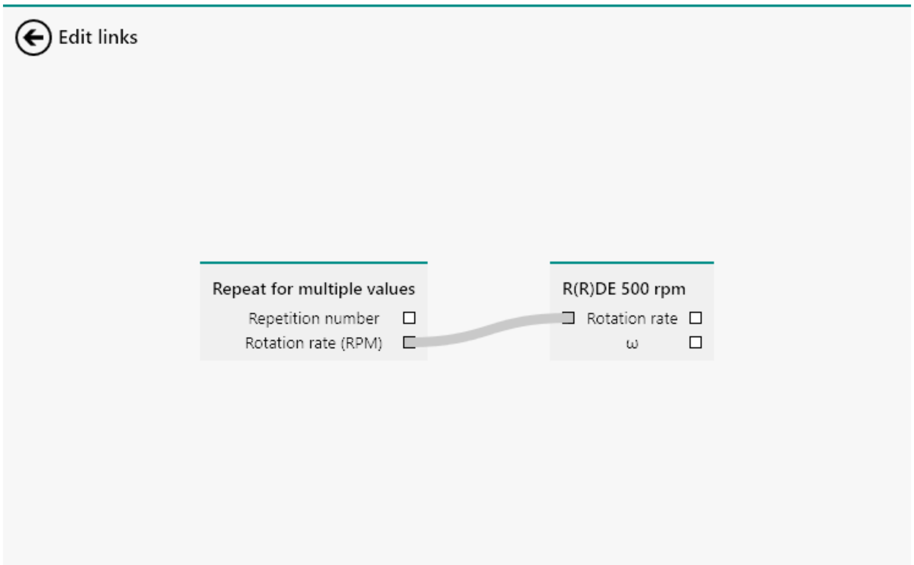


Figure 595 The link used to control the rotation rate of the R(R)DE

This procedure is intended to be used with the **Remote** switch of the Autolab motor controller engaged (on the back plane of the controller) and with a BNC cable connected between the DAC164 ←1 connector (Vout for the PGSTAT204 and M204) and the Remote input plug on the back plane of the Autolab RDE motor controller *(see Figure 587, page 498)*.

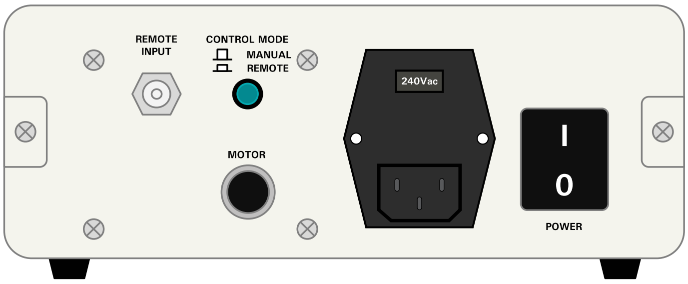


Figure 596 The back plane of the Autolab motor controller

The procedure has the following measurement properties, specified for the **LSV staircase** command *(see Figure 597, page 505)*:

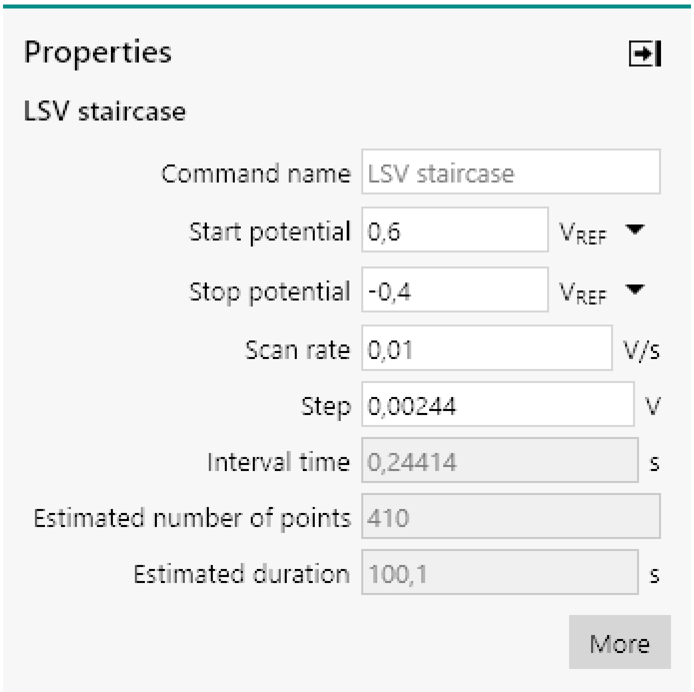


Figure 597 The measurement properties of the LSV staircase command

- **LSV staircase**
  - Start potential: 0.6 V, versus reference electrode
  - Stop potential: -0.4 V, versus reference electrode
  - Step: -0.00244 V
  - Scan rate: 100 mV/s

NOTE

The *Step potential* value is negative because the potential scan is performed in the negative going direction.

The settings of the **BA** module, used to control the ring, are defined using the **Autolab control** command located at the beginning of the procedure *(see Figure 598, page 506)*.

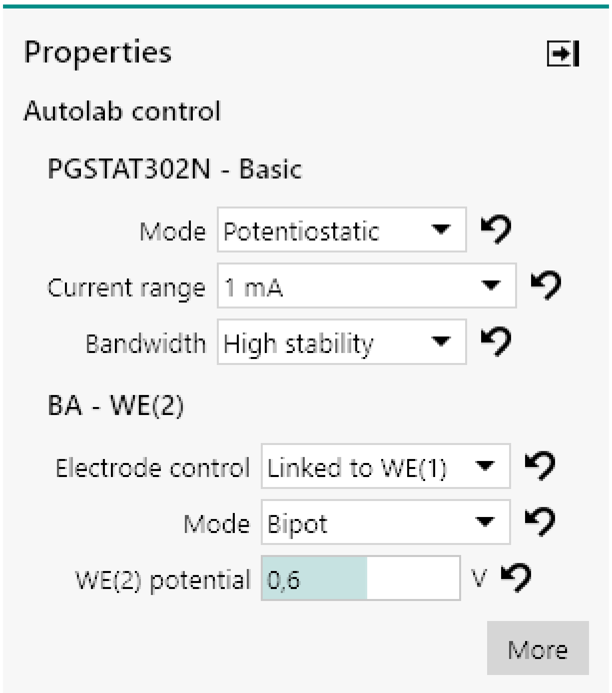


Figure 598 The BA module settings are defined in the Autolab control command

The following settings are specified:

- Electrode control: Linked to WE(1)
- Mode: Bipot
- WE(2) potential: 0.6 V

NOTE

For more information on the **BA** module, please refer to *Chapter 16.3.2.3*.

The procedure samples the following signals *(see Figure 599, page 507)*:

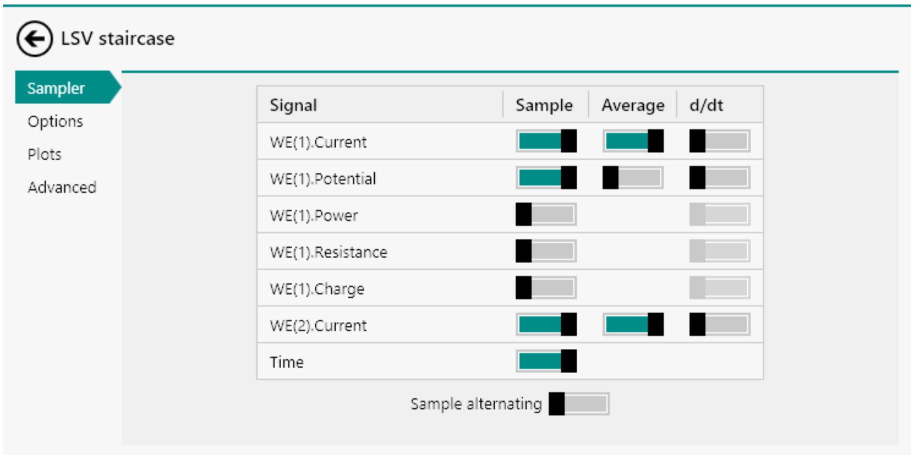


Figure 599 The sampler of the LSV staircase command

- WE(1).Current (averaged)
- WE(1).Potential
- WE(2).Current (averaged)
- Time

The procedure uses the following options *(see Figure 600, page 507)*:

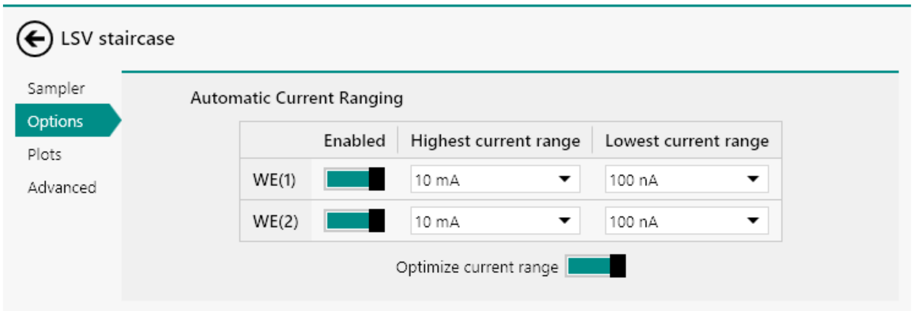


Figure 600 The options of the LSV staircase command

- Automatic current ranging
  - Highest current range: 10 mA
  - Lowest current range: 100 nA

The procedure plots the following data *(see Figure 591, page 501)*:

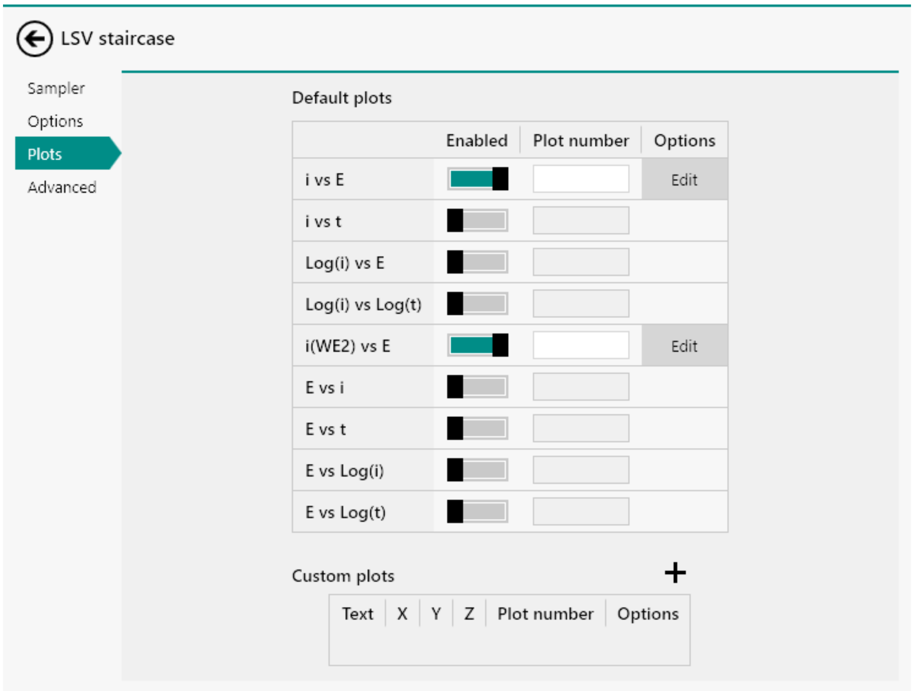


Figure 601 The plots of the LSV staircase command

- i vs E: WE(1).Current versus Potential applied
- i(WE2) vs E: WE(2).Current versus Potential applied

The procedure also has the value of alpha property available in the Advanced section. This value is set to the default value of 1 *(see Figure 592, page 501)*.

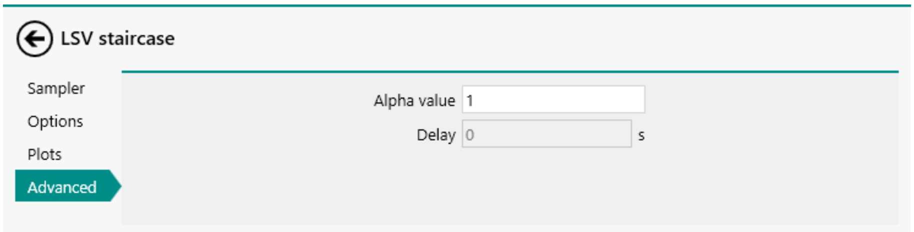


Figure 602 The advanced settings of the LSV staircase command

NOTE

The procedure includes a **Hydrodynamic analysis** command to automatically analyze the measured data. Please refer to *Chapter 7.8.10* for more information.

8.2.6 Spectroelectrochemical linear sweep

CAUTION

This procedure requires an optional **Autolab spectrophotometer** or supported **Avantes spectrophotometer** connected to the Autolab using the required trigger cable.

The default **Spectroelectrochemical linear sweep** procedure provides an example of a typical *staircase* linear sweep voltammetry procedure in potentiostatic mode in combination with the **Autolab spectrophotometer** or supported **Avantes spectrophotometer***(see Figure 603, page 509)*.

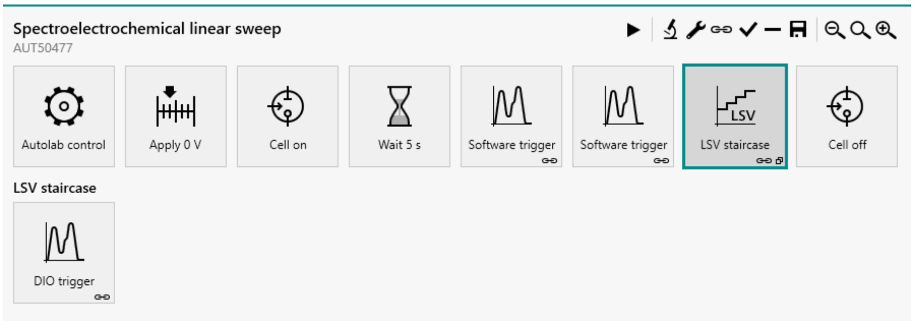


Figure 603 The default Spectroelectrochemical linear sweep procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The **Spectroelectrochemical linear sweep voltammetry** procedure performs a linear sweep voltammetry using the spectrophotometer connected to the computer. The **Spectroscopy** command, included three times in this procedure, is used to measure the dark and reference spectra of the sample, before the linear sweep voltammetry measurement starts and the sample spectra during the execution of the LSV staircase commands, synchronized using a dedicated counter.

The procedure has the following measurement properties, specified for the **LSV staircase** command *(see Figure 604, page 510)*:

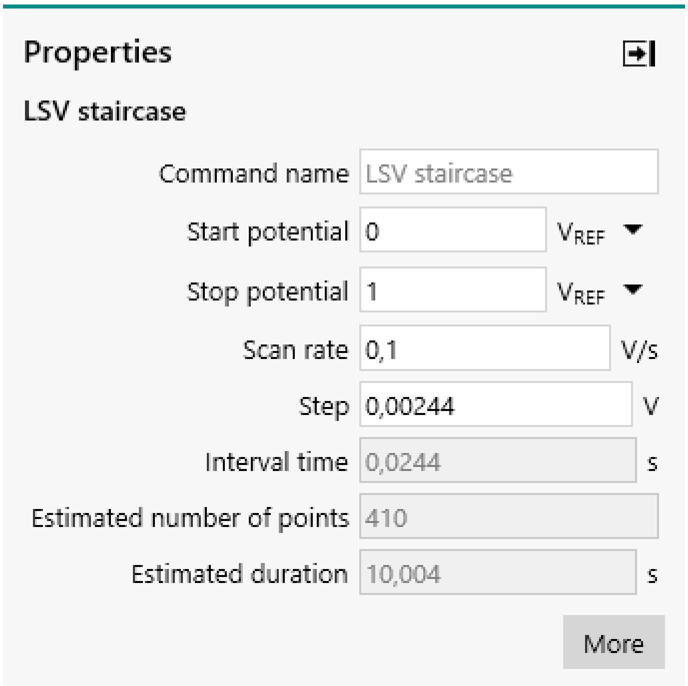


Figure 604 The measurement properties of the LSV staircase command

- **LSV staircase**
  - Start potential: 0 V, versus reference electrode
  - Stop potential: 1 V, versus reference electrode
  - Step: 0.00244 V
  - Scan rate: 100 mV/s

The procedure samples the following signals *(see Figure 605, page 510)*:

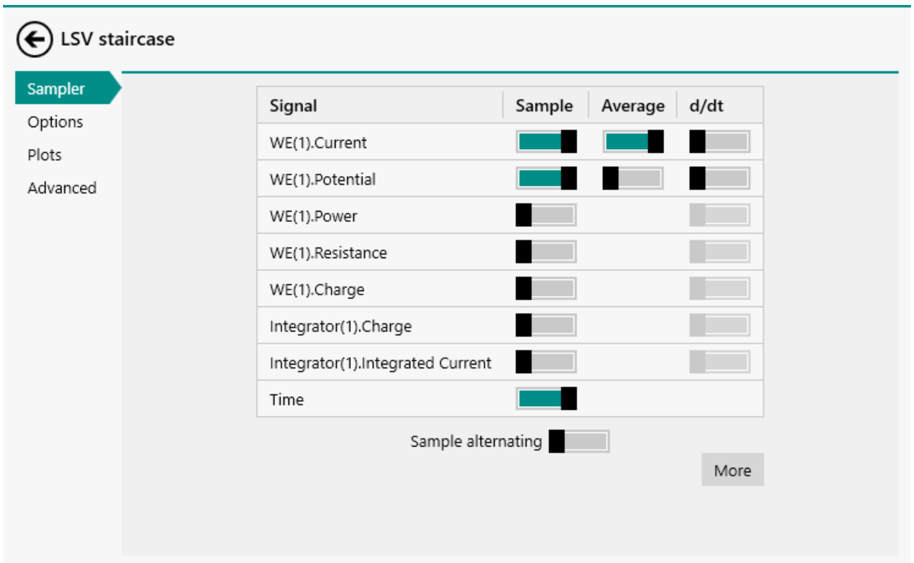


Figure 605 The sampler of the LSV staircase command

- WE(1).Current (averaged)

- WE(1).Potential
- Time

The procedure uses the following options *(see Figure 606, page 511)*:

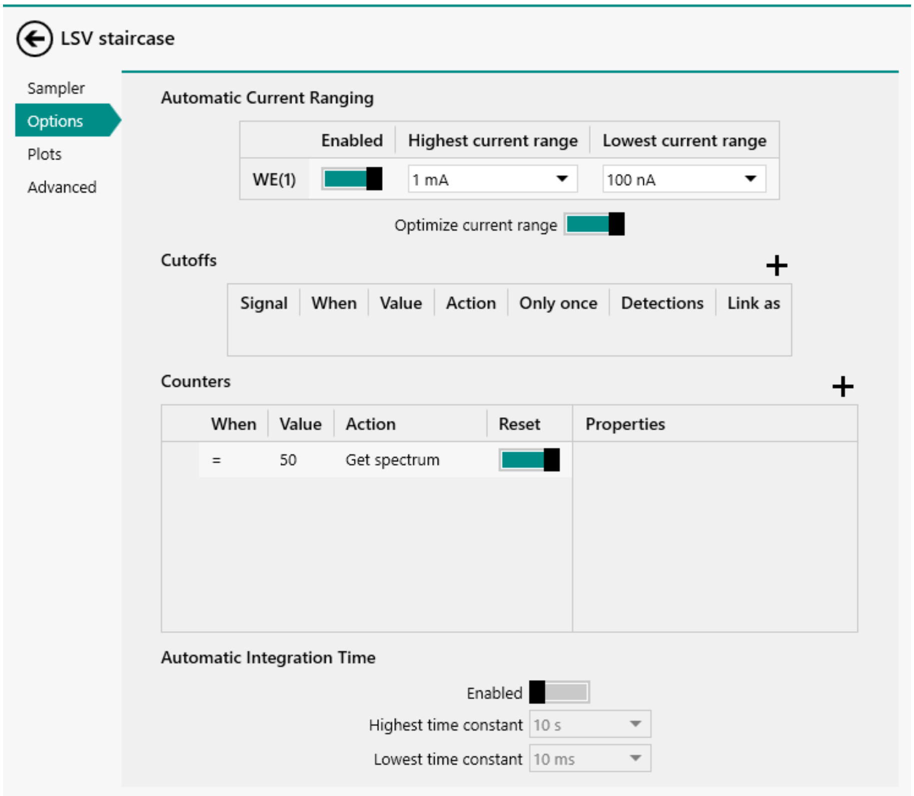


Figure 606 The options of the LSV staircase command

- Automatic current ranging
  - Highest current range: 1 mA
  - Lowest current range: 100 nA
- Counters

  Get spectrum when counter = 50, reset option on

NOTE

The counter option specified in the options of the **LSV staircase** command is used to trigger the acquisition of a spectrum on the connected Autolab or Avantes spectrophotometer. This counter is repeated every 50 points.

The procedure plots the following data *(see Figure 607, page 512)*:

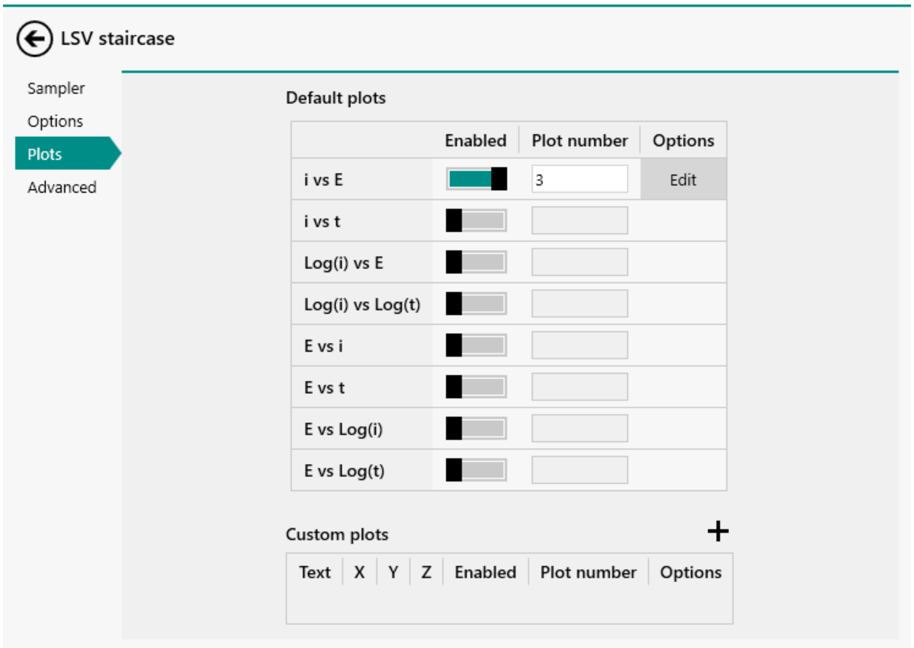


Figure 607 The plots of the LSV staircase command

- i vs E: WE(1).Current versus Potential applied

The procedure also has the value of alpha property available in the Advanced section. This value is set to the default value of 1 *(see Figure 608, page 512)*.

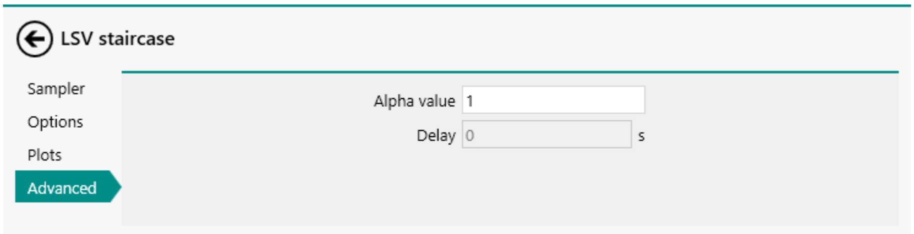


Figure 608 The advanced settings of the LSV staircase command

The **Spectroscopy** command stacked on the **LSV staircase** command is used to acquire the spectroscopy data during the measurement and collect all the of the measured data at the end of the measurement. This command has a number of additional pre-defined plots *(see Figure 609, page 513)*:

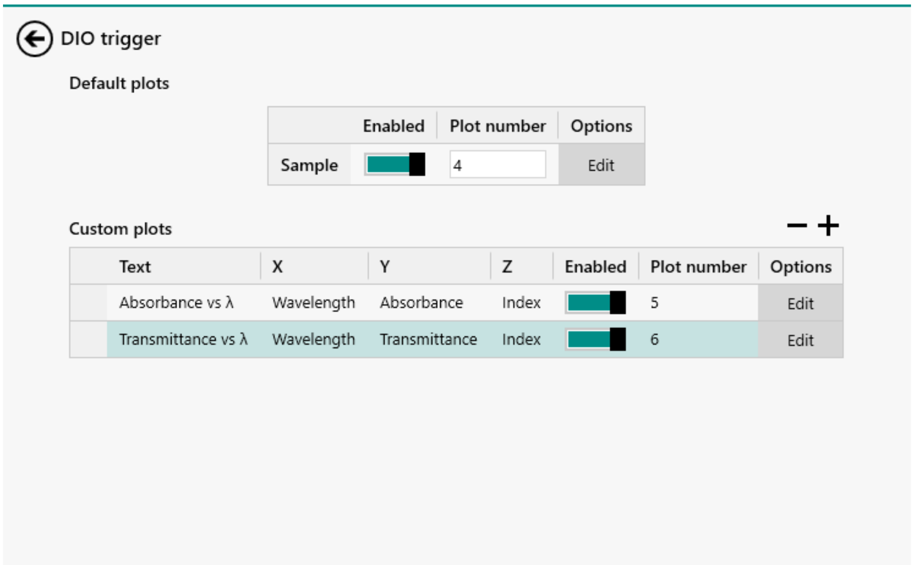


Figure 609 Additional plots defined in the Spectroscopy command

- Sample: measured spectroscopy data versus wavelength
- Absorbance vs λ: calculated absorbance versus wavelength
- Transmittance vs λ: calculated transmittance versus wavelength

NOTE

The absorbance and transmittance values are calculated using the dark and reference data collected by the two **Spectroscopy** commands located before the **LSV staircase** command in the procedure.

8.3 Voltammetric analysis

NOVA provides six default procedures for voltammetric analysis. These procedures can be used to perform a potential sweep with optional pulses or sinewaves and record the response of the cell.

CAUTION

All the procedures included in this group require the optional **IME663** or the optional **IME303**. Please refer to *Chapter 16.3.2.15* and *Chapter 16.3.2.14* for more information.

The following procedures are available:

- Sampled DC polarography
- Normal pulse voltammetry
- Differential pulse voltammetry

- Differential normal pulse voltammetry
- Square wave voltammetry
- AC voltammetry

8.3.1 Sampled DC polarography

CAUTION

This procedure requires a **IME663** *(see Chapter 16.3.2.15, page 1109)* or **IME303** *(see Chapter 16.3.2.14, page 1103)* connected to the Autolab. When this procedure is used without a **IME663** or **IME303**, an **error** will be displayed for the command.

NOTE

To use this procedure without the optional **IME663** or the **IME303**, please delete the **Electrode preconditioning** command group and the **Equilibration** command group.

The default **Sampled DC polarography** procedure provides an example of a typical measurement using the *Sampled DC* method *(see Figure 610, page 514)*.

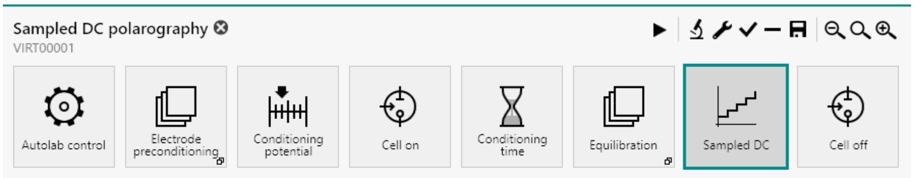


Figure 610 The default Sampled DC polarography procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

This procedure include two command groups, used to the control the mercury drop electrode.

- **Electrode preconditioning:** this command group is used to create new mercury drops at the beginning of the procedure. The commands in this group are used to purge the solution for the specified duration, create the specified number of new drops and switch the stirrer on *(see Figure 611, page 515)*.

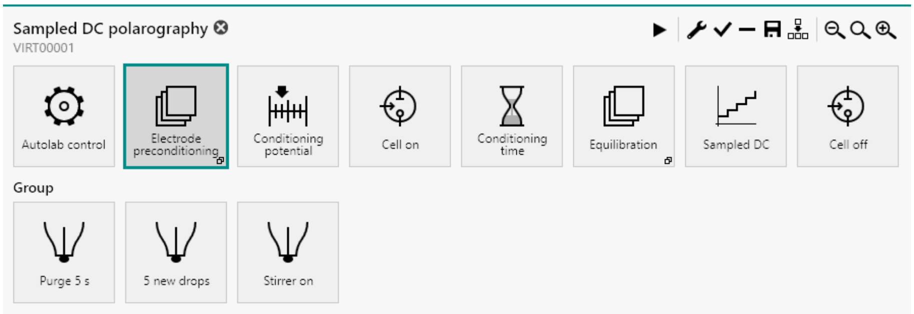


Figure 611 The Electrode preconditioning group

- **Equilibration:** this command group is used to create an equilibration step in the procedure. The commands in this group are used to switch the stirrer off and wait for the specified amount of time *(see Figure 612, page 515)*.

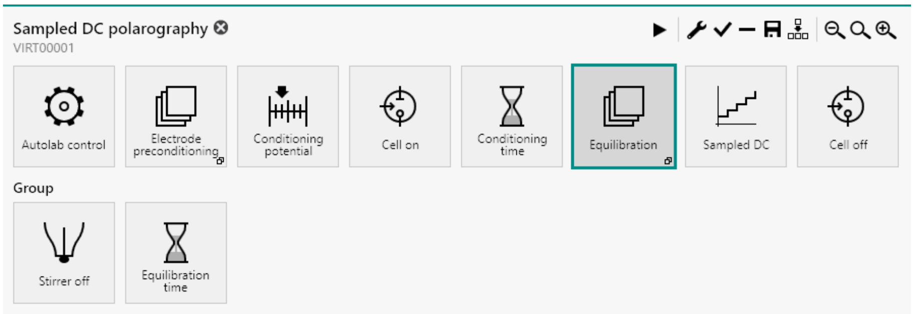


Figure 612 The Equilibration group

The procedure has the following measurement properties, specified for the **Sampled DC** command *(see Figure 613, page 516)*:

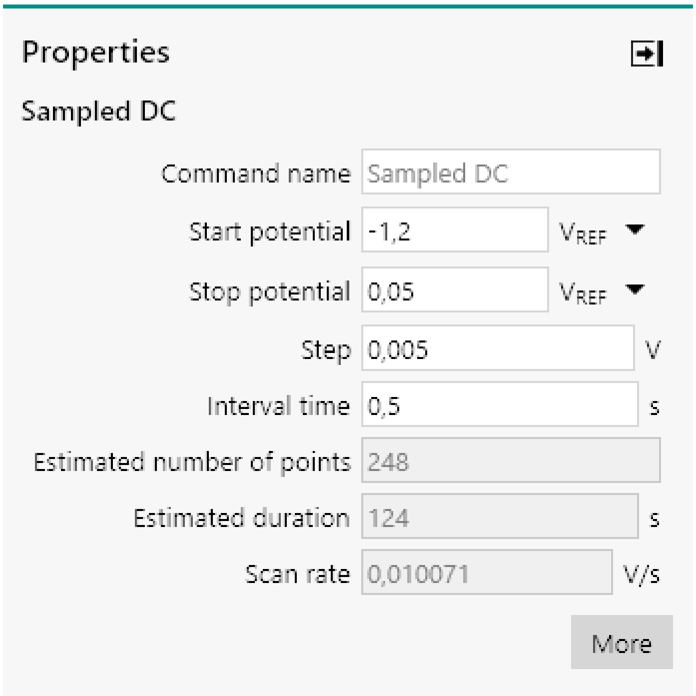


Figure 613 The measurement properties of the Sampled DC command

- **Sampled DC**
  - Start potential: -1.2 V, versus reference electrode
  - Stop potential: 0.05 V, versus reference electrode
  - Step: 0.005 V
  - Interval time: 0.5 s

The procedure samples the following signals *(see Figure 614, page 516)*:

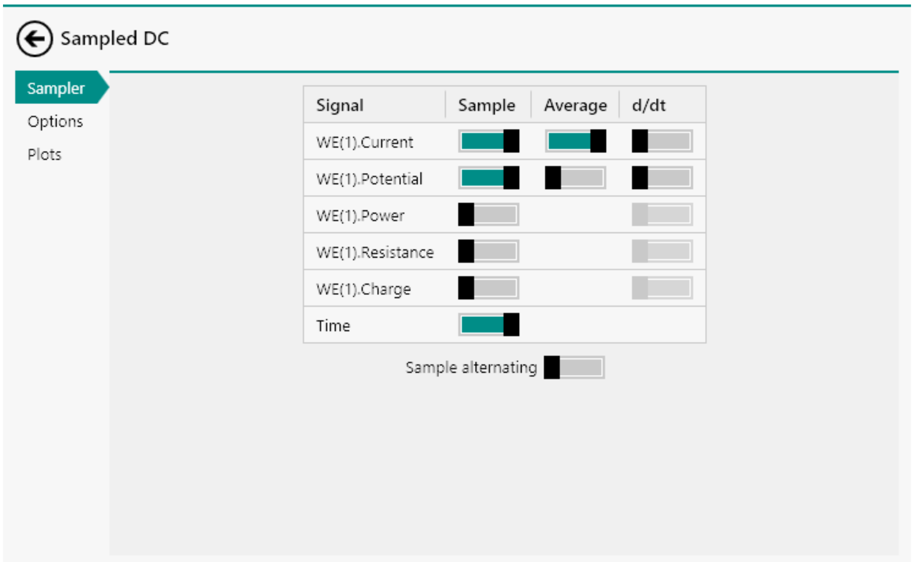


Figure 614 The sampler of the Sampled DC command

- WE(1).Current (averaged)
- WE(1).Potential

- Time

The procedure uses the following options *(see Figure 615, page 517)*:

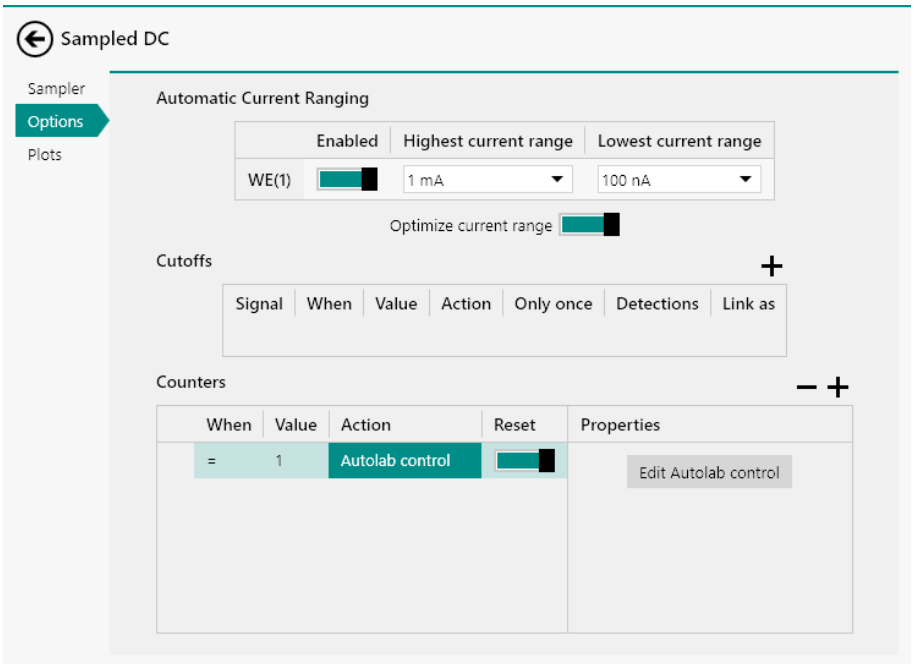


Figure 615 The options of the Sampled DC command

- Automatic current ranging
  - Highest current range: 1 mA
  - Lowest current range: 100 nA
- Counters
  - When counter = 1, Autolab control, Reset

The Counters option, using the procedure, is used to create a new drop with every potential step. The details of the Autolab control action are shown in *Figure 616*.

Figure 616 The Autolab control option triggered with the Counter option

The procedure plots the following data *(see Figure 617, page 518)*:

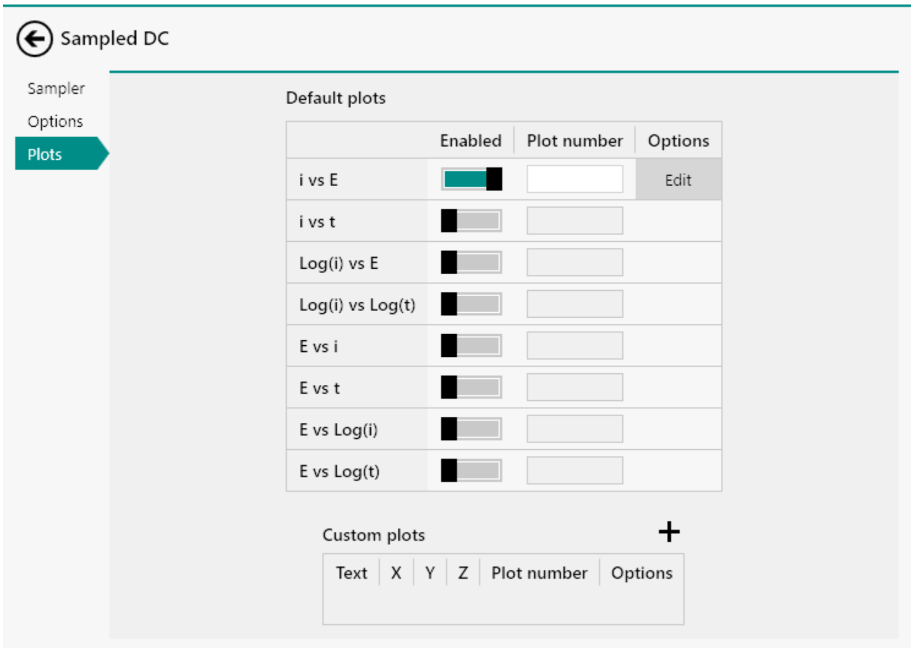


Figure 617 The plots of the Sampled DC command

- i vs E: WE(1).Current versus Potential applied

8.3.2 Normal pulse voltammetry

CAUTION

This procedure requires a **IME663** *(see Chapter 16.3.2.15, page 1109)* or **IME303** *(see Chapter 16.3.2.14, page 1103)* connected to the Autolab. When this procedure is used without a **IME663** or **IME303**, an **error** will be displayed for the command.

NOTE

To use this procedure without the optional **IME663** or the **IME303**, please delete the **Electrode preconditioning** command group and the **Equilibration** command group.

The default **Normal pulse voltammetry** procedure provides an example of a typical measurement using the *Normal pulse* method *(see Figure 618, page 519)*.

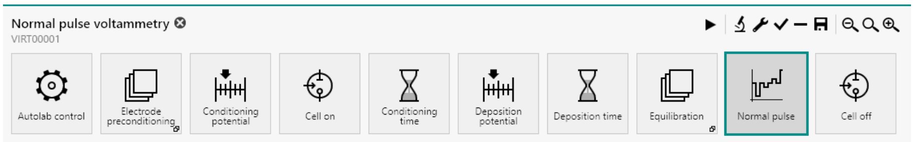


Figure 618 The default Normal pulse voltammetry procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

This procedure include two command groups, used to the control the mercury drop electrode.

- **Electrode preconditioning:** this command group is used to create new mercury drops at the beginning of the procedure. The commands in this group are used to purge the solution for the specified duration, create the specified number of new drops and switch the stirrer on *(see Figure 619, page 520)*.

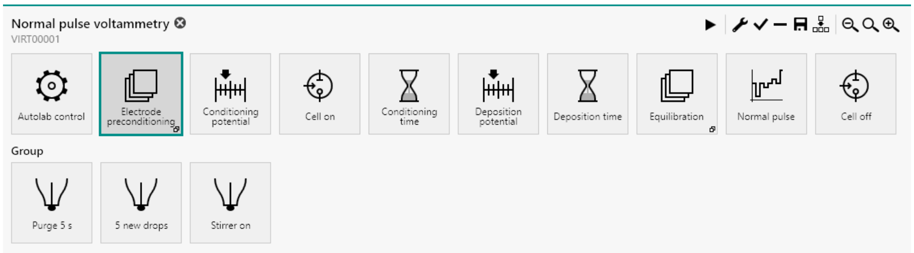


Figure 619 The Electrode preconditioning group

- **Equilibration:** this command group is used to create an equilibration step in the procedure. The commands in this group are used to switch the stirrer off and wait for the specified amount of time *(see Figure 620, page 520)*.

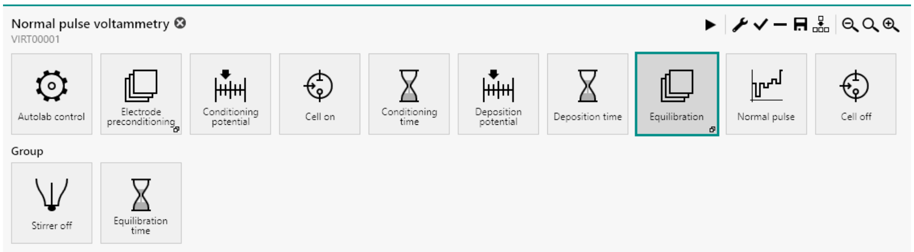


Figure 620 The Equilibration group

The procedure has the following measurement properties, specified for the **Normal pulse** command *(see Figure 621, page 521)*:

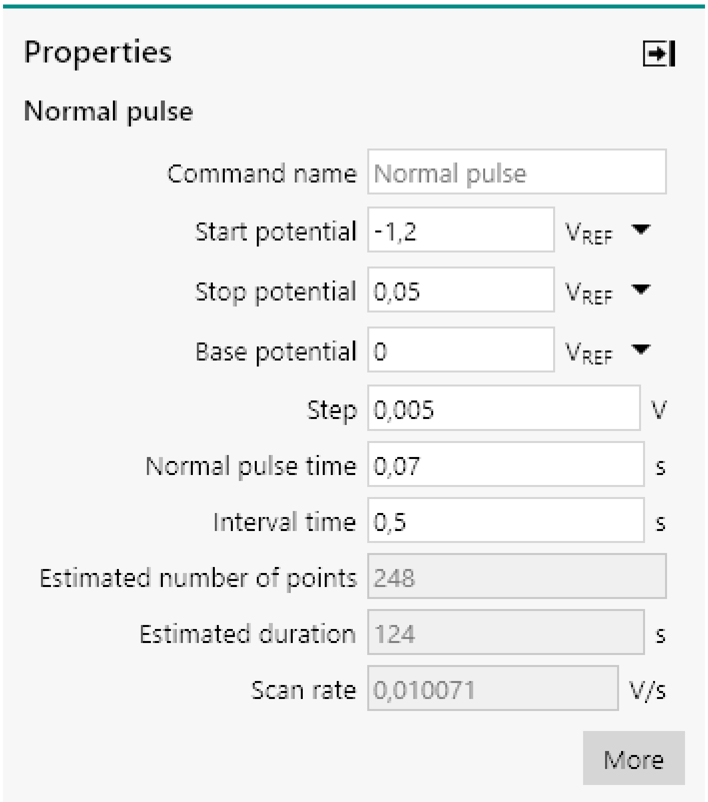


Figure 621 The measurement properties of the Normal pulse command

- **Normal pulse**
  - Start potential: -1.2 V, versus reference electrode
  - Stop potential: 0.07 V, versus reference electrode
  - Step: 0.005 V
  - Base potential: 0 V, versus reference electrode
  - Normal pulse time: 0.07 s
  - Interval time: 0.5 V

The procedure samples the following signals *(see Figure 622, page 522)*:

Figure 622 The sampler of the Normal pulse command

- WE(1).Current (averaged)
- WE(1).Potential
- Time

The procedure uses the following options *(see Figure 623, page 522)*:

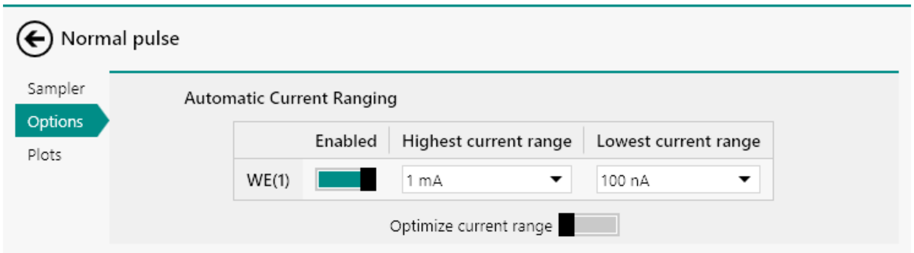


Figure 623 The options of the Normal pulse command

- Automatic current ranging
  - Highest current range: 1 mA
  - Lowest current range: 100 nA

The procedure plots the following data *(see Figure 624, page 523)*:

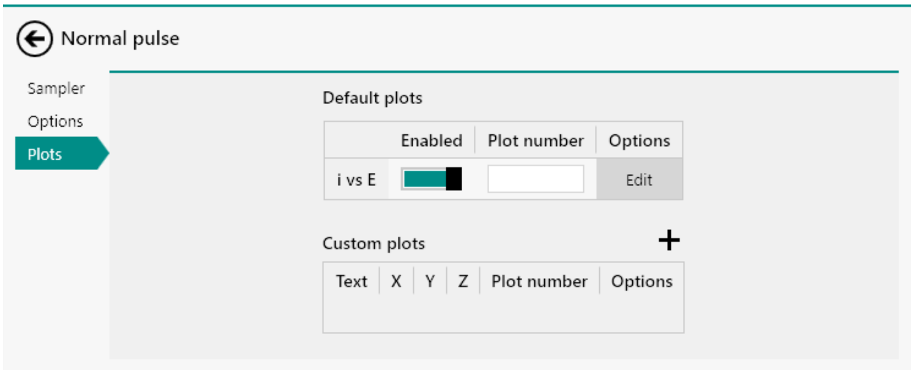


Figure 624 The plots of the Normal pulse command

- i vs E: WE(1).Current versus Potential applied

8.3.3 Differential pulse voltammetry

CAUTION

This procedure requires a **IME663** *(see Chapter 16.3.2.15, page 1109)* or **IME303** *(see Chapter 16.3.2.14, page 1103)* connected to the Autolab. When this procedure is used without a **IME663** or **IME303**, an **error** will be displayed for the command.

NOTE

To use this procedure without the optional **IME663** or the **IME303**, please delete the **Electrode preconditioning** command group and the **Equilibration** command group.

The default **Differential pulse voltammetry** procedure provides an example of a typical measurement using the *Differential pulse* method *(see Figure 625, page 523)*.

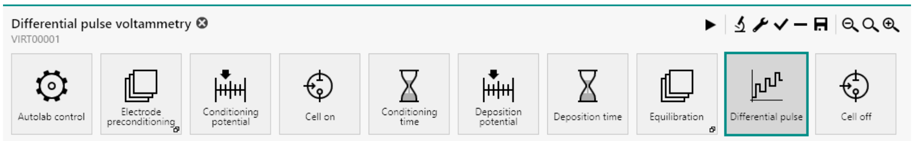


Figure 625 The default Differential pulse voltammetry procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

This procedure include two command groups, used to the control the mercury drop electrode.

- **Electrode preconditioning:** this command group is used to create new mercury drops at the beginning of the procedure. The commands in this group are used to purge the solution for the specified duration, create the specified number of new drops and switch the stirrer on *(see Figure 626, page 524)*.

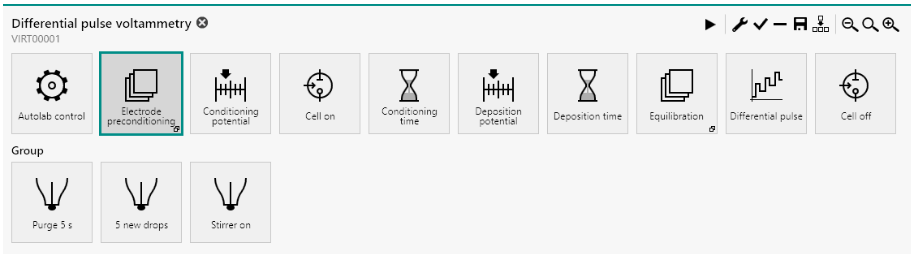


Figure 626 The Electrode preconditioning group

- **Equilibration:** this command group is used to create an equilibration step in the procedure. The commands in this group are used to switch the stirrer off and wait for the specified amount of time *(see Figure 627, page 524)*.

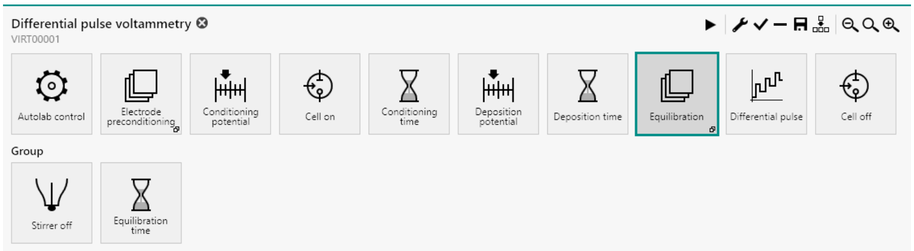


Figure 627 The Equilibration group

The procedure has the following measurement properties, specified for the **Differential pulse** command *(see Figure 628, page 525)*:

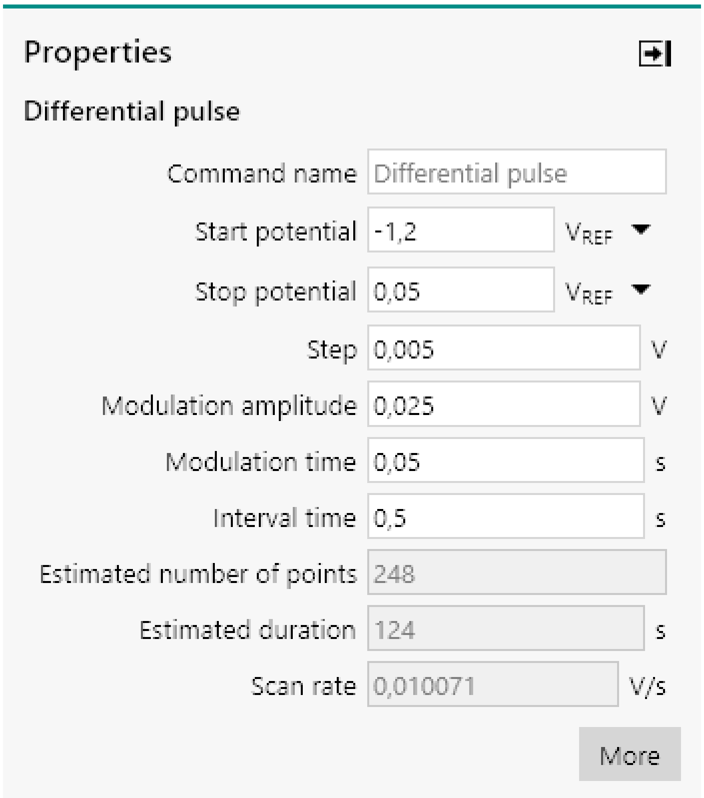


Figure 628 The measurement properties of the Differential pulse command

- **Differential pulse**
  - Start potential: -1.2 V, versus reference electrode
  - Stop potential: 0.05 V, versus reference electrode
  - Step: 0.005 V
  - Modulation amplitude: 0.025 V
  - Modulation time: 0.05 s
  - Interval time: 0.5 s

The procedure samples the following signals *(see Figure 629, page 526)*:

Figure 629 The sampler of the Differential pulse command

- WE(1).Current (averaged)
- WE(1).Potential
- Time

The procedure uses the following options *(see Figure 630, page 526)*:

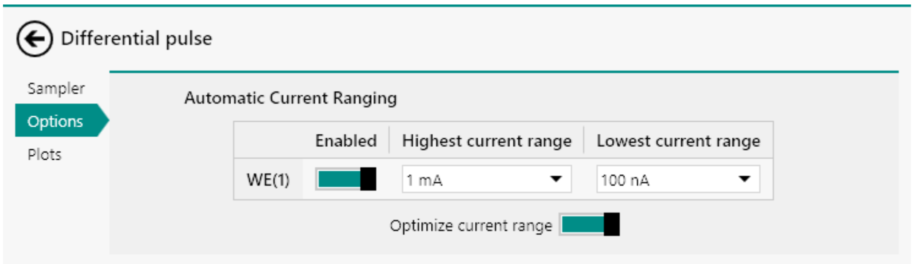


Figure 630 The options of the Differential pulse command

- Automatic current ranging
  - Highest current range: 1 mA
  - Lowest current range: 100 nA

The procedure plots the following data *(see Figure 631, page 527)*:

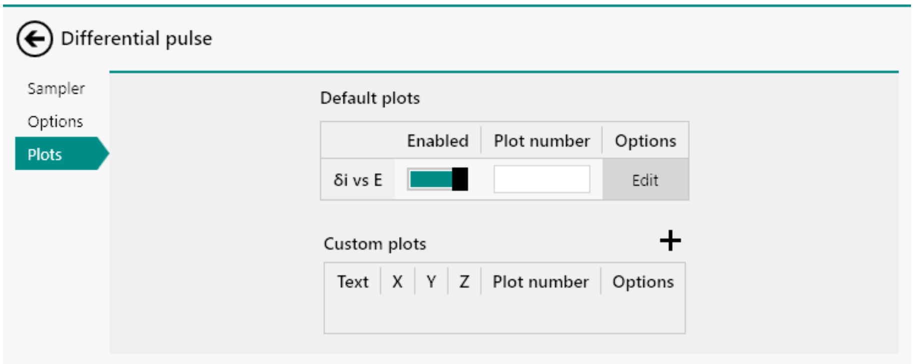


Figure 631 The plots of the Differential pulse command

- δi vs E: δ[WE(1).Current] versus Potential applied

8.3.4 Differential normal pulse voltammetry

CAUTION

This procedure requires a **IME663** *(see Chapter 16.3.2.15, page 1109)* or **IME303** *(see Chapter 16.3.2.14, page 1103)* connected to the Autolab. When this procedure is used without a **IME663** or **IME303**, an **error** will be displayed for the command.

NOTE

To use this procedure without the optional **IME663** or the **IME303**, please delete the **Electrode preconditioning** command group and the **Equilibration** command group.

The default **Differential normal pulse voltammetry** procedure provides an example of a typical measurement using the *Differential normal pulse* method *(see Figure 632, page 527)*.

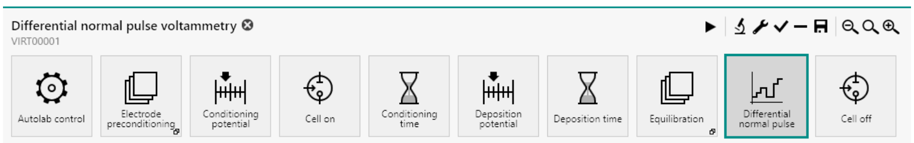


Figure 632 The default Differential normal pulse voltammetry procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

This procedure include two command groups, used to the control the mercury drop electrode.

- **Electrode preconditioning:** this command group is used to create new mercury drops at the beginning of the procedure. The commands in this group are used to purge the solution for the specified duration, create the specified number of new drops and switch the stirrer on *(see Figure 633, page 528)*.

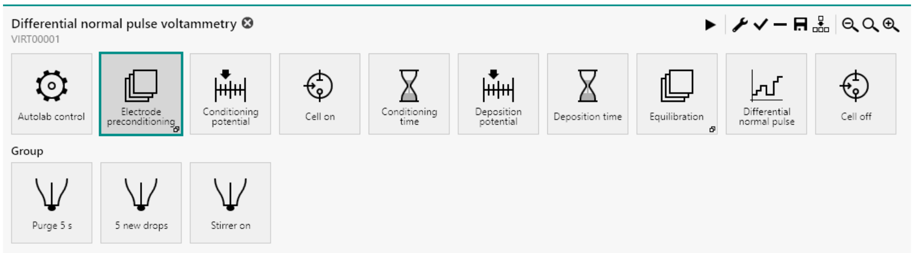


Figure 633 The Electrode preconditioning group

- **Equilibration:** this command group is used to create an equilibration step in the procedure. The commands in this group are used to switch the stirrer off and wait for the specified amount of time *(see Figure 634, page 528)*.

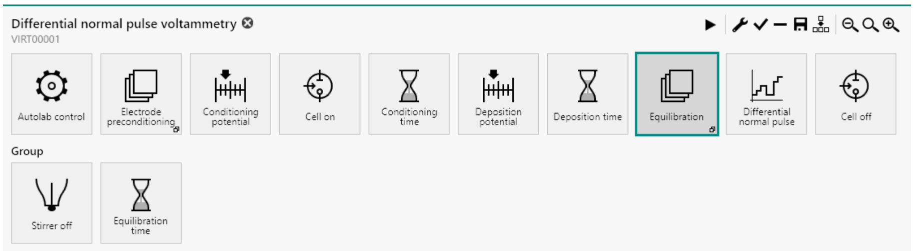


Figure 634 The Equilibration group

The procedure has the following measurement properties, specified for the **Differential normal pulse** command *(see Figure 635, page 529)*:

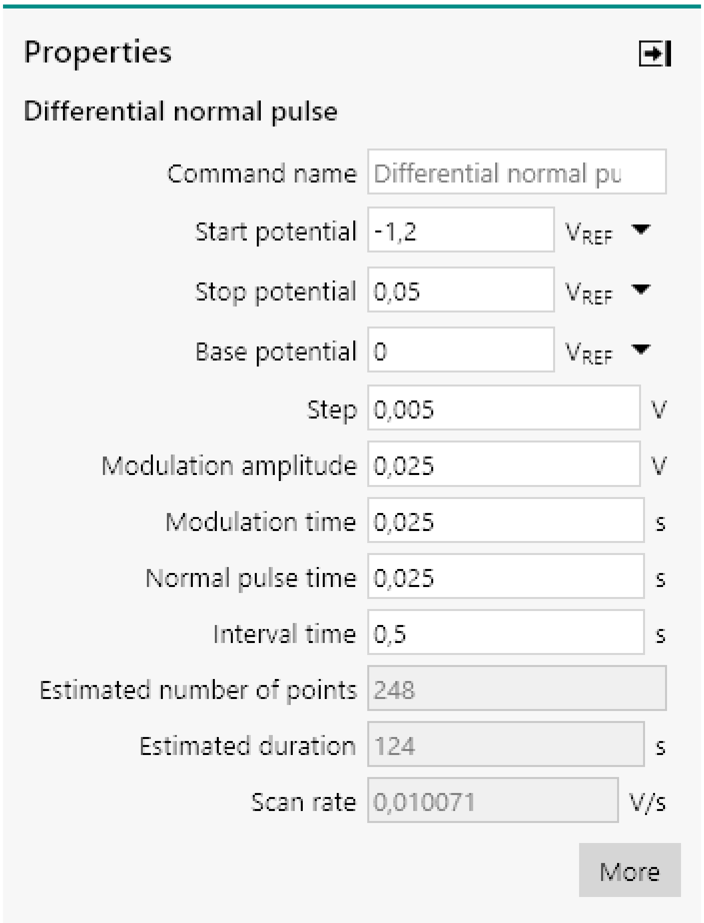


Figure 635 The measurement properties of the Differential normal pulse command

- **Differential normal pulse**
  - Start potential: -1.2 V, versus reference electrode
  - Stop potential: 0.07 V, versus reference electrode
  - Step: 0.005 V
  - Base potential: 0 V, versus reference electrode
  - Modulation amplitude: 0.025 V
  - Normal pulse time: 0.025 s
  - Interval time: 0.5 V
  - Modulation time: 0.025 s

The procedure samples the following signals *(see Figure 636, page 530)*:

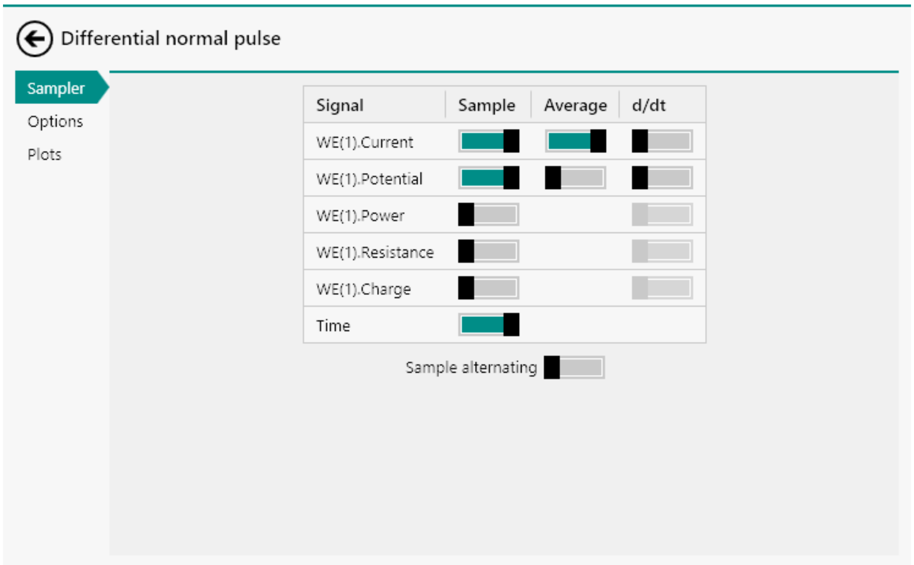


Figure 636 The sampler of the Differential normal pulse command

- WE(1).Current (averaged)
- WE(1).Potential
- Time

The procedure uses the following options *(see Figure 637, page 530)*:

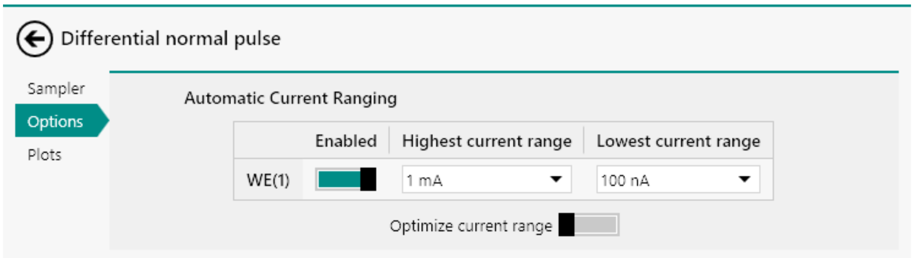


Figure 637 The options of the Differential normal pulse command

- Automatic current ranging
  - Highest current range: 1 mA
  - Lowest current range: 100 nA

The procedure plots the following data *(see Figure 638, page 530)*:

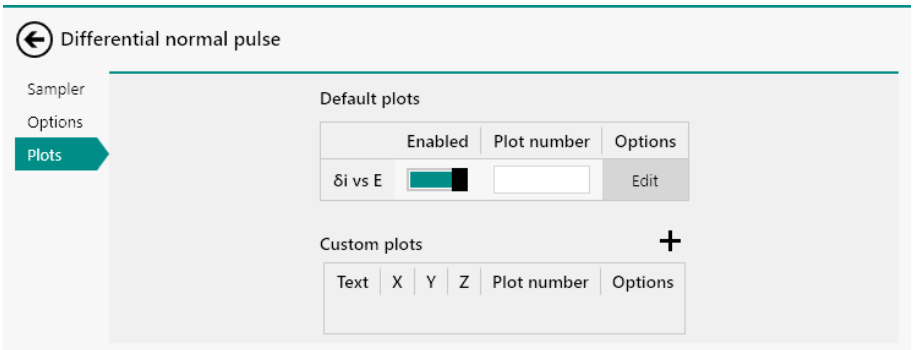


Figure 638 The plots of the Differential normal pulse command

- δi vs E: δ[WE(1).Current] versus Potential applied

8.3.5 Square wave voltammetry

CAUTION

This procedure requires a **IME663** *(see Chapter 16.3.2.15, page 1109)* or **IME303** *(see Chapter 16.3.2.14, page 1103)* connected to the Autolab. When this procedure is used without a **IME663** or **IME303**, an **error** will be displayed for the command.

NOTE

To use this procedure without the optional **IME663** or the **IME303**, please delete the **Electrode preconditioning** command group and the **Equilibration** command group.

The default **Square wave voltammetry** procedure provides an example of a typical measurement using the *Square wave* method *(see Figure 639, page 531)*.

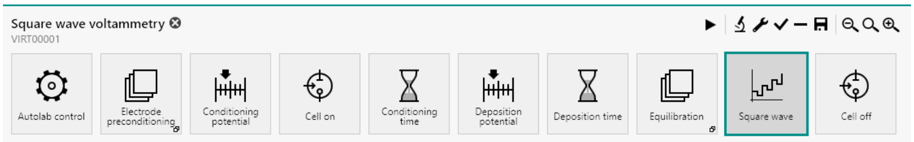


Figure 639 The default Square wave voltammetry procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

This procedure include two command groups, used to the control the mercury drop electrode.

- **Electrode preconditioning:** this command group is used to create new mercury drops at the beginning of the procedure. The commands in this group are used to purge the solution for the specified duration, create the specified number of new drops and switch the stirrer on *(see Figure 640, page 532)*.

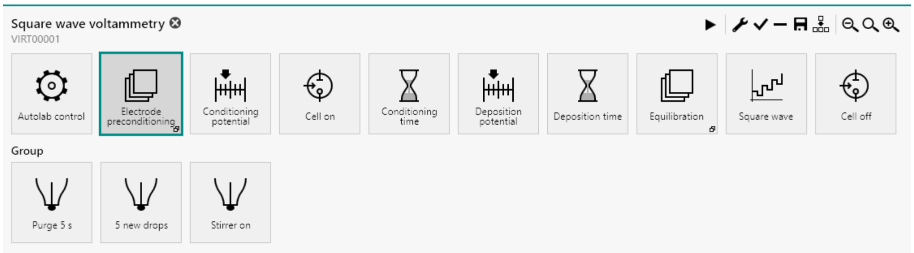


Figure 640 The Electrode preconditioning group

- **Equilibration:** this command group is used to create an equilibration step in the procedure. The commands in this group are used to switch the stirrer off and wait for the specified amount of time *(see Figure 641, page 532)*.

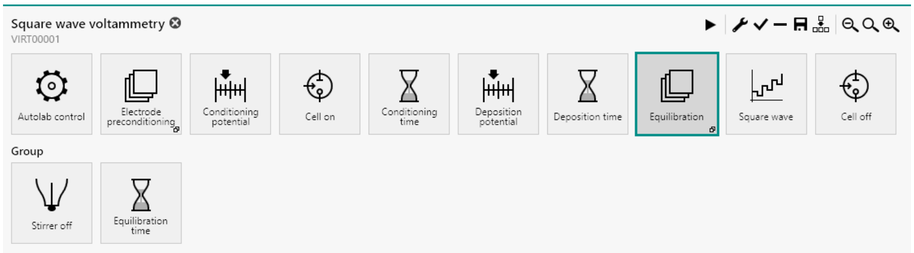


Figure 641 The Equilibration group

The procedure has the following measurement properties, specified for the **Square wave** command *(see Figure 642, page 533)*:

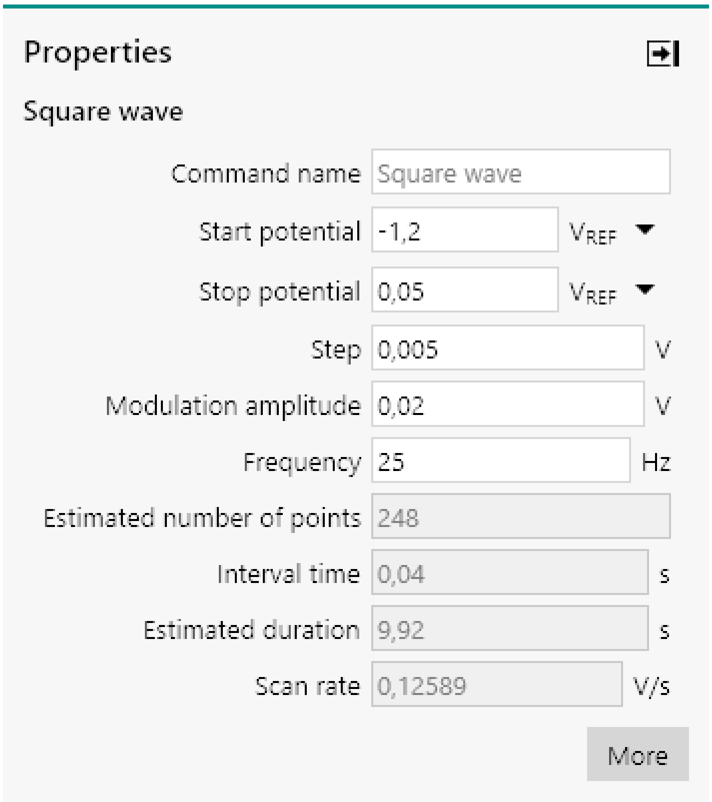


Figure 642 The measurement properties of the Square wave command

- **Square wave**
  - Start potential: -1.2 V, versus reference electrode
  - Stop potential: 0.07 V, versus reference electrode
  - Step: 0.005 V
  - Amplitude: 0.02 V
  - Frequency: 25 Hz

The procedure samples the following signals *(see Figure 643, page 534)*:

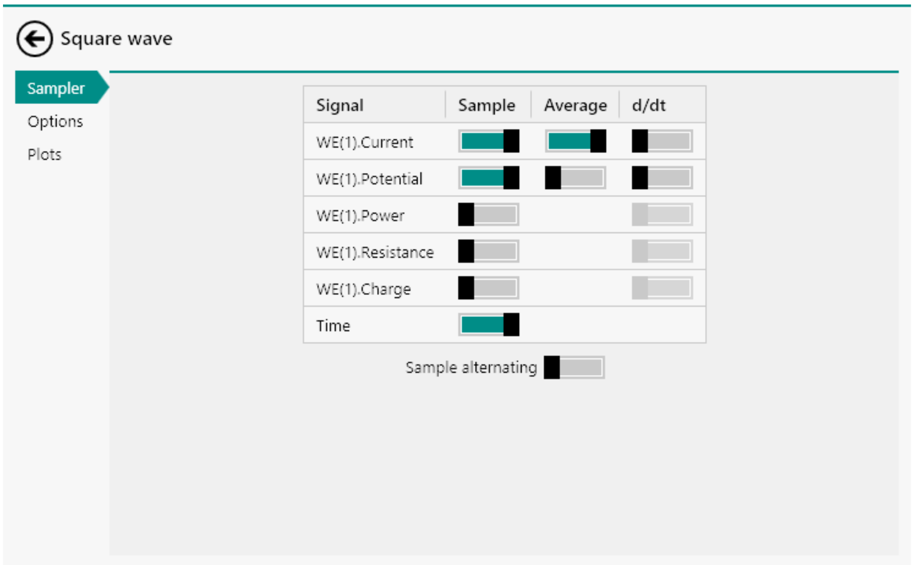


Figure 643 The sampler of the Square wave command

- WE(1).Current (averaged)
- WE(1).Potential
- Time

The procedure uses the following options *(see Figure 644, page 534)*:

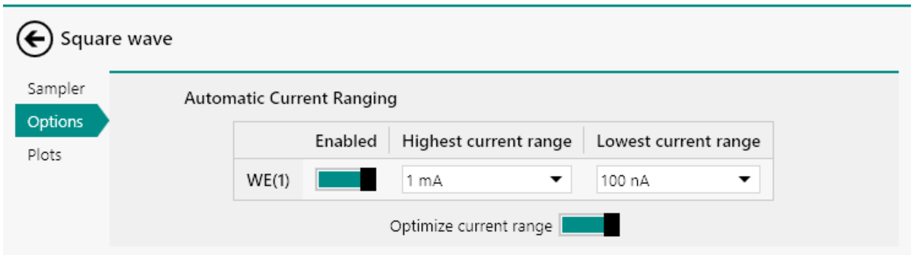


Figure 644 The options of the Square wave command

- Automatic current ranging
  - Highest current range: 1 mA
  - Lowest current range: 100 nA

The procedure plots the following data *(see Figure 645, page 535)*:

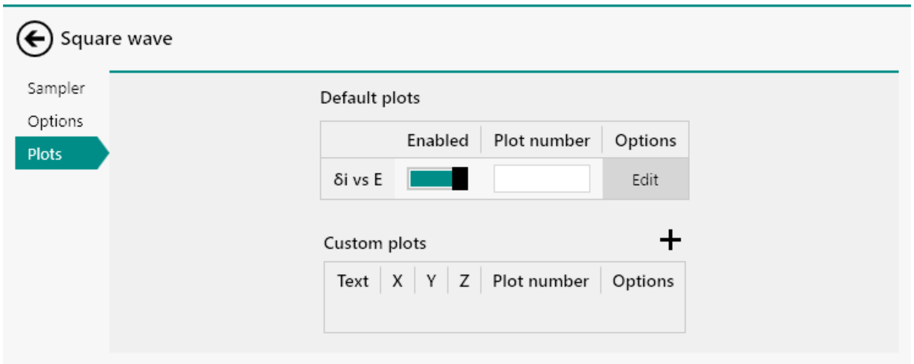


Figure 645 The plots of the Square wave command

- δi vs E: δ[WE(1).Current] versus Potential applied

8.3.6 AC voltammetry

CAUTION

This procedure requires a **IME663** *(see Chapter 16.3.2.15, page 1109)* or **IME303** *(see Chapter 16.3.2.14, page 1103)* connected to the Autolab. When this procedure is used without a **IME663** or **IME303**, an **error** will be displayed for the command.

NOTE

To use this procedure without the optional **IME663** or the **IME303**, please delete the **Electrode preconditioning** command group and the **Equilibration** command group.

The default **AC voltammetry** procedure provides an example of a typical measurement using the *AC voltammetry* method *(see Figure 646, page 535)*.

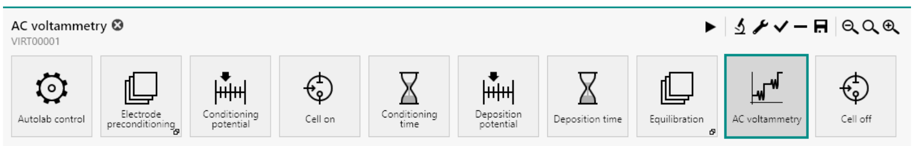


Figure 646 The default AC voltammetry procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

This procedure include two command groups, used to the control the mercury drop electrode.

- **Electrode preconditioning:** this command group is used to create new mercury drops at the beginning of the procedure. The commands in this group are used to purge the solution for the specified duration, create the specified number of new drops and switch the stirrer on *(see Figure 647, page 536)*.

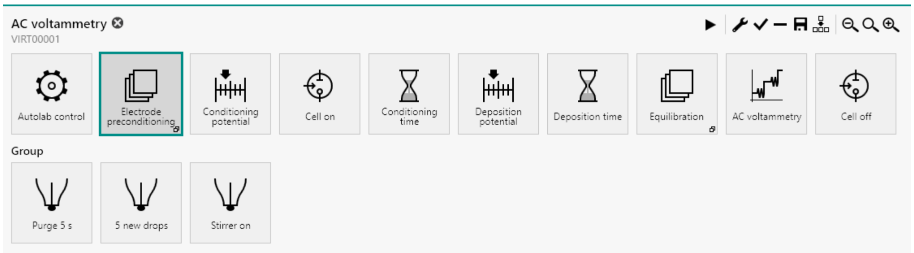


Figure 647 The Electrode preconditioning group

- **Equilibration:** this command group is used to create an equilibration step in the procedure. The commands in this group are used to switch the stirrer off and wait for the specified amount of time *(see Figure 648, page 536)*.

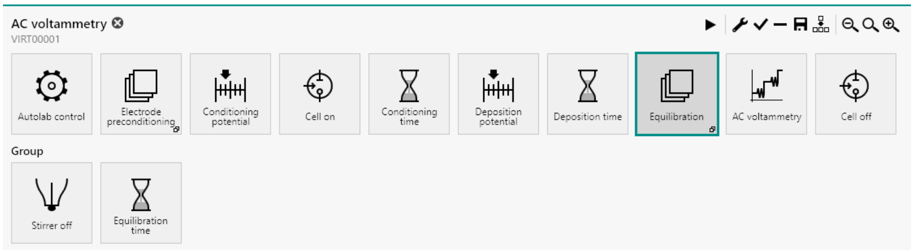


Figure 648 The Equilibration group

The procedure has the following measurement properties, specified for the **AC voltammetry** command *(see Figure 649, page 537)*:

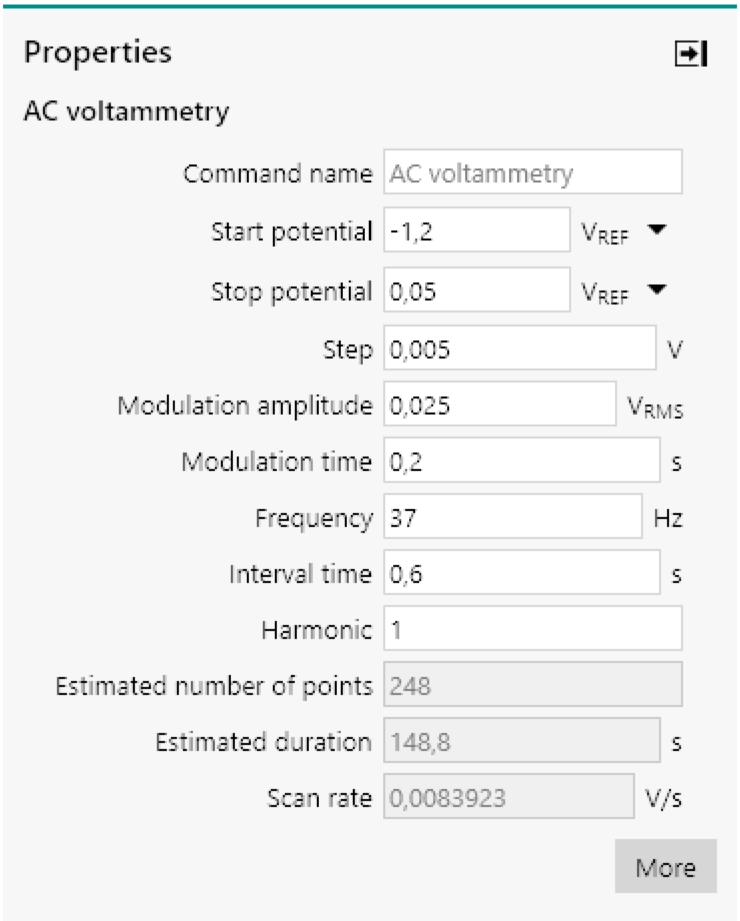


Figure 649 The measurement properties of the AC voltammetry command

- **AC voltammetry**
  - Start potential: -1.2 V, versus reference electrode
  - Stop potential: 0.05 V, versus reference electrode
  - Step: 0.005 V
  - Modulation amplitude: 0.025 V RMS
  - Modulation time: 0.2 s
  - Frequency: 37 Hz
  - Interval time: 0.6 s
  - Harmonic: 1

The procedure samples the following signals *(see Figure 650, page 538)*:

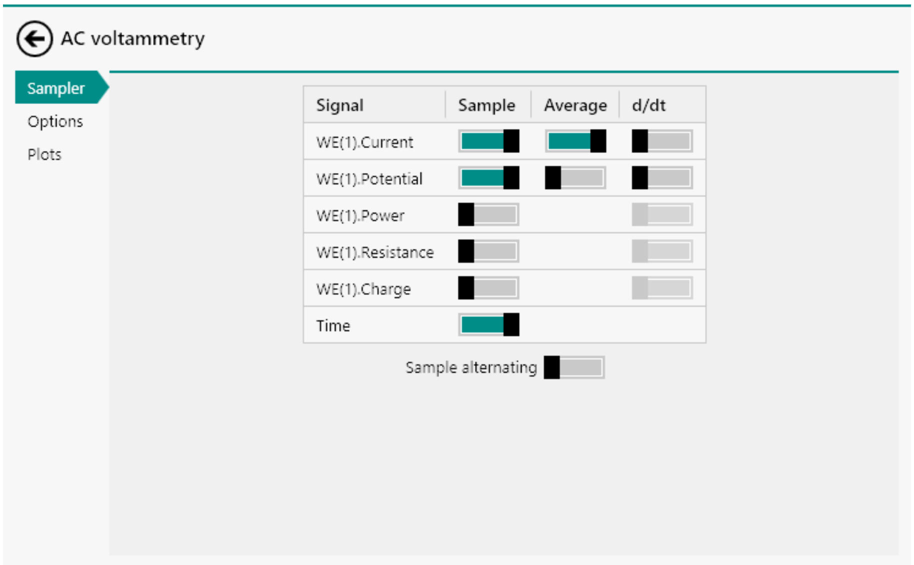


Figure 650 The sampler of the AC voltammetry command

- WE(1).Current (averaged)
- WE(1).Potential
- Time

The procedure uses the following options *(see Figure 651, page 538)*:

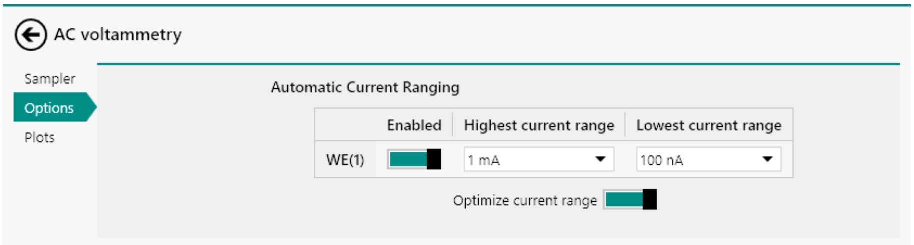


Figure 651 The options of the AC voltammetry command

- Automatic current ranging
  - Highest current range: 1 mA
  - Lowest current range: 100 nA

The procedure plots the following data *(see Figure 652, page 539)*:

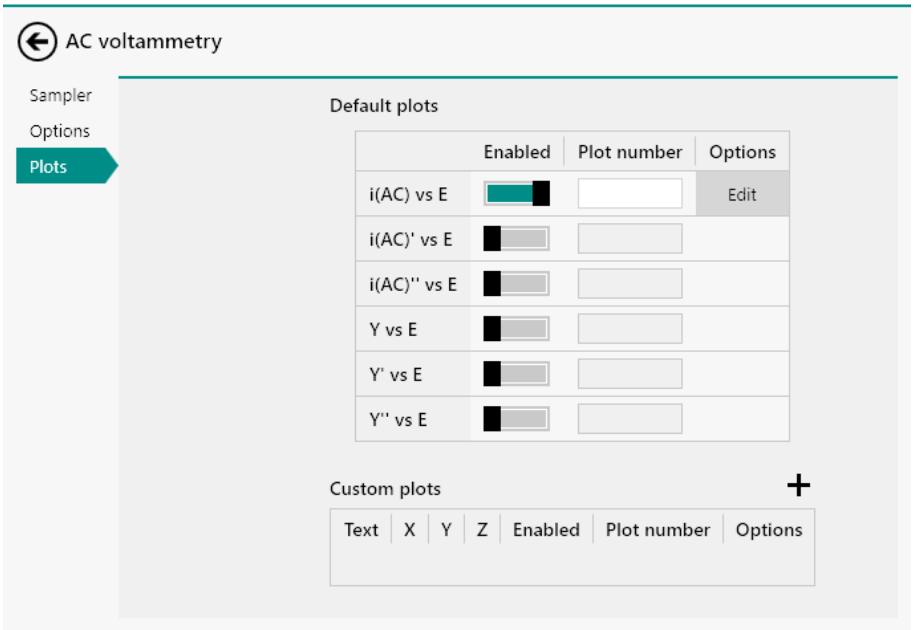


Figure 652 The plots of the AC voltammetry command

- i(AC) vs E: AC current versus Potential applied

8.4 Chrono methods

NOVA provides nine default procedures for chrono methods. These procedures can be used to perform time resolved measurements. Some of these procedures require optional hardware extensions.

The following procedures are available:

- Chrono amperometry (Δt > 1 ms)
- Chrono coulometry (Δt > 1 ms) (requires the **FI20** module or the **on-board integrator**, please refer to *Chapter 16.3.2.11* for more information)
- Chrono potentiometry (Δt > 1 ms)
- Chrono amperometry fast
- Chrono coulometry fast (requires the **FI20** module or the **on-board integrator**, please refer to *Chapter 16.3.2.11* for more information)
- Chrono potentiometry fast
- Chrono amperometry high speed (requires the **ADC10M** or **ADC750** module, please refer to *Chapter 16.3.2.1* for more information)
- Chrono potentiometry high speed (requires the **ADC10M** or **ADC750** module, please refer to *Chapter 16.3.2.1* for more information)
- Chrono charge discharge

8.4.1 Chrono amperometry (Δt > 1 ms)

The default **Chrono amperometry (Δt > 1 ms)** procedure provides an example of a typical chrono amperometric measurement using a sequence of potential steps *(see Figure 653, page 540)*.

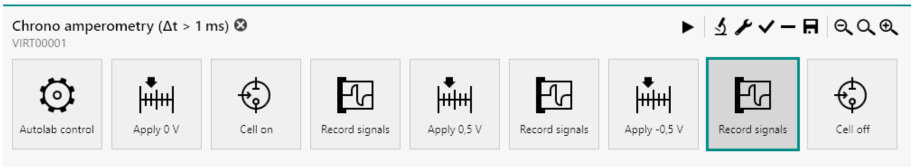


Figure 653 The default Chrono amperometry (Δt > 1 ms) procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure uses a sequence of three potential values (specified through the **Apply** command) followed by three **Record signals** commands.

NOTE

The smallest possible interval time for the **Record signals** command is 1.3 ms.

The potential values applied are 0 V, 0.5 V and -0.5 V. The **Record signals** commands have the following measurement properties *(see Figure 654, page 541)*:

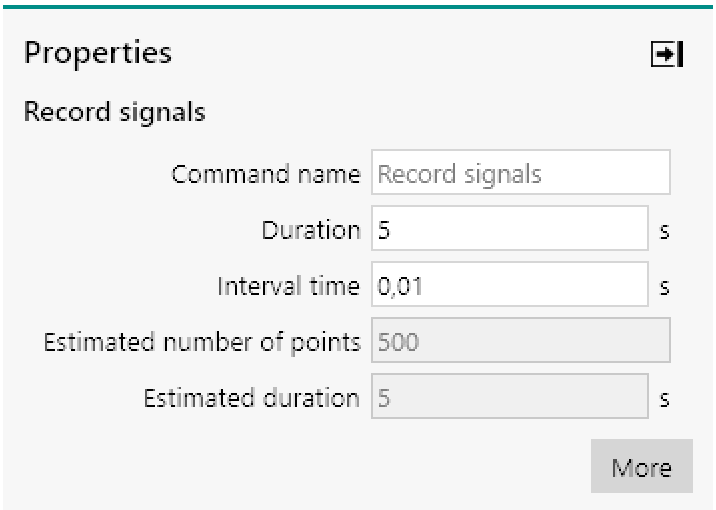


Figure 654 The measurement properties of the Record signals command

- **Record signals**
  - Duration: 5 s
  - Interval time: 0.01 s

The procedure samples the following signals *(see Figure 655, page 541)*:

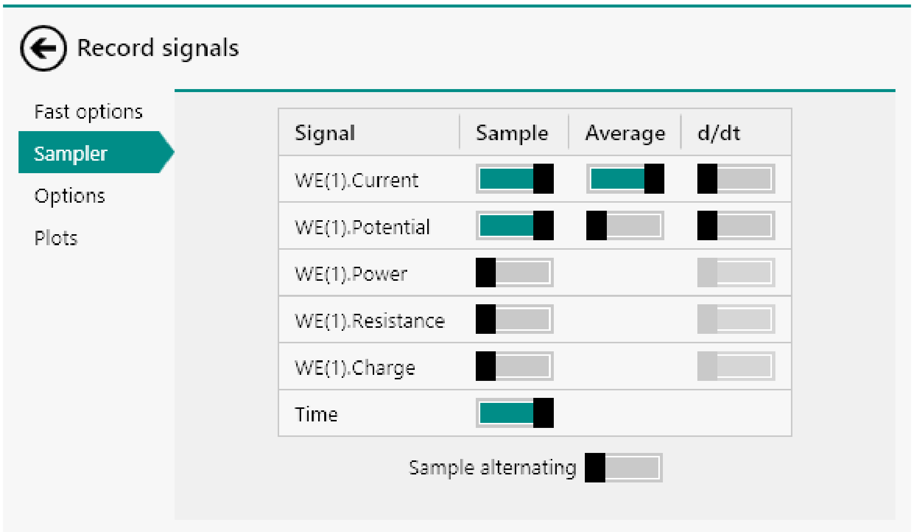


Figure 655 The sampler of the Record signals command

- WE(1).Current (averaged)
- WE(1).Potential
- Time

The procedure uses the following options *(see Figure 656, page 542)*:

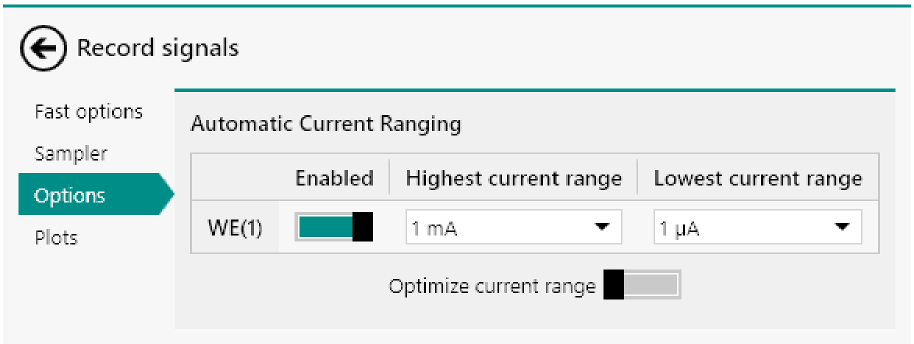


Figure 656 The options of the Record signals command

- Automatic current ranging
  - Highest current range: 1 mA
  - Lowest current range: 1 µA

The procedure plots the following data *(see Figure 657, page 542)*:

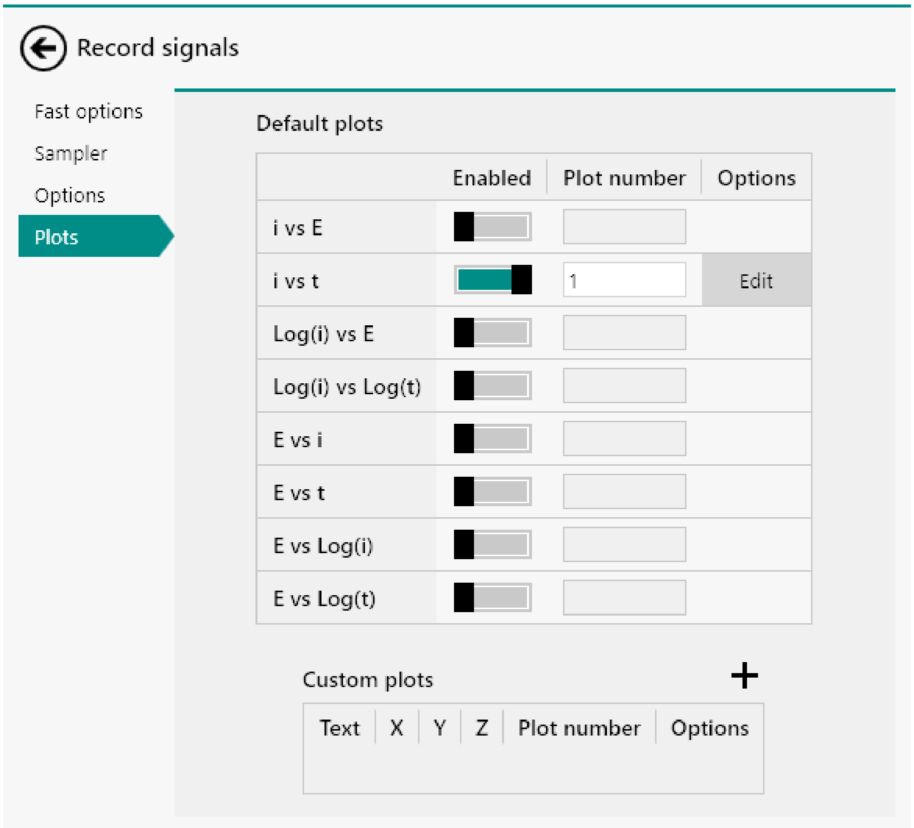


Figure 657 The plots of the Record signals command

- i vs t: WE(1).Current versus time

8.4.2 Chrono coulometry (Δt > 1 ms)

CAUTION

This procedure requires the optional **FI20** module or the **on-board integrator** *(see Chapter 16.3.2.11, page 1061)*.

The default **Chrono coulometry (Δt > 1 ms)** procedure provides an example of a typical chrono coulometric measurement using a sequence of potential steps *(see Figure 658, page 543)*.

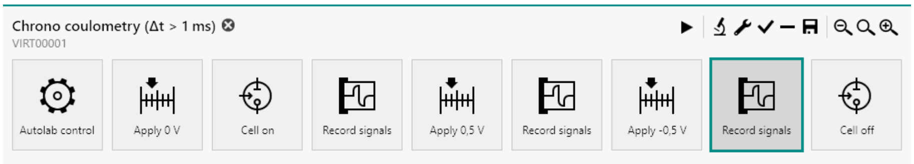


Figure 658 The default Chrono coulometry (Δt > 1 ms) procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure uses a sequence of three potential values (specified through the **Apply** command) followed by three **Record signals** commands.

NOTE

The smallest possible interval time for the **Record signals** command is 1.3 ms.

The potential values applied are 0 V, 0.5 V and -0.5 V. The **Record signals** commands have the following measurement properties *(see Figure 659, page 544)*:

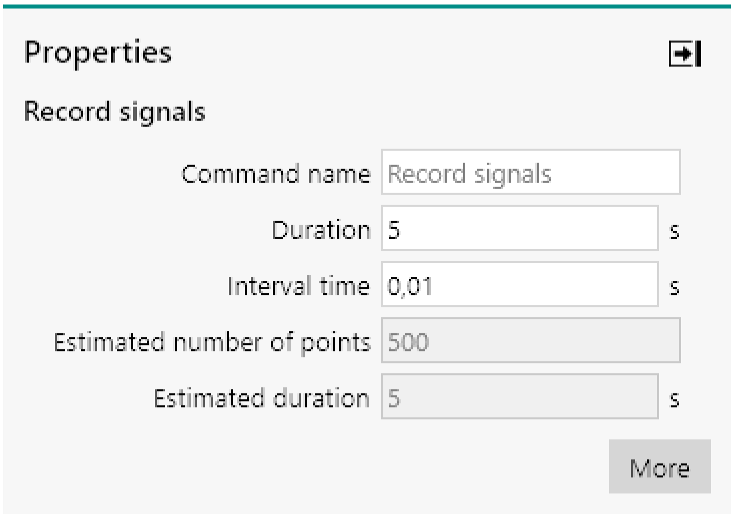


Figure 659 The measurement properties of the Record signals command

- **Record signals**
  - Duration: 5 s
  - Interval time: 0.01 s

The procedure samples the following signals *(see Figure 660, page 544)*:

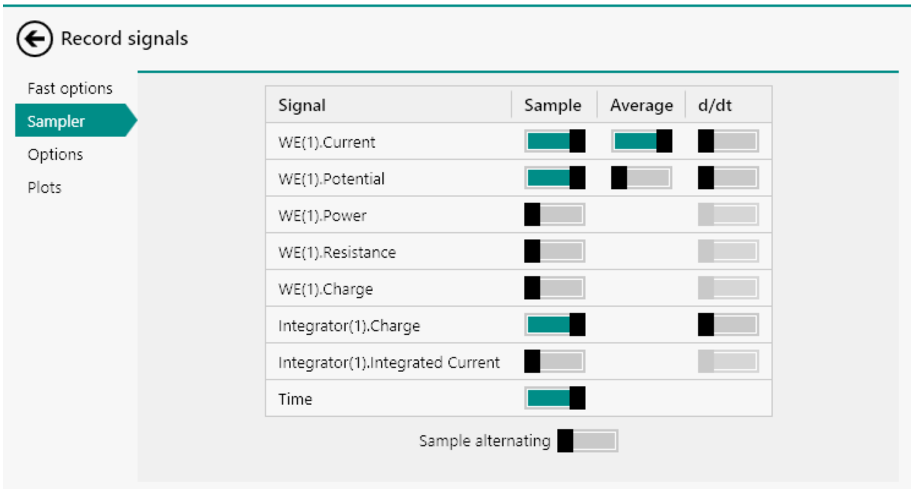


Figure 660 The sampler of the Record signals command

- WE(1).Current (averaged)
- WE(1).Potential
- Integrator(1).Charge
- Time

The procedure plots the following data *(see Figure 661, page 545)*:

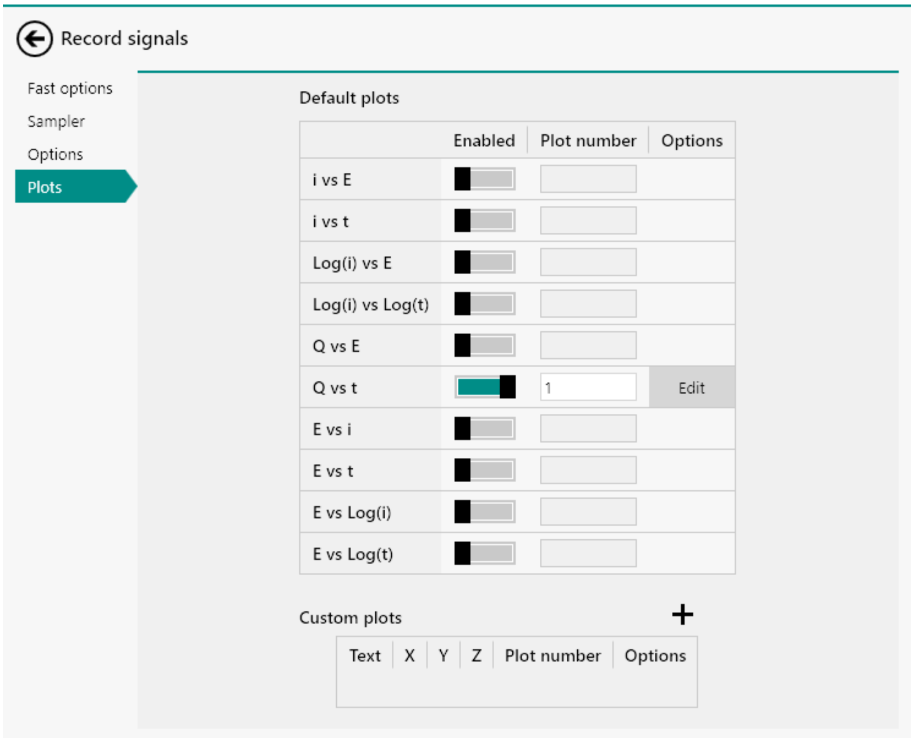


Figure 661 The plots of the Record signals command

- Q vs t: Integrator(1).Charge versus time

8.4.3 Chrono potentiometry (Δt > 1 ms)

The default **Chrono potentiometry (Δt > 1 ms)** procedure provides an example of a typical chrono potentiometric measurement using a sequence of current steps *(see Figure 662, page 545)*.

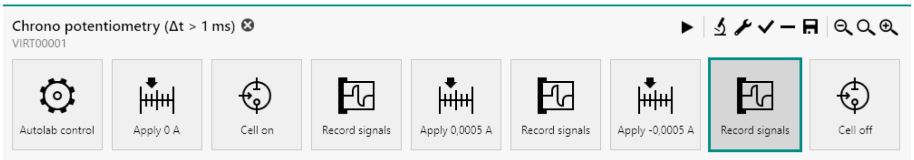


Figure 662 The default Chrono potentiometry (Δt > 1 ms) procedure

NOTE

The galvanostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure uses a sequence of three current values (specified through the **Apply** command) followed by three **Record signals** commands.

NOTE

The smallest possible interval time for the **Record signals** command is 1.3 ms.

The current values applied are 0 A, 500 µA and -500 µA. The **Record signals** commands have the following measurement properties *(see Figure 663, page 546)*:

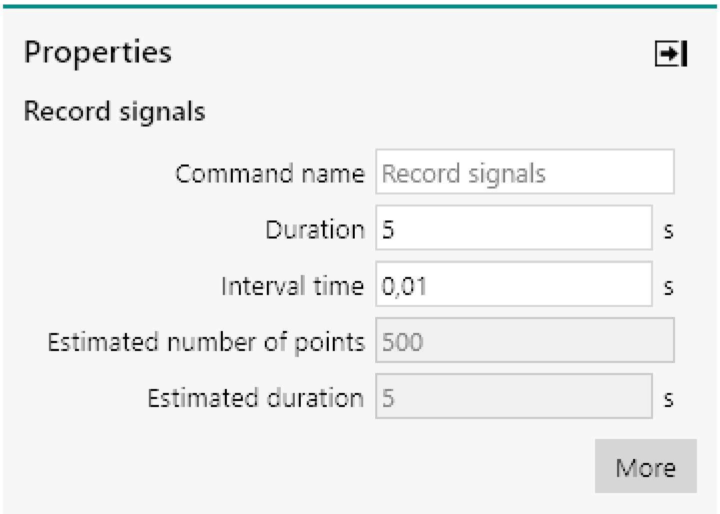


Figure 663 The measurement properties of the Record signals command

- **Record signals**
  - Duration: 5 s
  - Interval time: 0.01 s

The procedure samples the following signals *(see Figure 664, page 547)*:

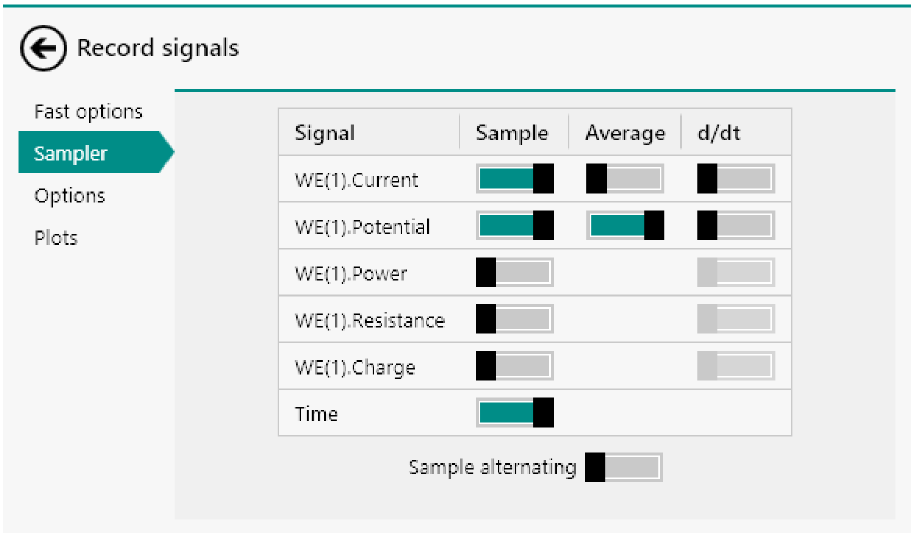


Figure 664 The sampler of the Record signals command

- WE(1).Potential (averaged)
- WE(1).Current
- Time

The procedure plots the following data *(see Figure 665, page 547)*:

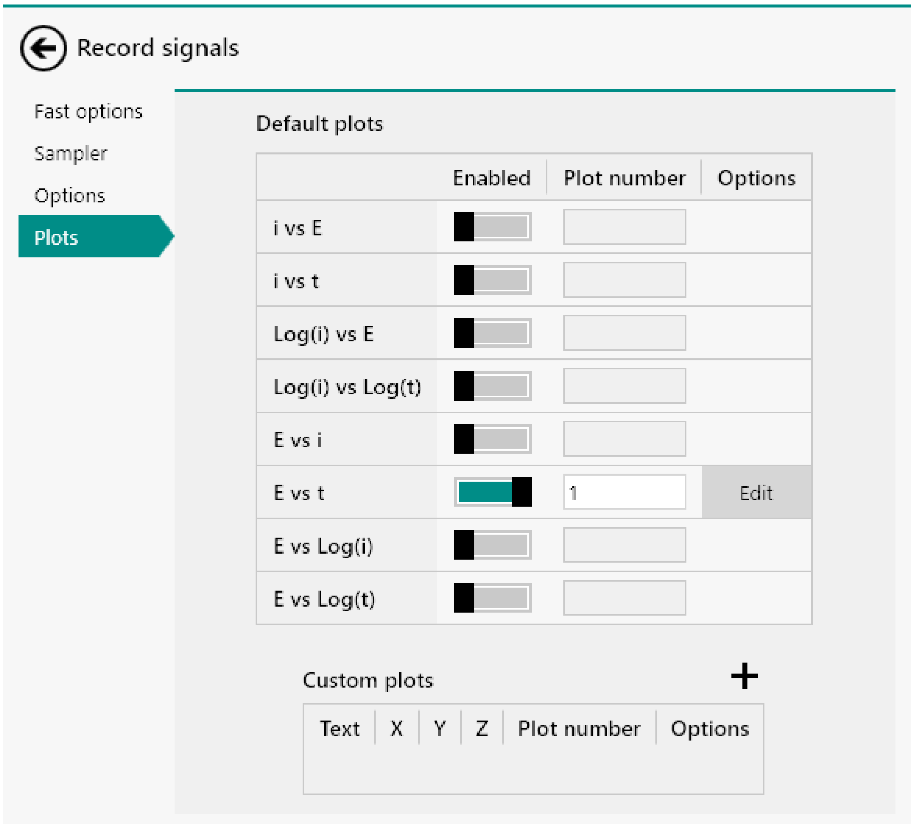


Figure 665 The plots of the Record signals command

- E vs t: WE(1)Potential versus time

8.4.4 Chrono amperometry fast

The default **Chrono amperometry fast** procedure provides an example of a typical chrono amperometric measurement using a sequence of potential steps, with a short interval time *(see Figure 666, page 548)*.

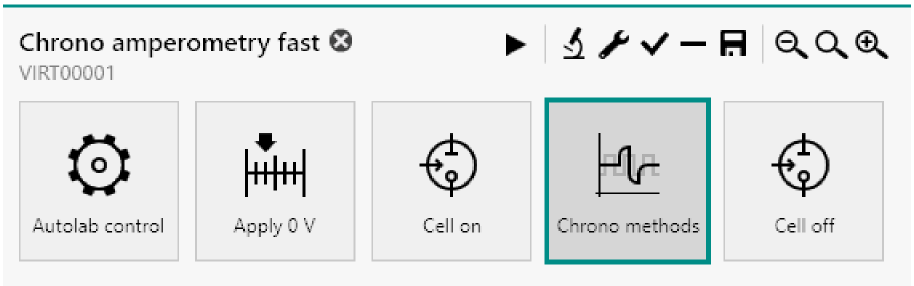


Figure 666 The default Chrono amperometry fast procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure uses a sequence of four potential values (specified in the **Chrono methods** command).

NOTE

The smallest possible interval time for the **Chrono methods** command is 100 µs.

The **Chrono methods** command has the following measurement properties *(see Figure 667, page 549)*:

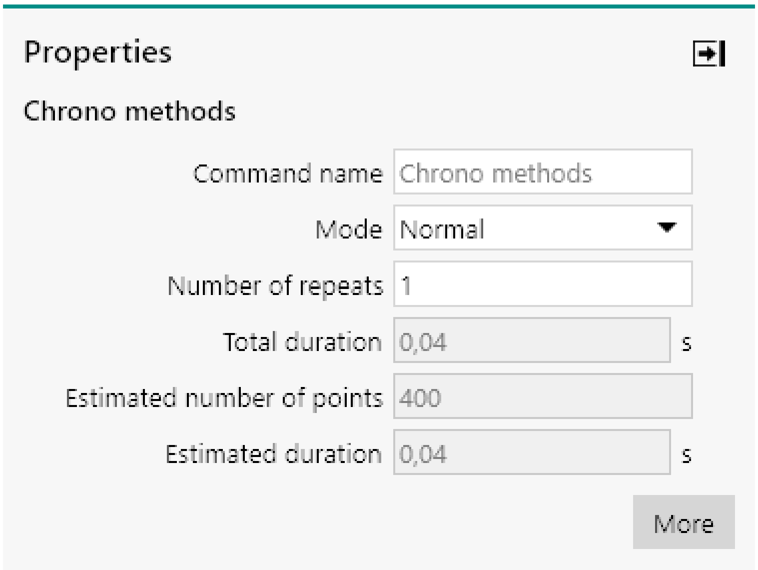


Figure 667 The measurement properties of the Chrono methods command

The procedure uses a sequence of four potential values applied and measured through the **Chrono methods** command *(see Figure 668, page 549)*.

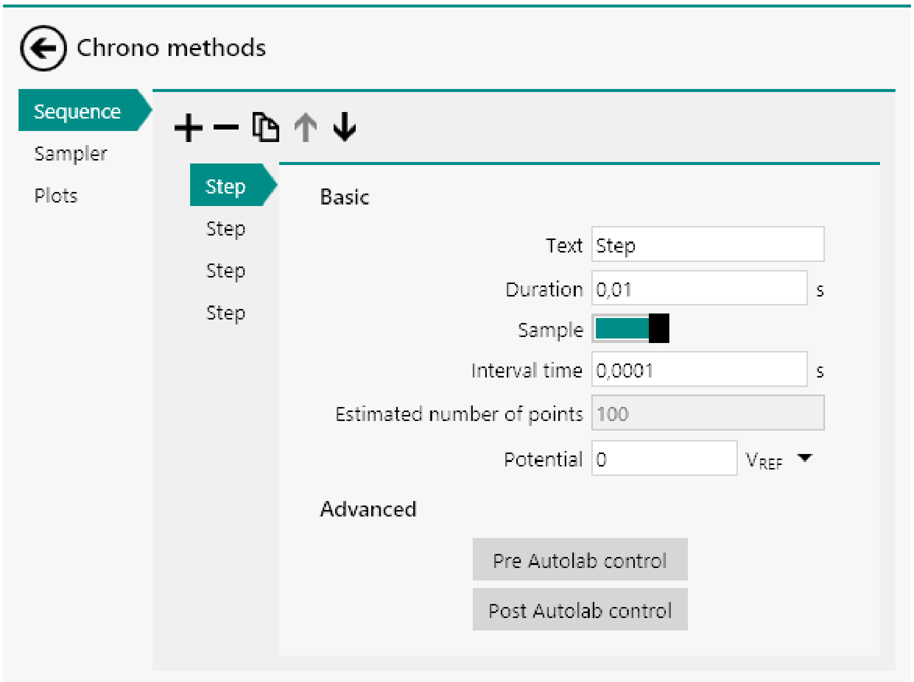


Figure 668 The details of the Chrono methods command

The potential values applied are 0 V, 0.3 V, -0.3 V and 0 V, applied versus the reference potential. The **Chrono methods** command has the following measurement properties *(see Figure 668, page 549)*:

- **Chrono methods**
  - Duration: 0.01 s
  - Interval time: 0.0001 s

The procedure samples the following signals *(see Figure 669, page 550)*:

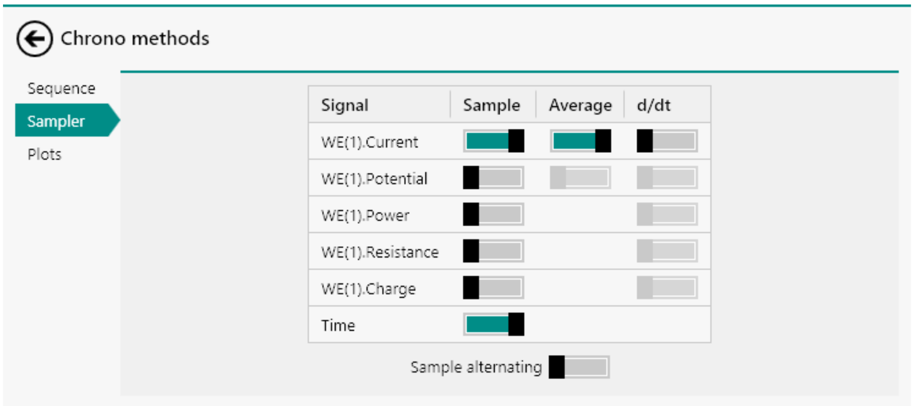


Figure 669 The sampler of the Chrono methods command

- WE(1).Current (averaged)
- Time

The procedure plots the following data *(see Figure 670, page 550)*:

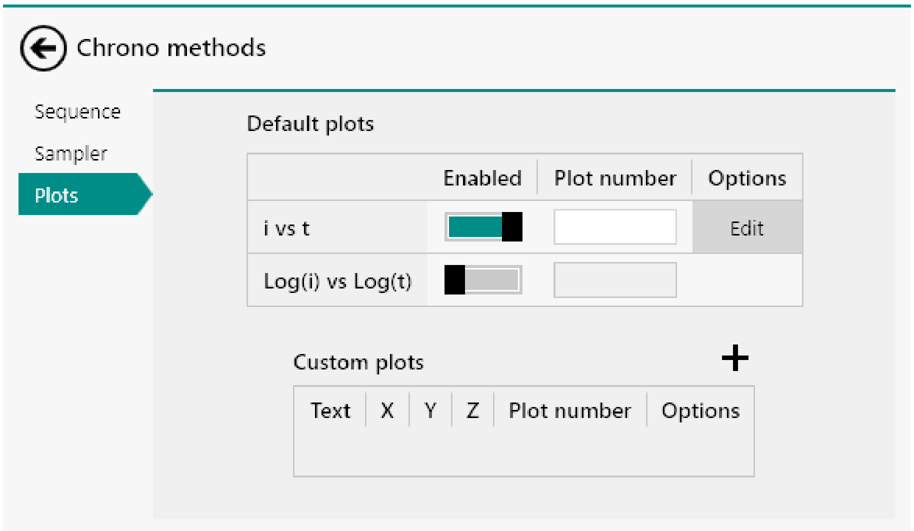


Figure 670 The plots of the Chrono methods command

- i vs t: WE(1).Current versus time

8.4.5 Chrono coulometry fast

CAUTION

This procedure requires the optional **FI20** module or the **on-board integrator** *(see Chapter 16.3.2.11, page 1061)*.

The default **Chrono coulometry fast** procedure provides an example of a typical chrono coulometric measurement using a sequence of potential steps, with a short interval time *(see Figure 671, page 551)*.

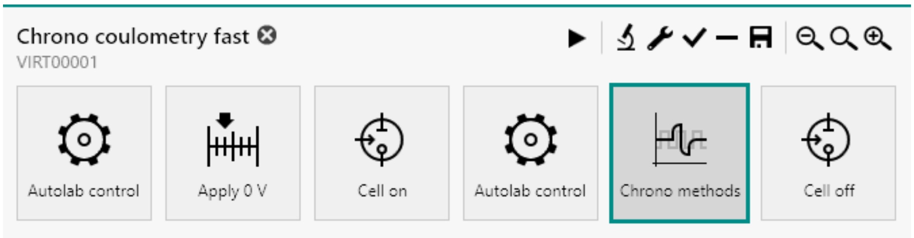


Figure 671 The default Chrono coulometry fast procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure uses a sequence of two potential values (specified in the **Chrono methods** command).

NOTE

The smallest possible interval time for the **Chrono methods** command is 100 µs.

The **Chrono methods** command has the following measurement properties *(see Figure 672, page 552)*:

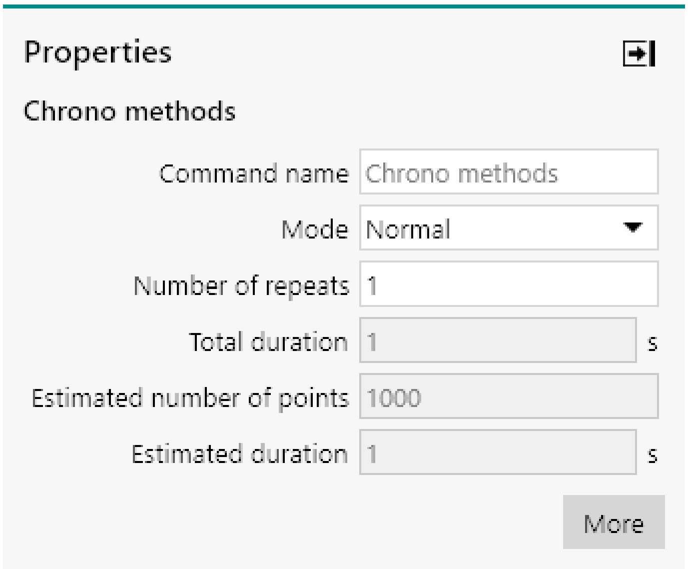


Figure 672 The measurement properties of the Chrono methods command

The procedure uses a sequence of two potential values applied and measured through the **Chrono methods** command *(see Figure 673, page 552)*.

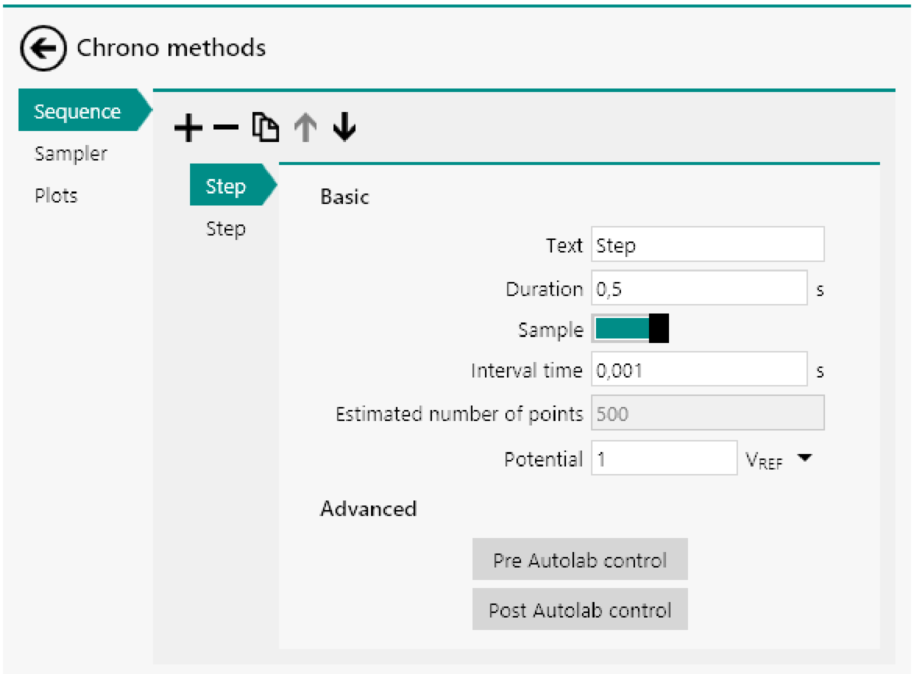


Figure 673 The details of the Chrono methods command

The potential values applied are 1 V and -1 V, applied versus the reference potential. The **Chrono methods** command has the following measurement properties *(see Figure 673, page 552)*:

- **Chrono methods**
  - Duration: 0.5 s
  - Interval time: 0.001 s

The procedure samples the following signals *(see Figure 674, page 553)*:

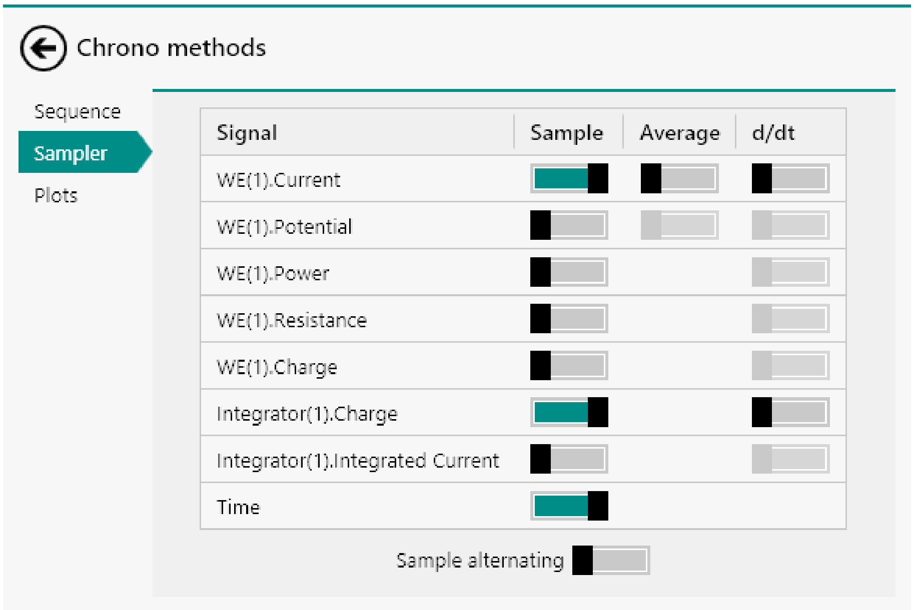


Figure 674 The sampler of the Chrono methods command

- WE(1).Current
- Integrator(1).Charge
- Time

The procedure plots the following data *(see Figure 675, page 553)*:

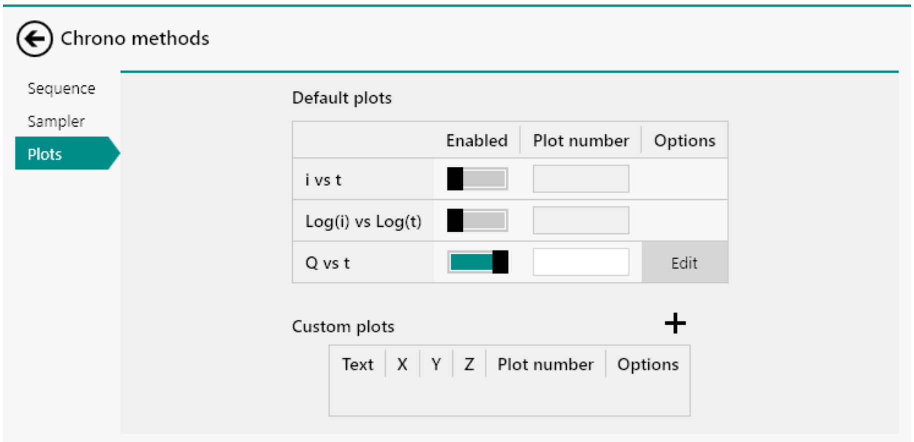


Figure 675 The plots of the Chrono methods command

- Q vs t: Integrator(1).Charge versus time

8.4.6 Chrono potentiometry fast

The default **Chrono potentiometry fast** procedure provides an example of a typical chrono potentiometric measurement using a sequence of current steps, with a short interval time *(see Figure 676, page 554)*.

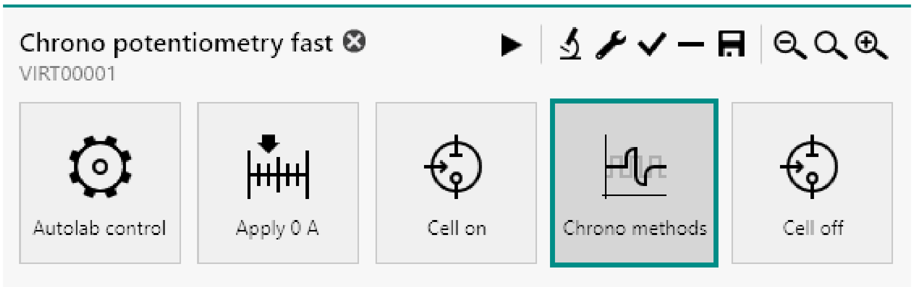


Figure 676 The default Chrono potentiometry fast procedure

NOTE

The galvanostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure uses a sequence of four current values (specified in the **Chrono methods** command).

NOTE

The smallest possible interval time for the **Chrono methods** command is 100 µs.

The **Chrono methods** command has the following measurement properties *(see Figure 677, page 555)*:

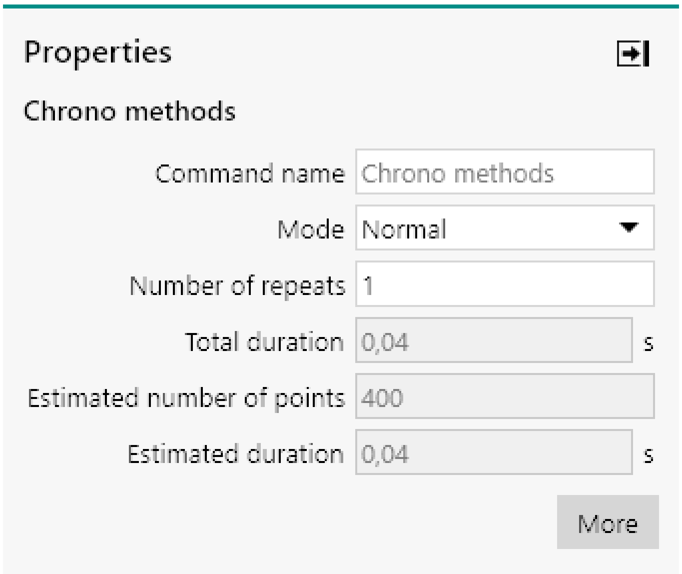


Figure 677 The measurement properties of the Chrono methods command

The procedure uses a sequence of four current values applied and measured through the **Chrono methods** command *(see Figure 678, page 555)*.

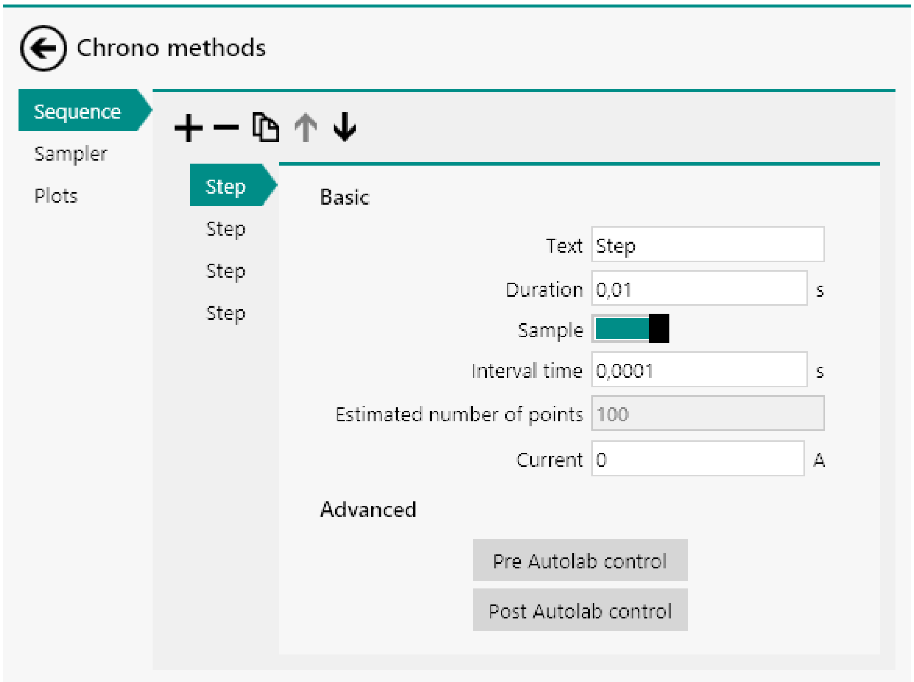


Figure 678 The details of the Chrono methods command

The current values applied are 0 A, 0.003 A, -0.003 A and 0 A. The **Chrono methods** command has the following measurement properties *(see Figure 678, page 555)*:

- **Chrono methods**
  - Duration: 0.01 s
  - Interval time: 0.0001 s

The procedure samples the following signals *(see Figure 679, page 556)*:

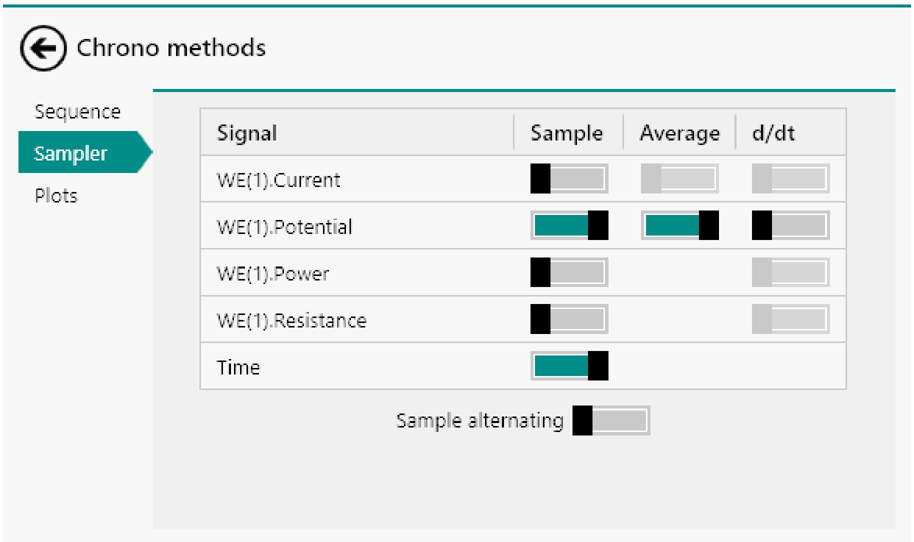


Figure 679 The sampler of the Chrono methods command

- WE(1).Potential (averaged)
- Time

The procedure plots the following data *(see Figure 680, page 556)*:

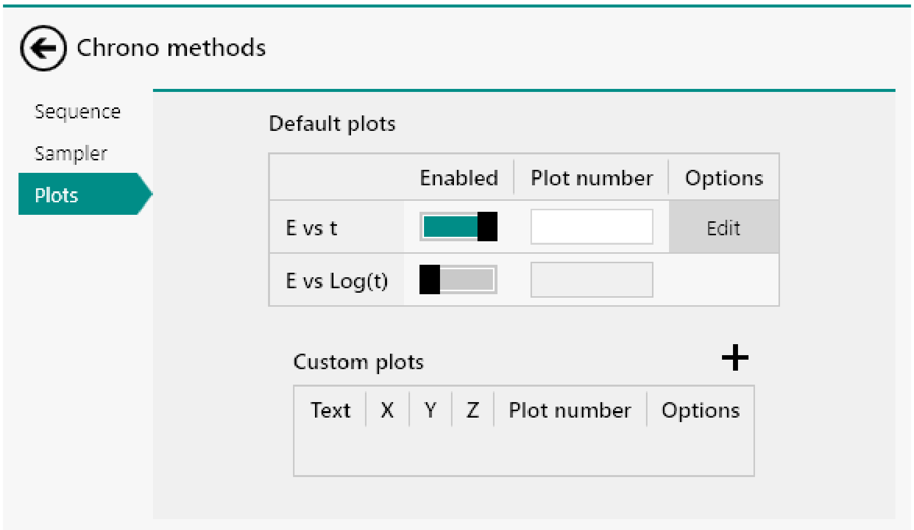


Figure 680 The plots of the Chrono methods command

- E vs t: WE(1).Potential versus time

8.4.7 Chrono amperometry high speed

CAUTION

This procedure requires the optional **ADC10M** or **ADC750** module *(see Chapter 16.3.2.1, page 977)*.

The default **Chrono amperometry high speed** procedure provides an example of a typical chrono amperometric measurement using a sequence of potential steps, with a short interval time *(see Figure 681, page 557)*.

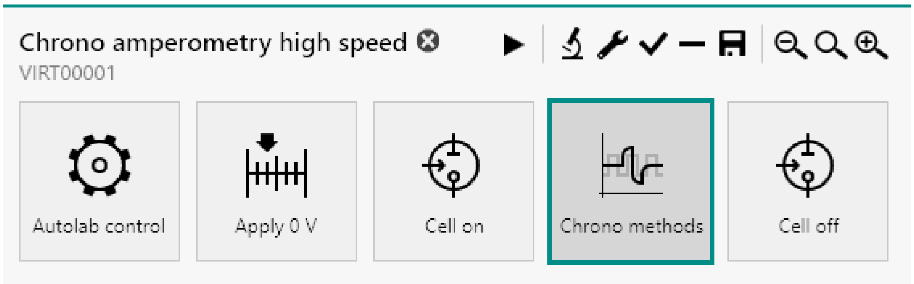


Figure 681 The default Chrono amperometry high speed procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure uses a sequence of four potential values (specified in the **Chrono methods** command).

NOTE

The smallest possible interval time for the **Chrono methods** command in *high speed* mode is 100 ns with the **ADC10M** and 1.33 µs with the **ADC750**.

The **Chrono methods** command has the following measurement properties *(see Figure 682, page 558)*:

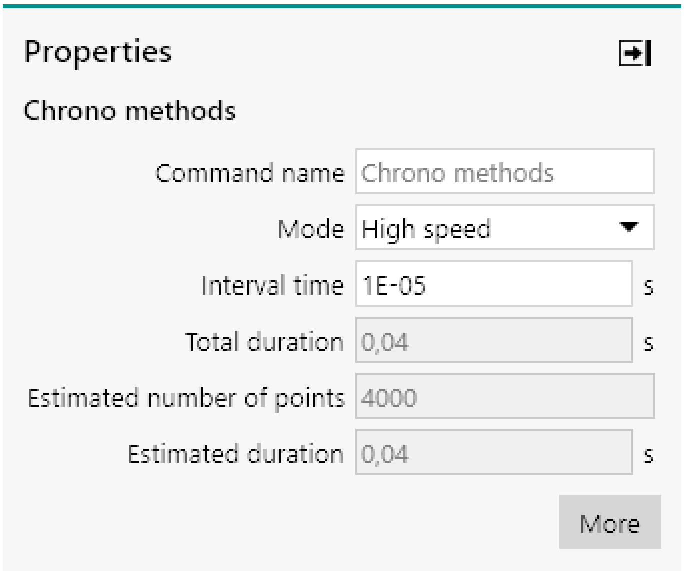


Figure 682 The measurement properties of the Chrono methods command

- **Interval time:** the interval time is set to 10 µs for all steps.

The procedure uses a sequence of four potential values applied and measured through the **Chrono methods** command *(see Figure 683, page 558)*.

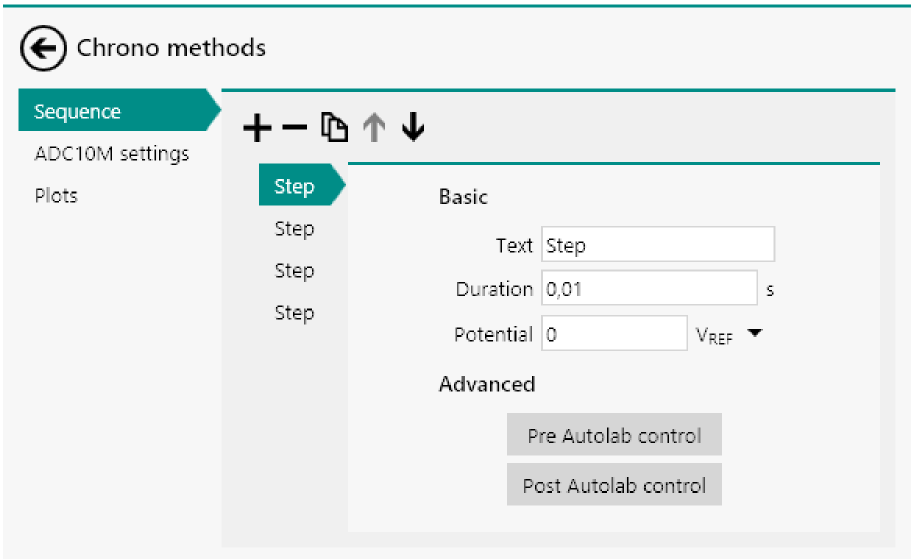


Figure 683 The details of the Chrono methods command

The potential values applied are 0 V, 0.3 V, -0.3 V and 0 V, applied versus the reference potential. The **Chrono methods** command has the following measurement properties *(see Figure 683, page 558)*:

- **Chrono methods**
  - Duration: 0.01 s

The procedure samples the following **ADC10M** or **ADC750** settings *(see Figure 684, page 559)*:

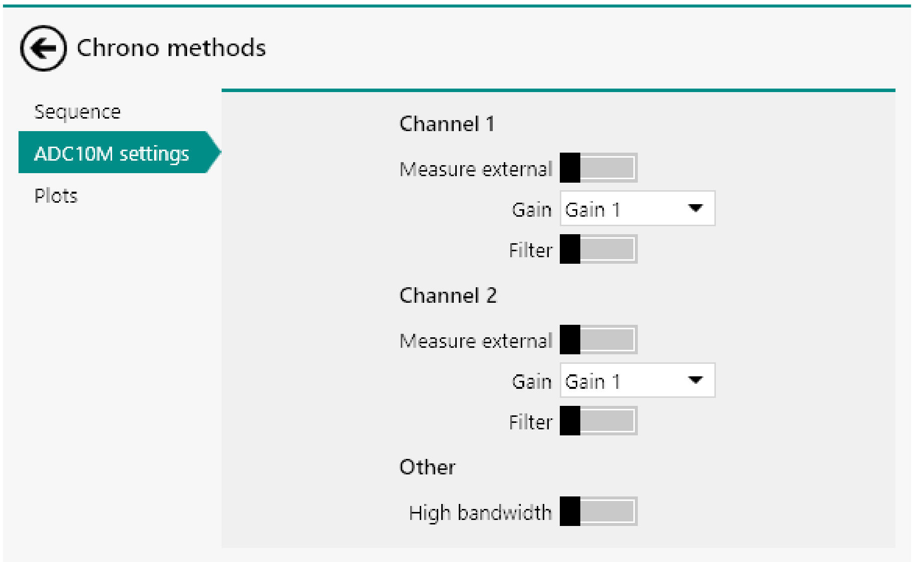


Figure 684 The ADC10M or ADC750 settings of the Chrono methods command

- Channel 1: WE(1).Potential, Gain 1, unfiltered
- Channel 2: WE(1).Current, Gain 1, unfiltered

The procedure plots the following data *(see Figure 685, page 560)*:

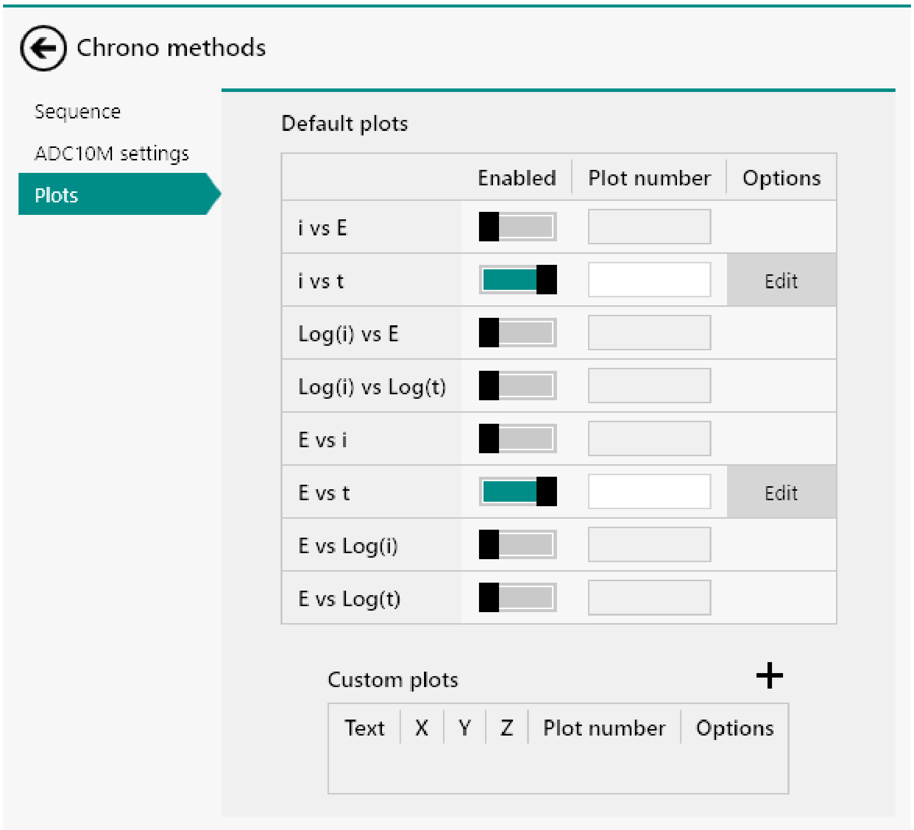


Figure 685 The plots of the Chrono methods command

- i vs t: WE(1).Current versus time
- E vs t: WE(1).Potential versus time

8.4.8 Chrono potentiometry high speed

CAUTION

This procedure requires the optional **ADC10M** or **ADC750** module *(see Chapter 16.3.2.1, page 977)*.

The default **Chrono potentiometry high speed** procedure provides an example of a typical chrono potentiometric measurement using a sequence of current steps, with a short interval time *(see Figure 686, page 561)*.

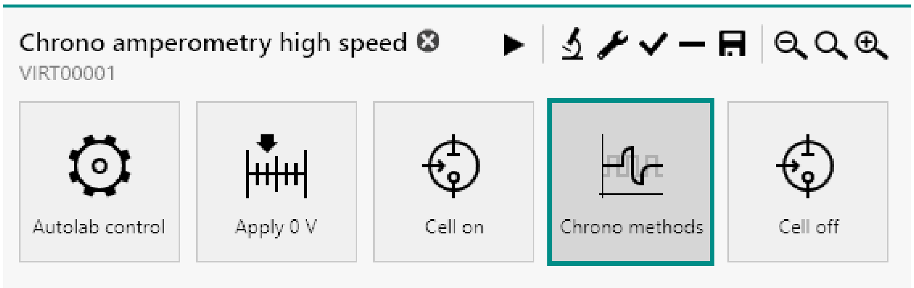


Figure 686 The default Chrono potentiometry high speed procedure

NOTE

The galvanostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure uses a sequence of four current values (specified in the **Chrono methods** command).

NOTE

The smallest possible interval time for the **Chrono methods** command in *high speed* mode is 100 ns with the **ADC10M** and 1.33 µs with the **ADC750**.

The **Chrono methods** command has the following measurement properties *(see Figure 687, page 561)*:

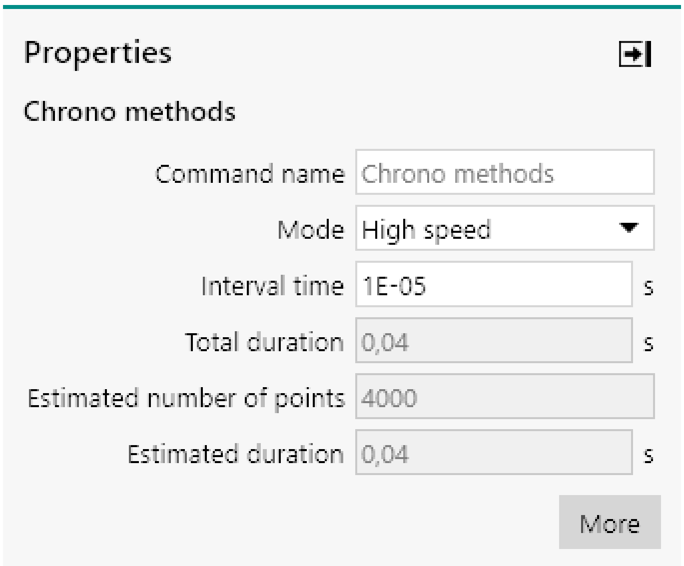


Figure 687 The measurement properties of the Chrono methods command

- **Interval time:** the interval time is set to 10 µs for all steps.

The procedure uses a sequence of four current values applied and measured through the **Chrono methods** command *(see Figure 688, page 562)*.

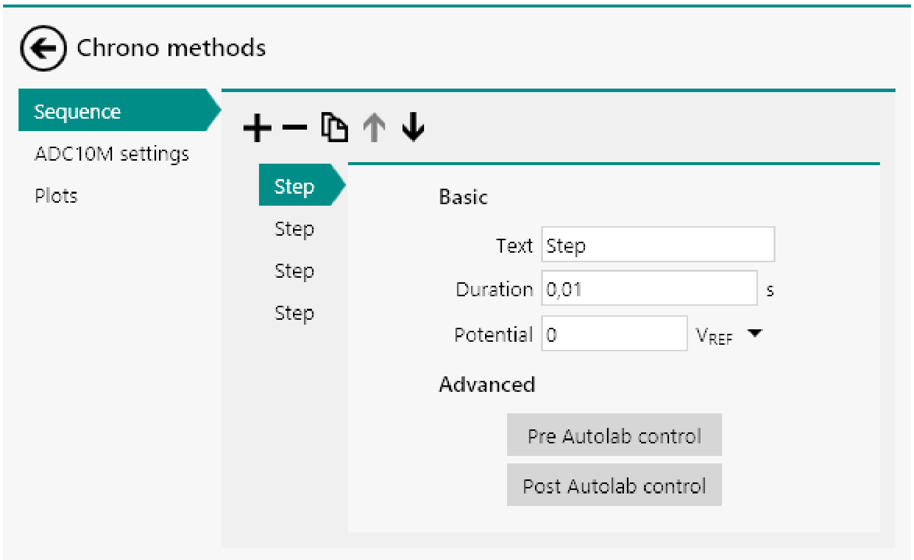


Figure 688 The details of the Chrono methods command

The current values applied are 0 A, 0.003 A, -0.003 A and 0 A. The **Chrono methods** command has the following measurement properties *(see Figure 688, page 562)*:

- **Chrono methods**
  - Duration: 0.01 s

The procedure samples the following **ADC10M** or **ADC750** settings *(see Figure 689, page 563)*:

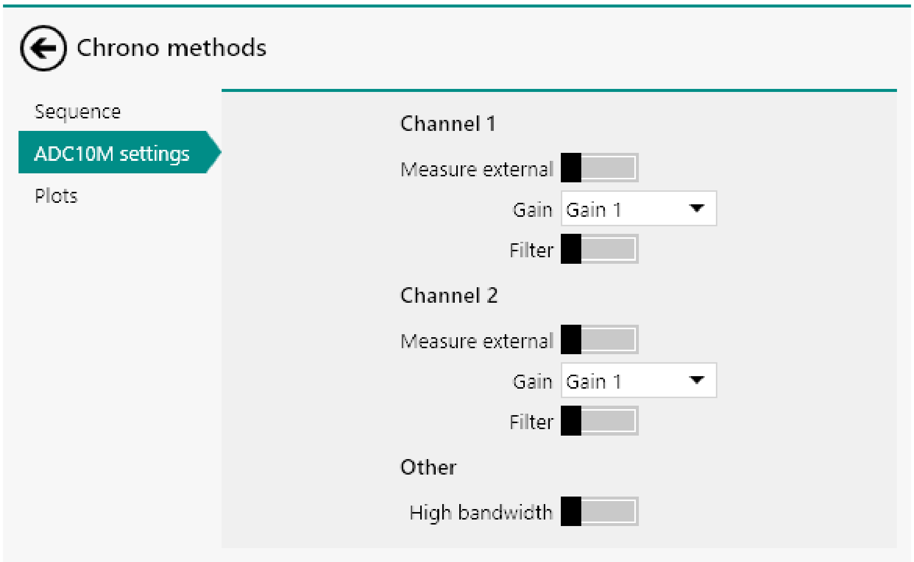


Figure 689 The ADC10M or ADC750 settings of the Chrono methods command

- Channel 1: WE(1).Potential, Gain 1, unfiltered
- Channel 2: WE(1).Current, Gain 1, unfiltered

The procedure plots the following data *(see Figure 690, page 563)*:

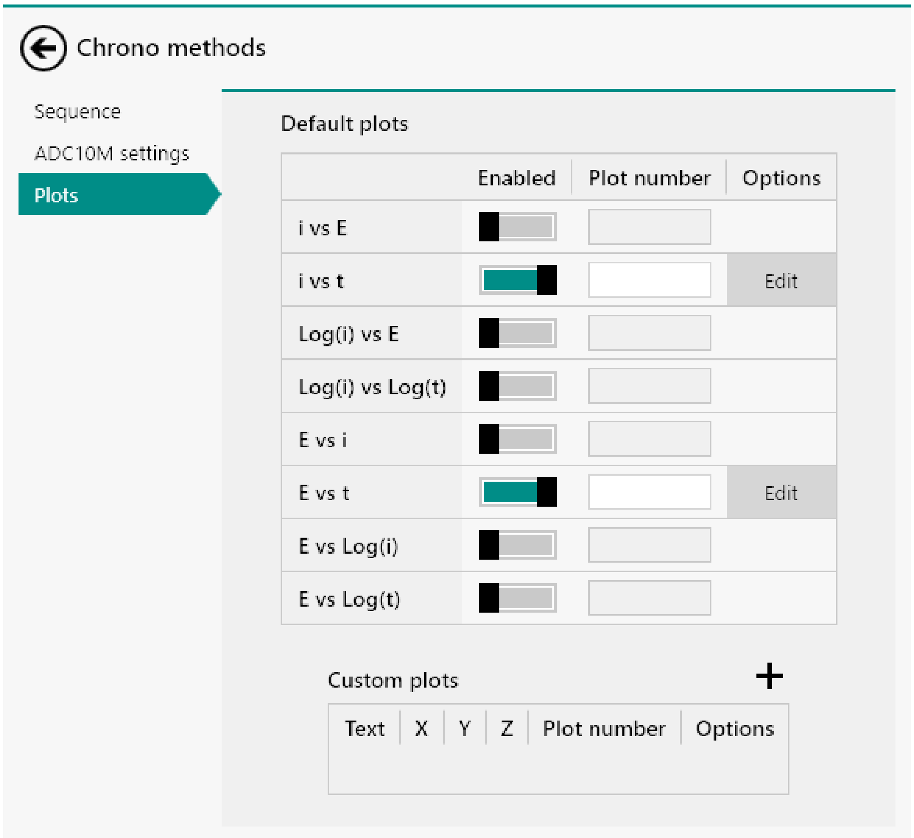


Figure 690 The plots of the Chrono methods command

- i vs t: WE(1).Current versus time
- E vs t: WE(1).Potential versus time

8.4.9 Chrono charge discharge

The default **Chrono charge discharge** procedure provides an example of a typical charge and discharge measurement using a sequence of potential steps *(see Figure 691, page 564)*.

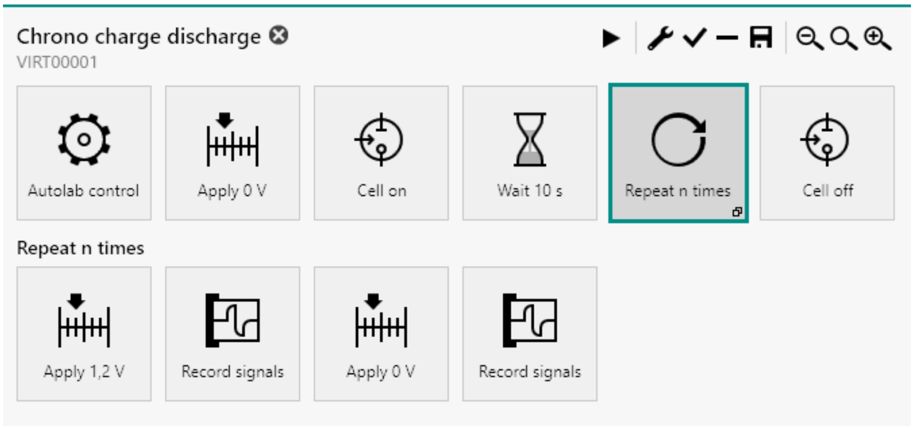


Figure 691 The default Chrono charge discharge procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure uses a **Repeat** command, used in the *Repeat n times* mode, containing a sequence of two potential values (specified through the **Apply** command) followed by two **Record signals** commands.

NOTE

The smallest possible interval time for the **Record signals** command is 1.3 ms.

The potential values applied are 1.2 V and -0.5 V. The **Record signals** commands have the following measurement properties *(see Figure 692, page 565)*:

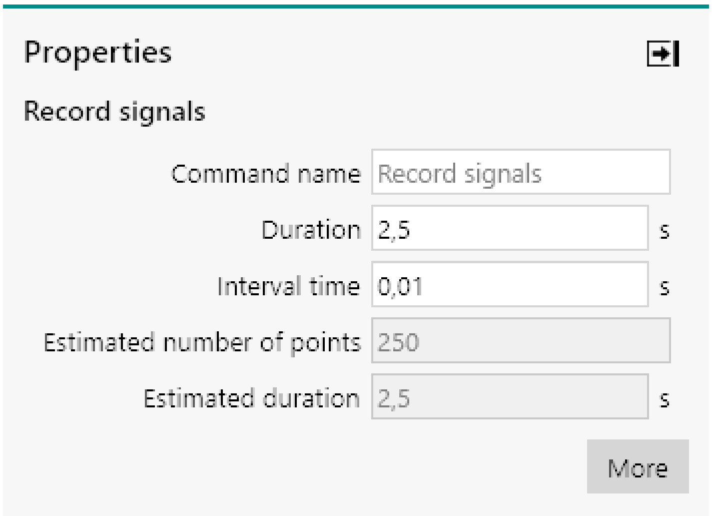


Figure 692 The measurement properties of the Record signals command

- **Record signals**
  - Duration: 2.5 s
  - Interval time: 0.01 s

The procedure samples the following signals *(see Figure 693, page 565)*:

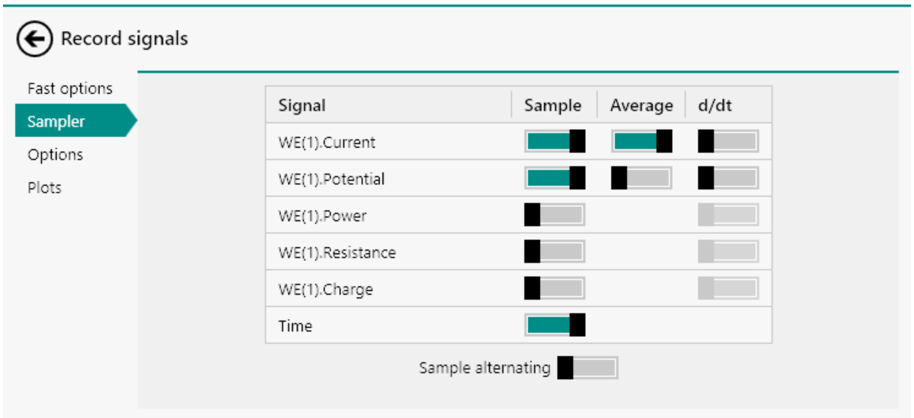


Figure 693 The sampler of the Record signals command

- WE(1).Current (averaged)
- WE(1).Potential
- Time

The procedure uses the following options *(see Figure 694, page 566)*:

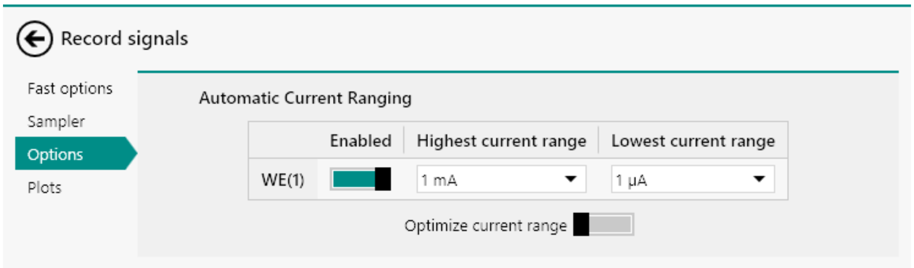


Figure 694 The options of the Record signals command

- Automatic current ranging
  - Highest current range: 1 mA
  - Lowest current range: 1 µA

The procedure plots the following data *(see Figure 695, page 566)*:

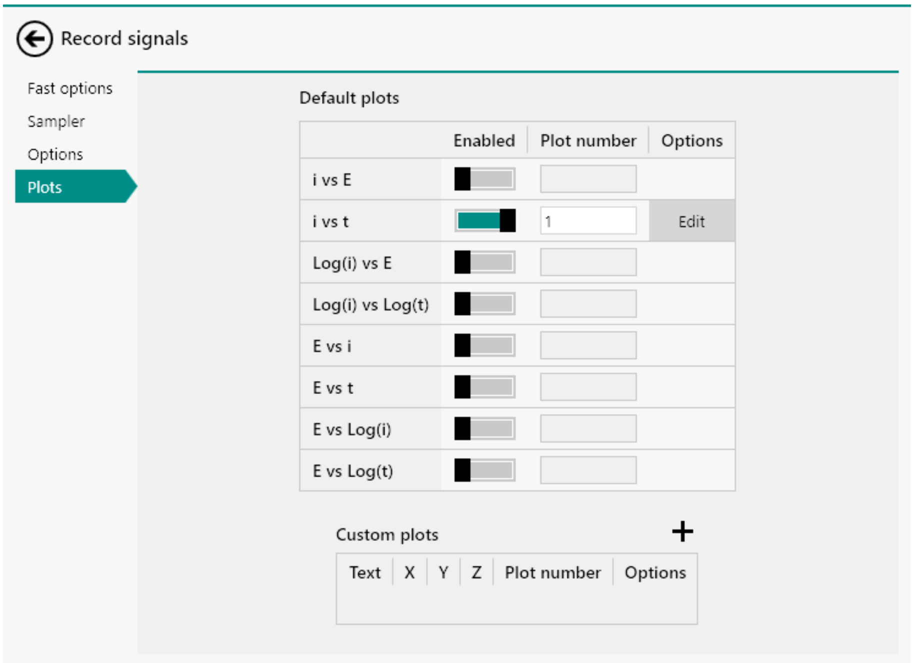


Figure 695 The plots of the Record signals command

- i vs t: WE(1).Current versus time

8.5 Potentiometric stripping analysis

NOVA provides two default procedures for potentiometric stripping analysis (PSA).

The following procedures are available:

- Potentiometric stripping analysis
- Potentiometric stripping analysis (Constant current)

8.5.1 Potentiometric stripping analysis

The default **Potentiometric stripping analysis** procedure provides an example of a typical measurement using the **PSA** command *(see Figure 696, page 567)*.

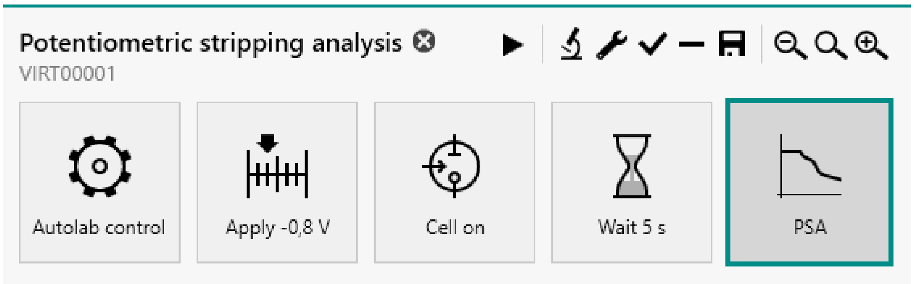


Figure 696 The default Potentiometric stripping analysis procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure has the following measurement properties, specified for the **PSA** command *(see Figure 697, page 568)*:

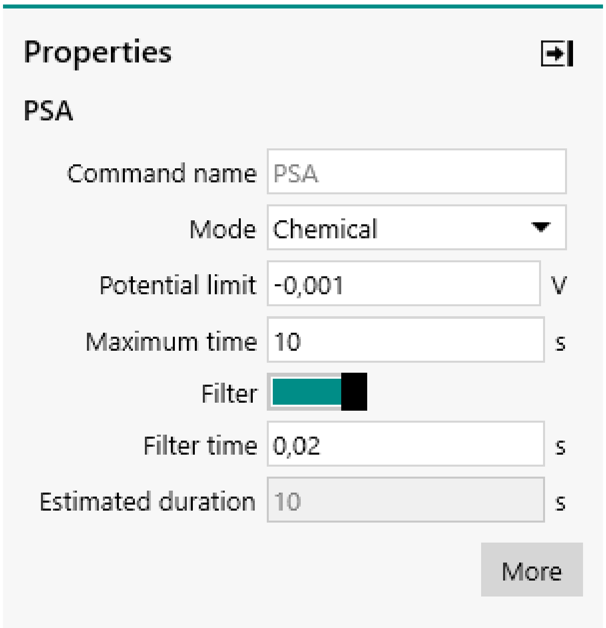


Figure 697 The measurement properties of the PSA command

- **PSA**
  - Potential limit: -0.001 V, versus reference electrode
  - Maximum time: 10 s
  - Filter: on
  - Filter time: 0.020 s or 0.0166 s

The procedure plots the following data *(see Figure 698, page 568)*:

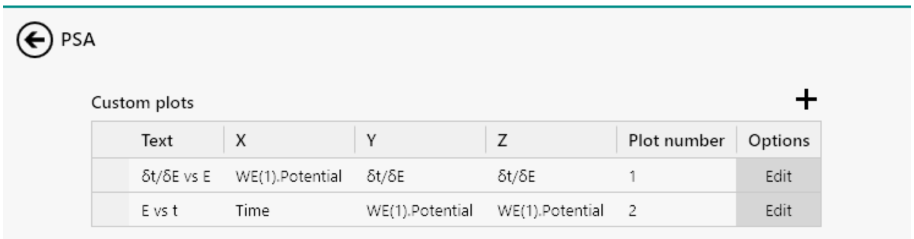


Figure 698 The plots of the PSA command

- δt/δE vs E: δt/δWE(1).Potential versus WE(1).Potential
- E vs t: WE(1).Potential versus time

8.5.2 Potentiometric stripping analysis constant current

The default **Potentiometric stripping analysis** procedure provides an example of a typical measurement using the **PSA constant current** command *(see Figure 699, page 569)*.

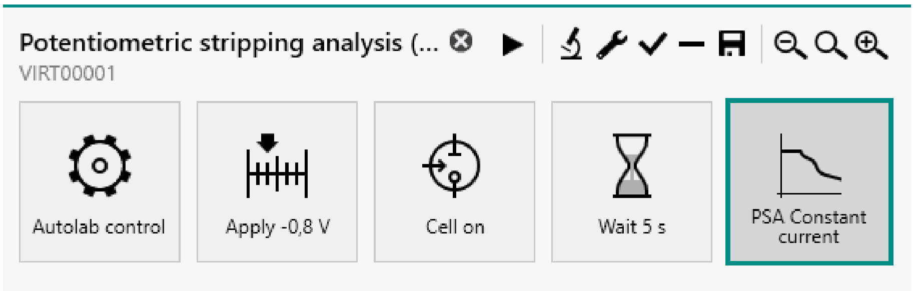


Figure 699 The default Potentiometric stripping analysis (Constant current) procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure has the following measurement properties, specified for the **PSA constant current** command *(see Figure 700, page 569)*:

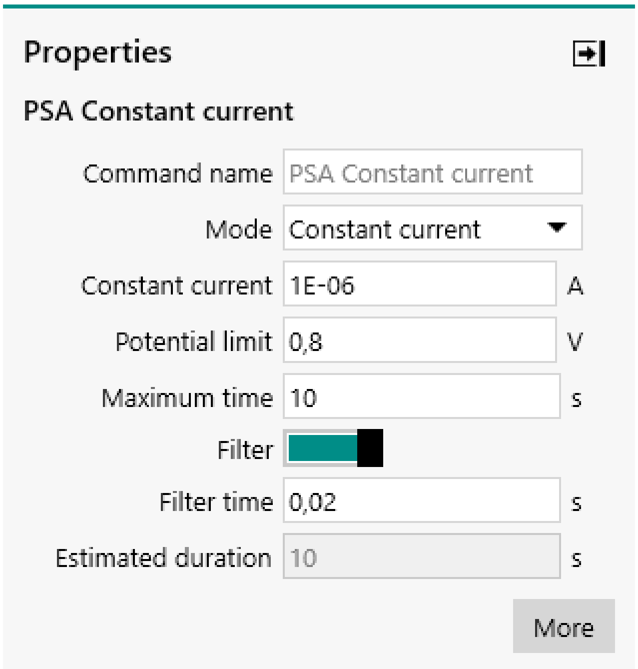


Figure 700 The measurement properties of the PSA constant current command

- **PSA**
  - Constant current: 1 µA
  - Potential limit: 0.8 V, versus reference electrode
  - Maximum time: 10 s
  - Filter: on
  - Filter time: 0.020 s or 0.0166 s

The procedure plots the following data *(see Figure 701, page 570)*:

Figure 701 The plots of the PSA constant current command

- δt/δE vs E: δt/δWE(1).Potential versus WE(1).Potential
- E vs t: WE(1).Potential versus time

8.6 Impedance spectroscopy

NOVA provides six default procedures for impedance spectroscopy. These procedures can be used to perform a cyclic potential or current scan and record the response of the cell.

CAUTION

These procedures require the optional **FRA32M** or **FRA2** module *(see Chapter 16.3.2.13, page 1091)*.

The following procedures are available:

- FRA impedance potentiostatic
- FRA impedance galvanostatic
- FRA potential scan
- FRA current scan
- FRA time scan potentiostatic
- FRA time scan galvanostatic

Additionally, a default procedure for Electrochemical Frequency Modulation measurement is also included in this group.

8.6.1 FRA impedance potentiostatic

The default FRA impedance potentiostatic procedure provides an example of an electrochemical impedance spectroscopy measurement in potentiostatic conditions *(see Figure 702, page 571)*.

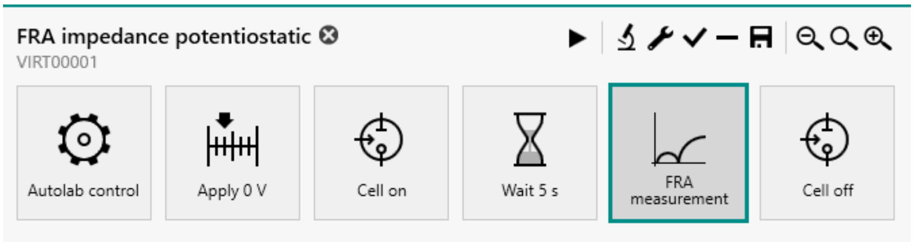


Figure 702 The default FRA impedance potentiostatic procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure has the following measurement properties, specified for the **FRA measurement** command *(see Figure 703, page 571)*:

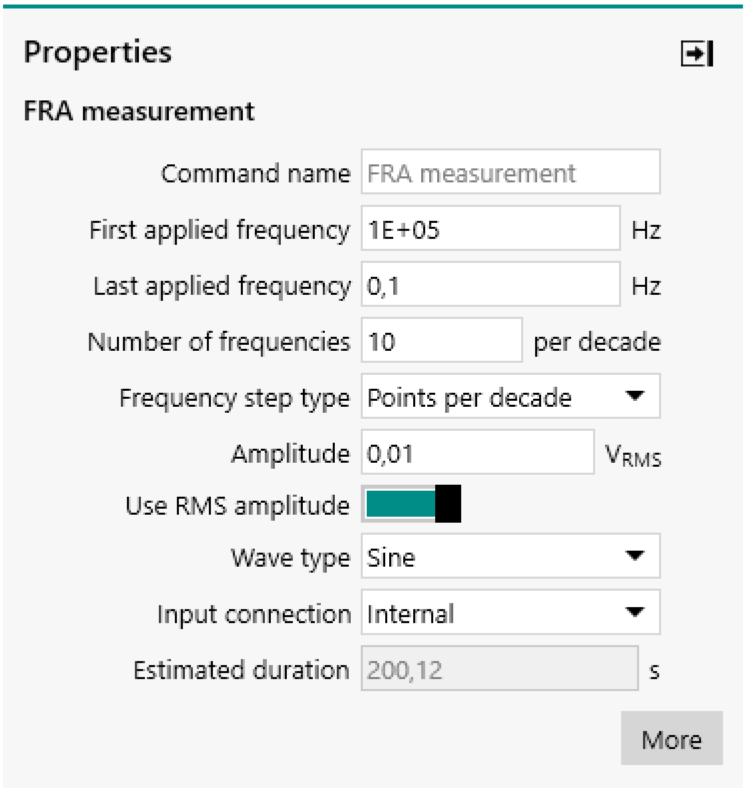


Figure 703 The measurement properties of the FRA measurement command

- **FRA measurement**
  - Start frequency: 100 kHz
  - Stop frequency: 0.1 Hz
  - Number of frequencies per decade: 10
  - Amplitude: 0.010 V
  - Use RMS amplitude: yes
  - Wave type: sine
  - Input connection: internal

The procedure samples the following signals *(see Figure 704, page 572)*:

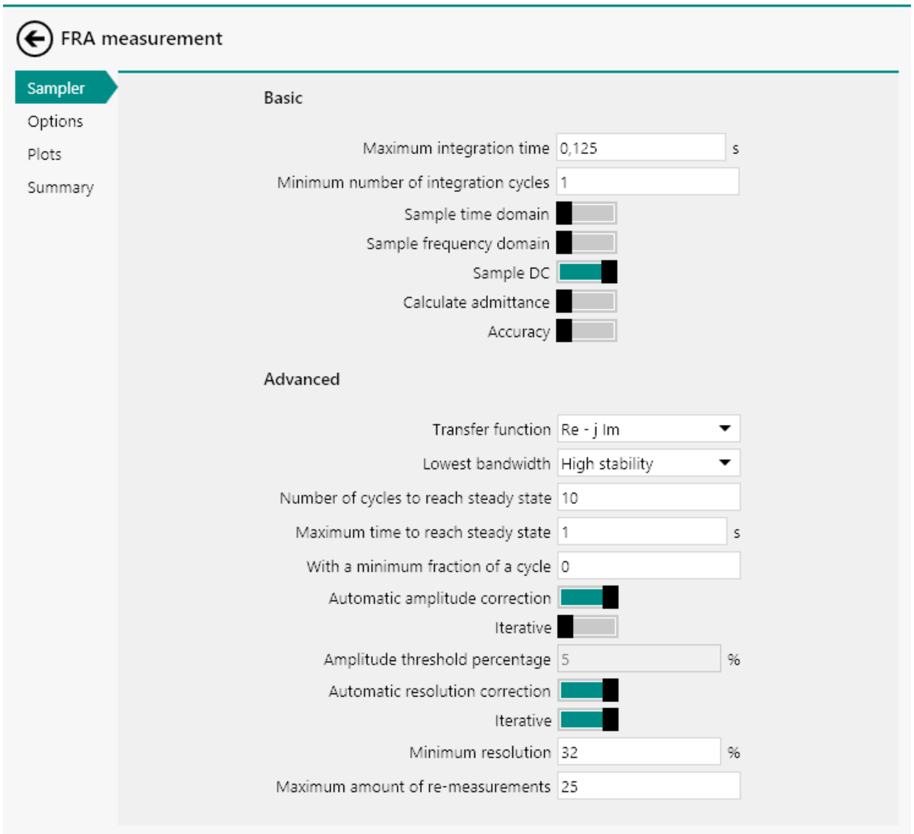


Figure 704 The sampler of the FRA measurement command

- DC signals

The procedure uses the following options *(see Figure 705, page 572)*:

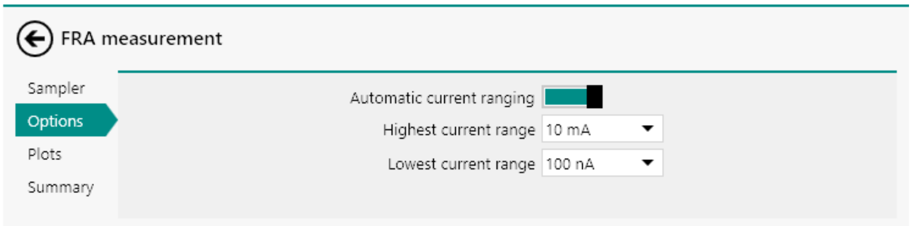


Figure 705 The options of the FRA measurement command

- Automatic current ranging
  - Highest current range: 10 mA
  - Lowest current range: 100 nA

The procedure plots the following data *(see Figure 706, page 573)*:

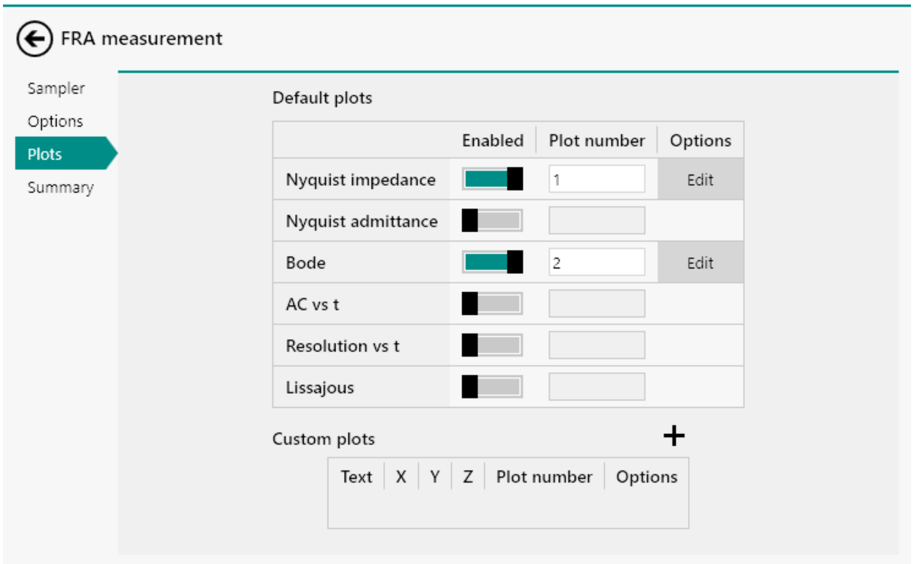


Figure 706 The plots of the FRA measurement command

- Nyquist impedance
- Bode

The Potential scan FRA data command located at the end of the procedure will also generate Mott-Schottky plots automatically. Please refer to for more information on the **Potential scan FRA data** command.

8.6.2 FRA impedance galvanostatic

The default FRA impedance galvanostatic procedure provides an example of an electrochemical impedance spectroscopy measurement in galvanostatic conditions *(see Figure 707, page 573)*.

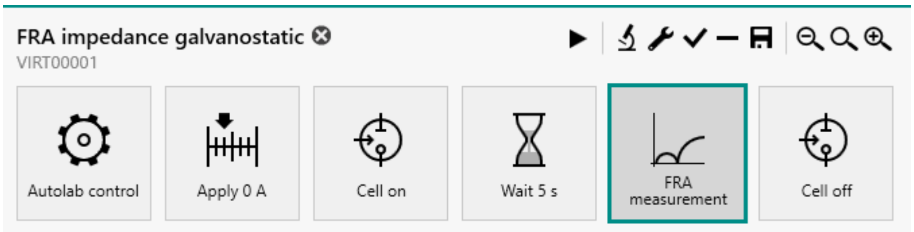


Figure 707 The default FRA impedance galvanostatic procedure

NOTE

The galvanostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure has the following measurement properties, specified for the **FRA measurement** command *(see Figure 708, page 574)*:

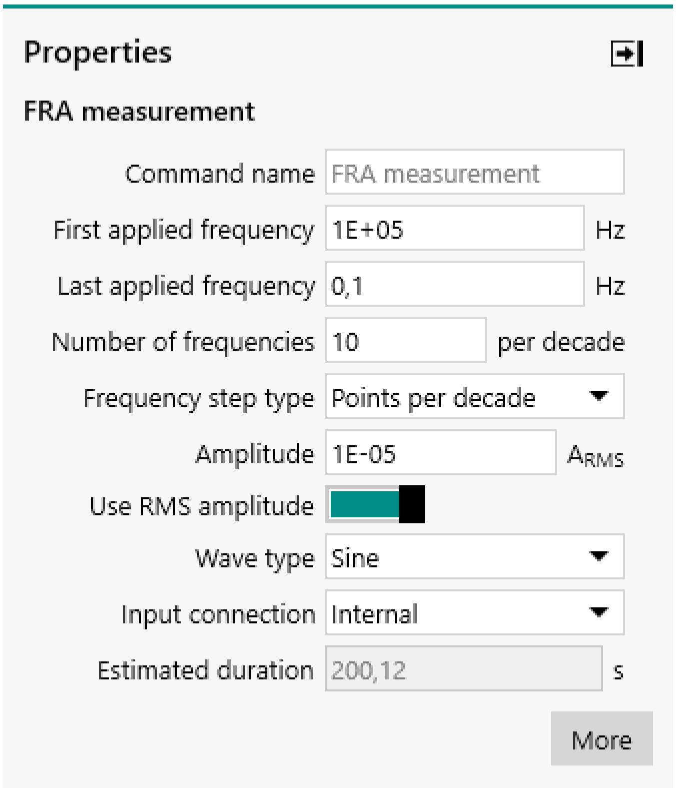


Figure 708 The measurement properties of the FRA measurement command

- **FRA measurement**
  - Start frequency: 100 kHz
  - Stop frequency: 0.1 Hz
  - Number of frequencies per decade: 10
  - Amplitude: 10 µA
  - Use RMS amplitude: yes
  - Wave type: sine
  - Input connection: internal

The procedure samples the following signals *(see Figure 709, page 575)*:

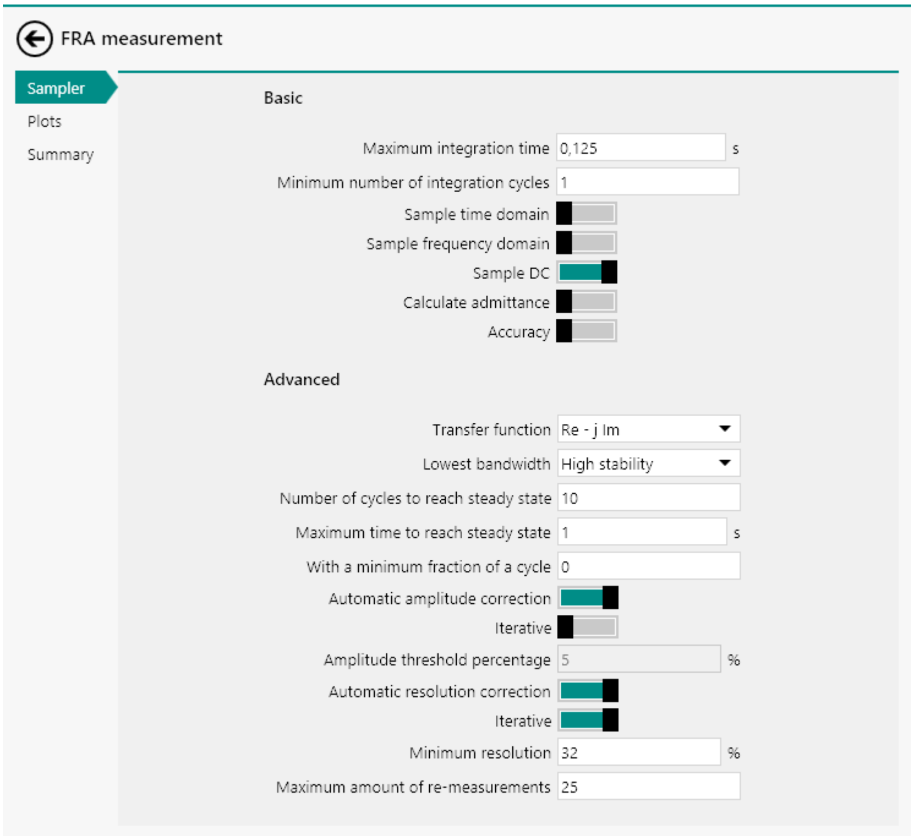


Figure 709 The sampler of the FRA measurement command

- DC signals

The procedure plots the following data *(see Figure 710, page 575)*:

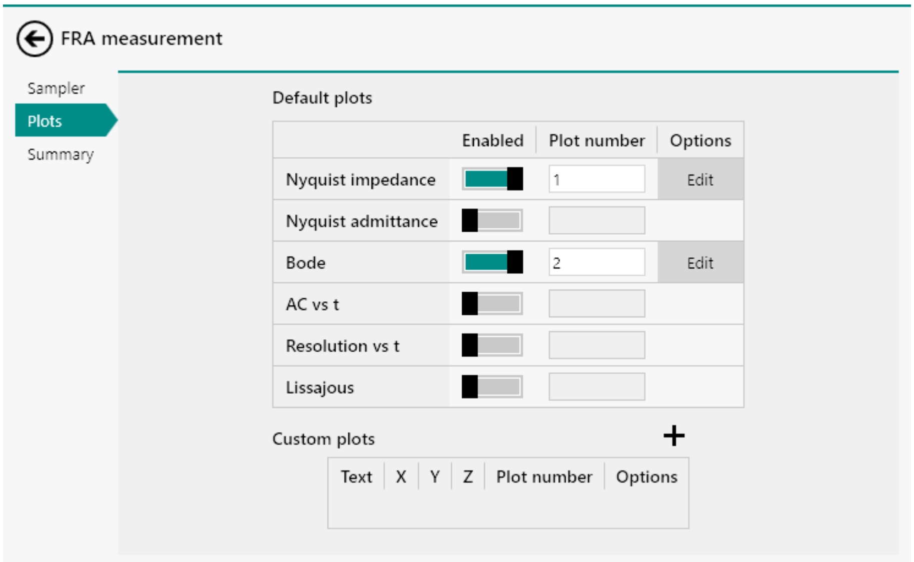


Figure 710 The plots of the FRA measurement command

- Nyquist impedance
- Bode

8.6.3 FRA potential scan

The default FRA potential scan procedure provides an example of an electrochemical impedance spectroscopy measured repeated for a pre-defined series of DC potentials *(see Figure 711, page 576)*.

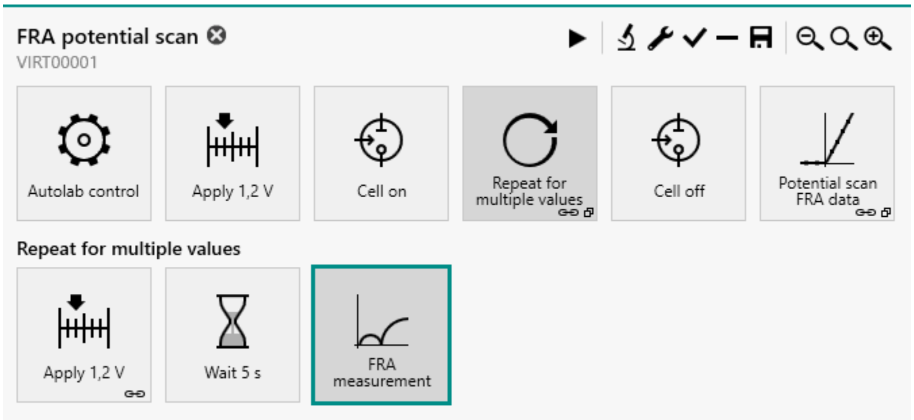


Figure 711 The default FRA potential scan procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The FRA potential scan procedure performs an impedance measurement at twelve different potential values. The potential values are set using a **Repeat** command.

The **Repeat** command is used in the *Repeat for multiple values* mode and is preconfigured to cycle through twelve potentials values, starting at 1.2 V until 0.1 V, using a linear distribution *(see Figure 712, page 577)*.

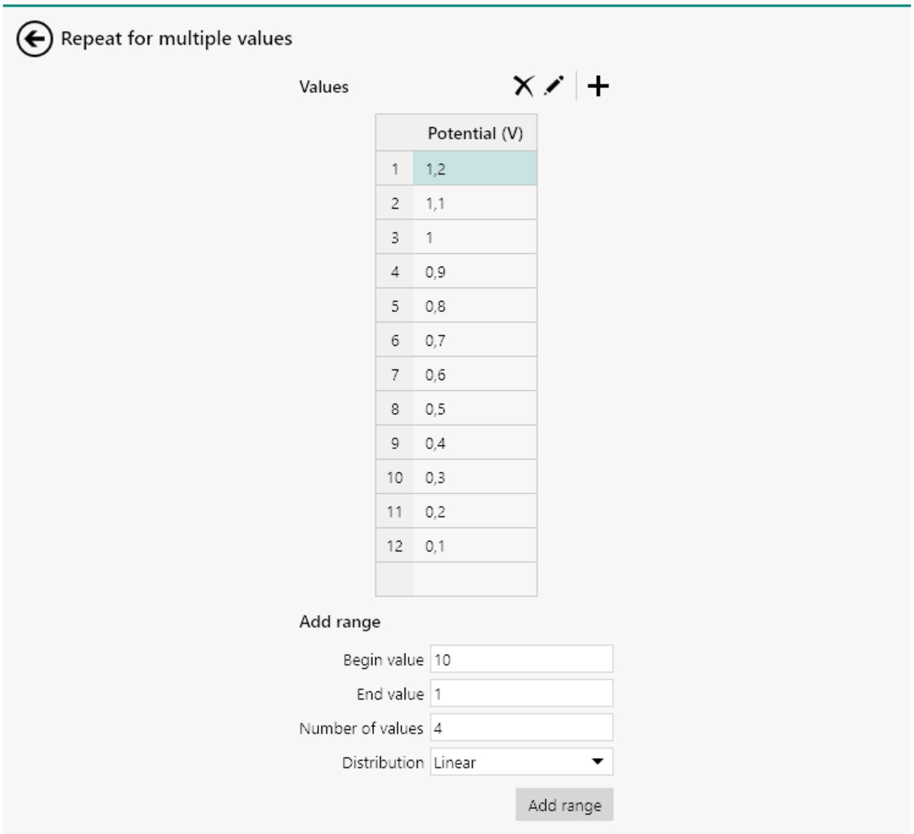


Figure 712 The repeat loop using the default FRA potential scan procedure

The *Potential* parameter, created by the **Repeat** command, is linked to the **Apply** command included in the repeat loop *(see Figure 713, page 577)*.

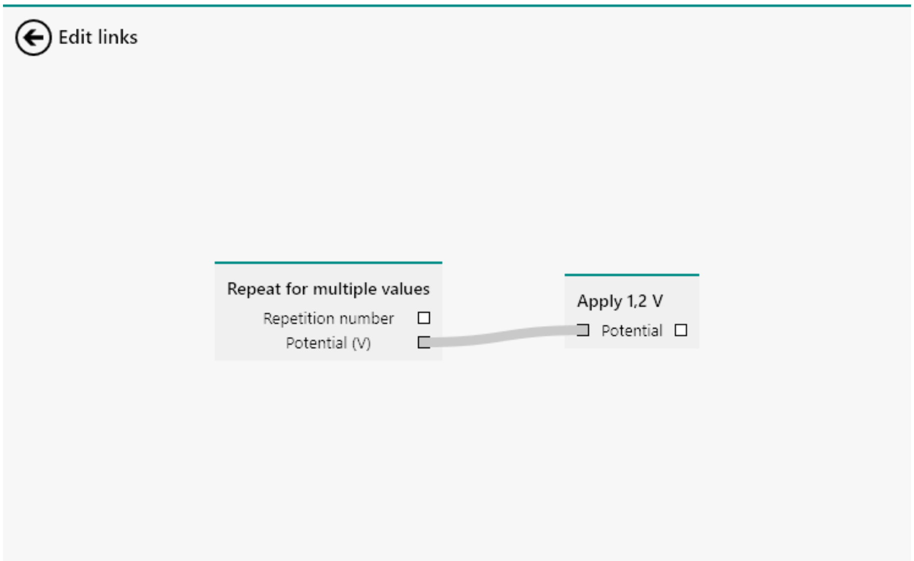


Figure 713 The link used to set the potential values

The procedure has the following measurement properties, specified for the **FRA measurement** command *(see Figure 714, page 578)*:

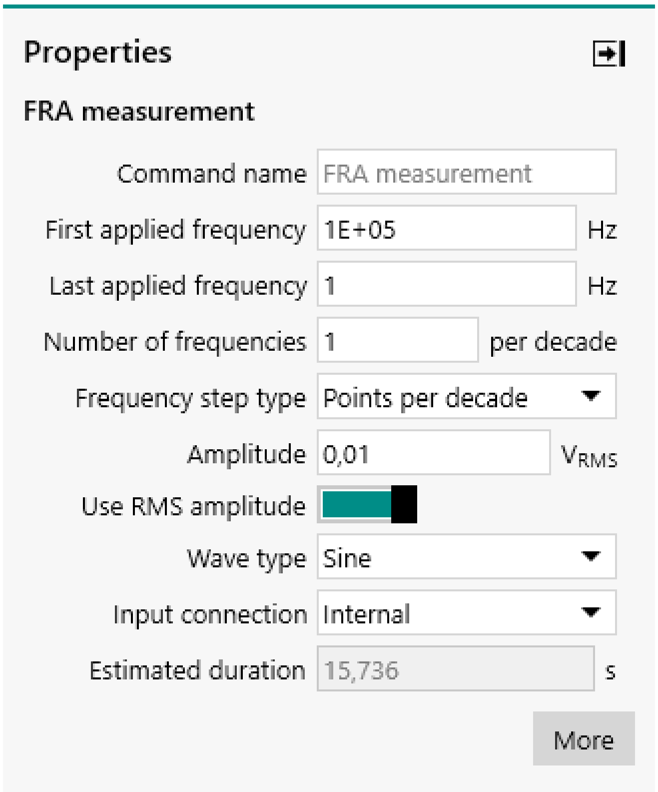


Figure 714 The measurement properties of the FRA measurement command

- **FRA measurement**
  - Start frequency: 100 kHz
  - Stop frequency: 1 Hz
  - Number of frequencies per decade: 1
  - Amplitude: 0.010 V
  - Use RMS amplitude: yes
  - Wave type: sine
  - Input connection: internal

The procedure samples the following signals *(see Figure 715, page 579)*:

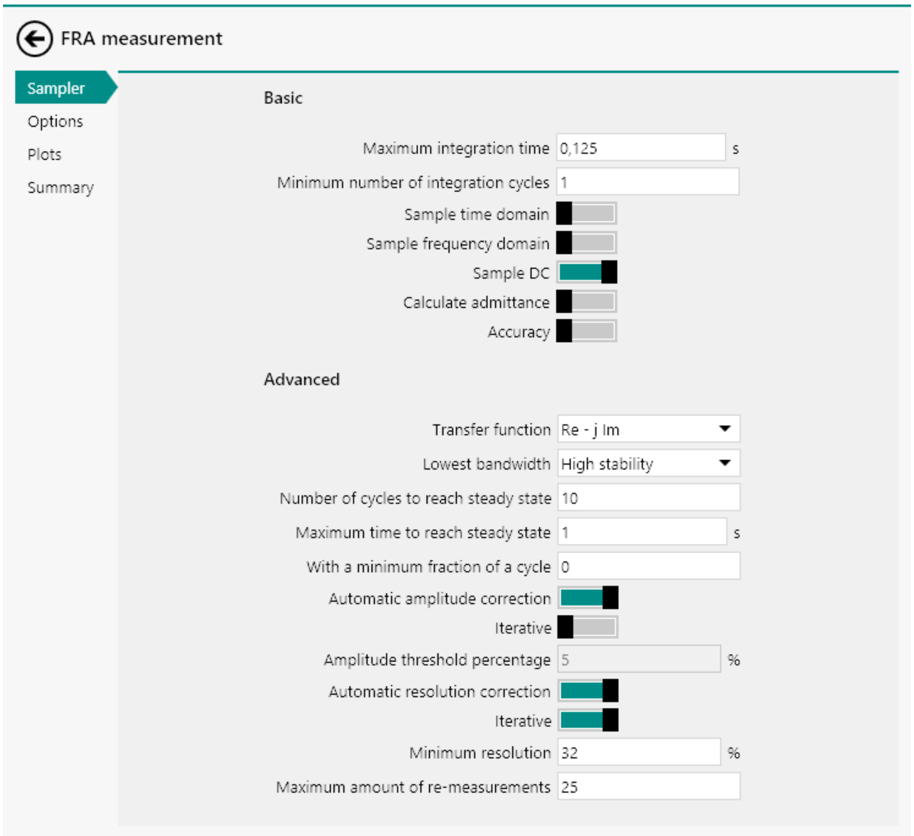


Figure 715 The sampler of the FRA measurement command

- DC signals

The procedure uses the following options *(see Figure 716, page 579)*:

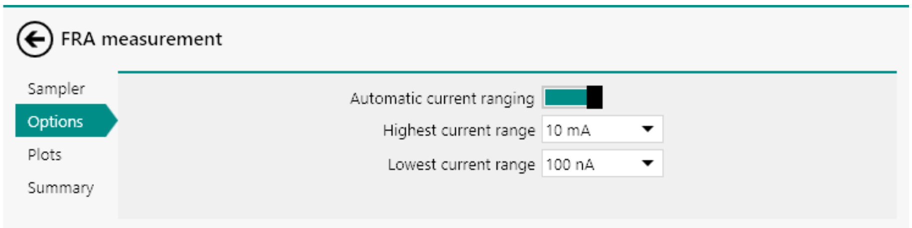


Figure 716 The options of the FRA measurement command

- Automatic current ranging
  - Highest current range: 100 mA
  - Lowest current range: 100 nA

The procedure plots the following data *(see Figure 717, page 580)*:

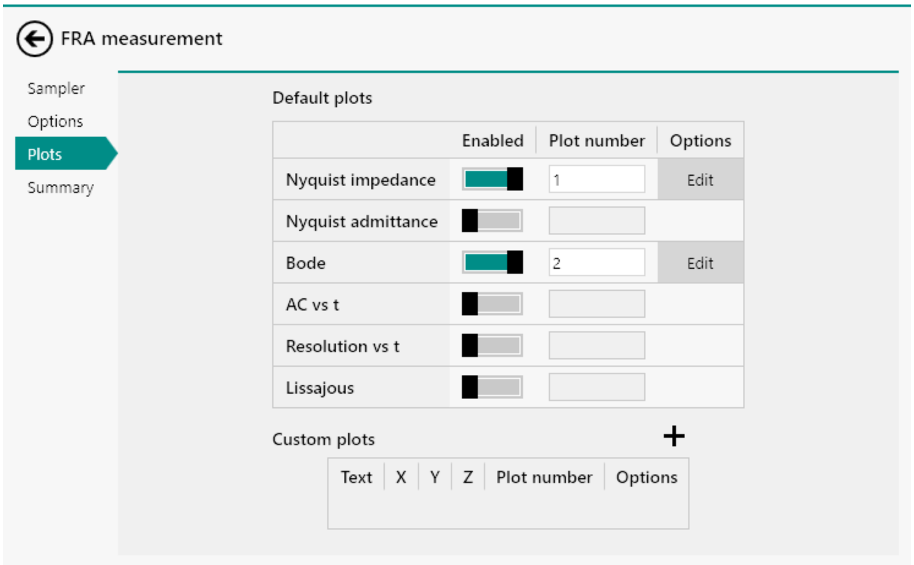


Figure 717 The plots of the FRA measurement command

- Nyquist impedance
- Bode

The **Potential scan FRA data** command located at the end of the procedure is used to automatically generate Mott-Schottky plots. For more information on this command, please refer to *Chapter 7.9.5*.

8.6.4 FRA current scan

The default FRA current scan procedure provides an example of an electrochemical impedance spectroscopy measured repeated for a pre-defined series of DC currents *(see Figure 718, page 580)*.

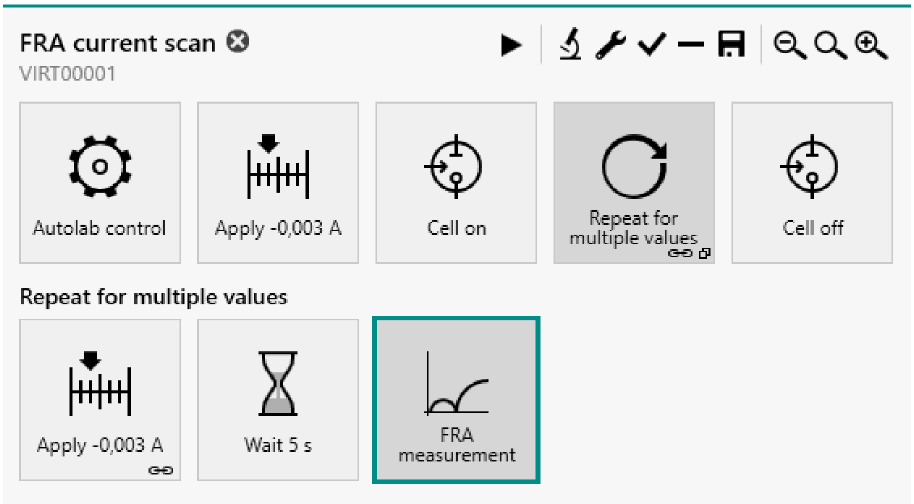


Figure 718 The default FRA current scan procedure

NOTE

The galvanostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The FRA current scan procedure performs an impedance measurement at seven different current values. The current values are set using a **Repeat** command.

The **Repeat** command is used in the *Repeat for multiple values* mode and is preconfigured to cycle through seven current values, starting at - 3 mA until 3 mA, using a linear distribution *(see Figure 719, page 581)*.

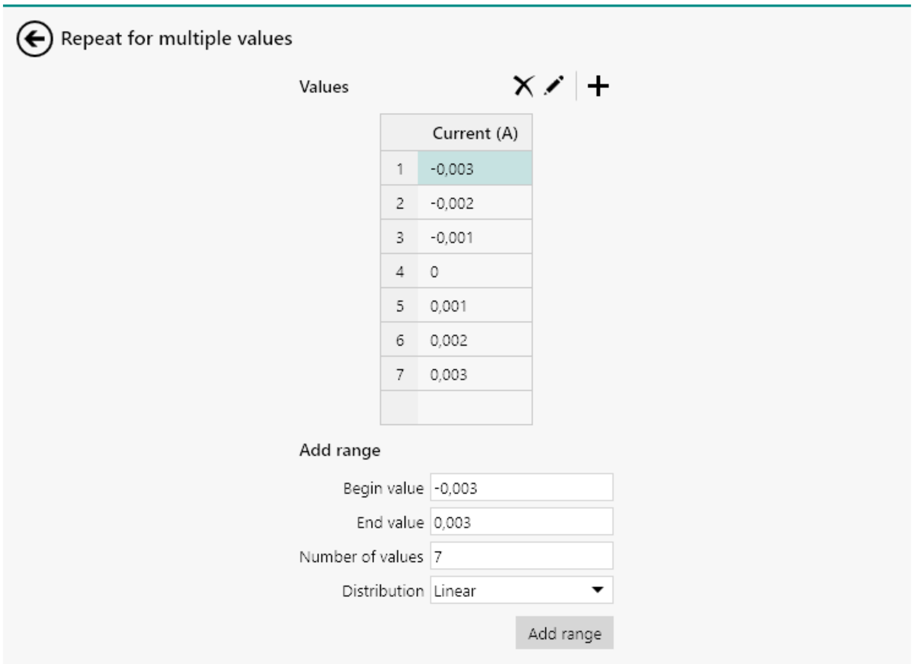


Figure 719 The repeat loop using the default FRA current scan procedure

The *Current* parameter, created by the **Repeat** command, is linked to the **Apply** command included in the repeat loop *(see Figure 720, page 582)*.

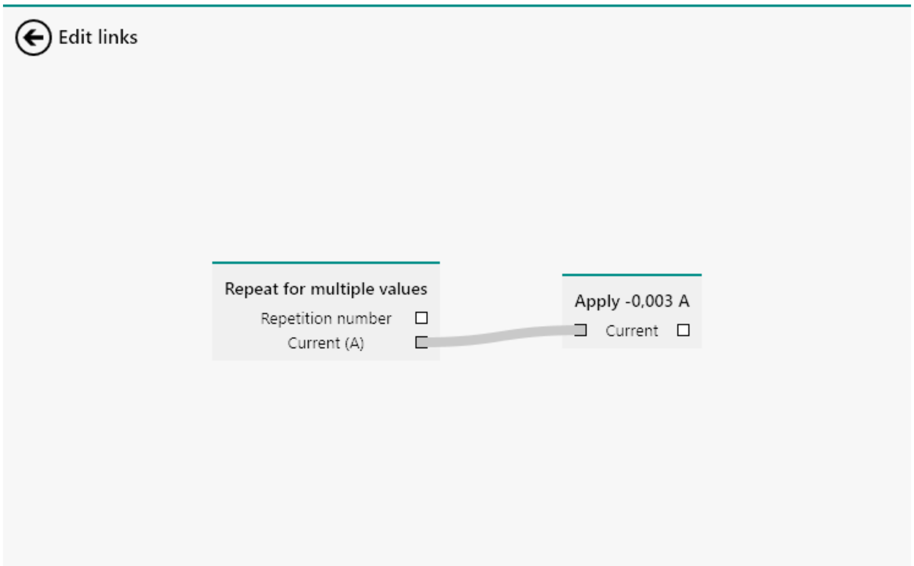


Figure 720 The link used to set the current values

The procedure has the following measurement properties, specified for the **FRA measurement** command *(see Figure 721, page 582)*:

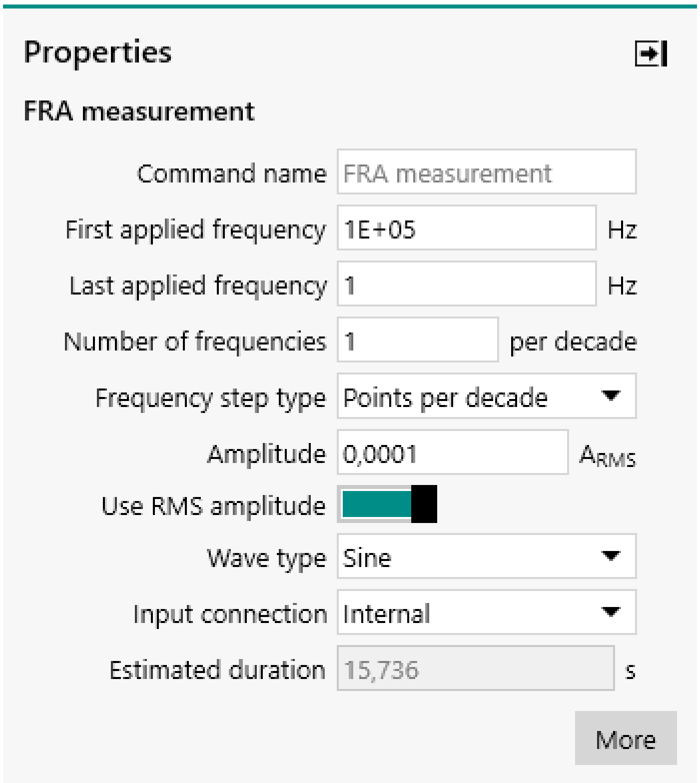


Figure 721 The measurement properties of the FRA measurement command

- **FRA measurement**
  - Start frequency: 100 kHz
  - Stop frequency: 1 Hz
  - Number of frequencies per decade: 1
  - Amplitude: 10 µA
  - Use RMS amplitude: yes
  - Wave type: sine
  - Input connection: internal

The procedure samples the following signals *(see Figure 722, page 583)*:

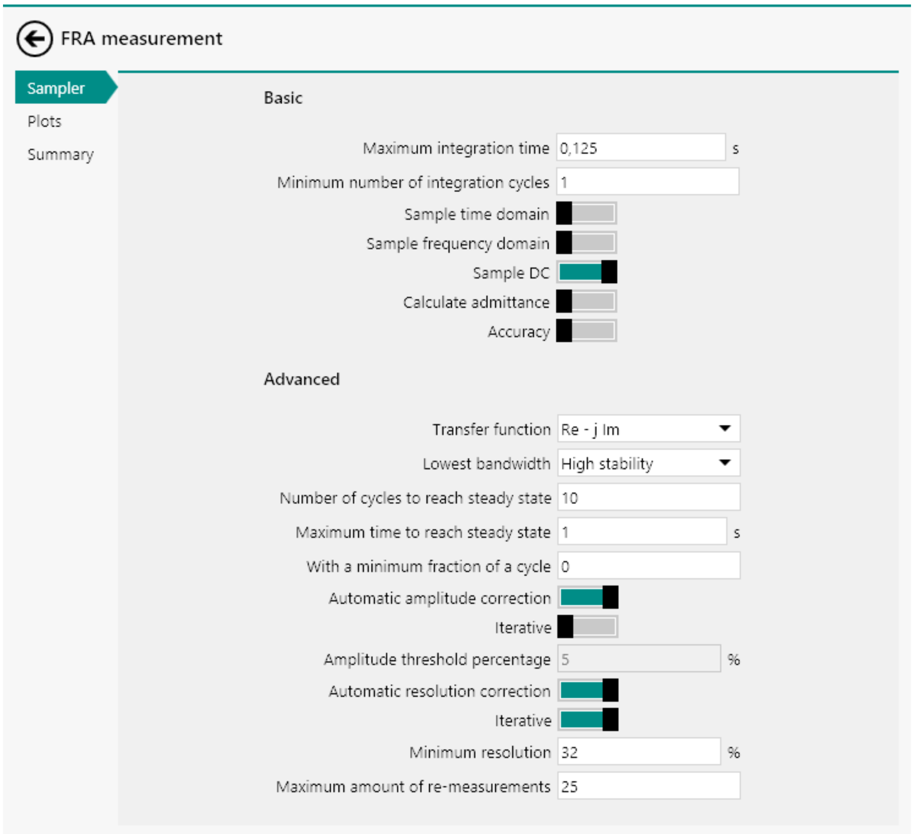


Figure 722 The sampler of the FRA measurement command

- DC signals

The procedure plots the following data *(see Figure 723, page 584)*:

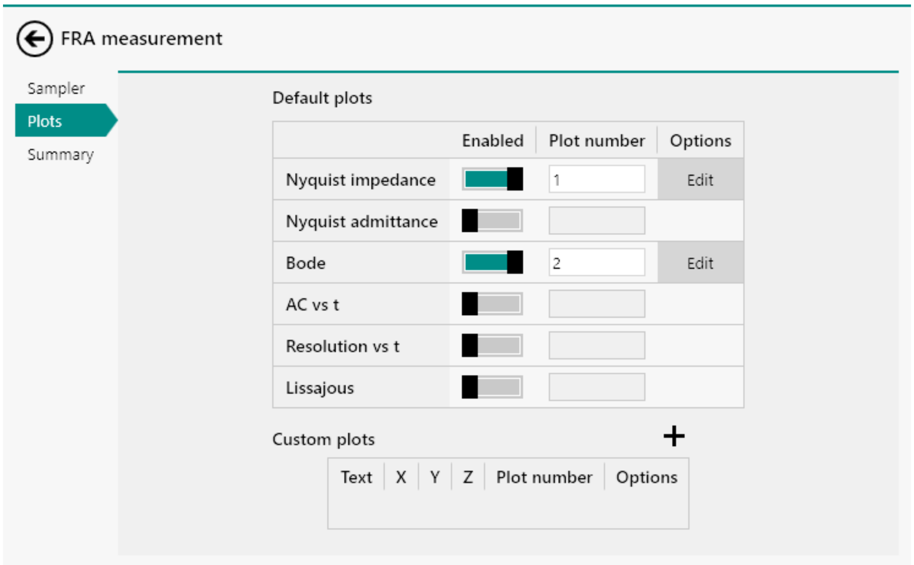


Figure 723 The plots of the FRA measurement command

- Nyquist impedance
- Bode

8.6.5 FRA time scan potentiostatic

The default FRA time scan procedure provides an example of an electrochemical impedance spectroscopy measured at fixed time intervals, in potentiostatic mode *(see Figure 724, page 584)*.

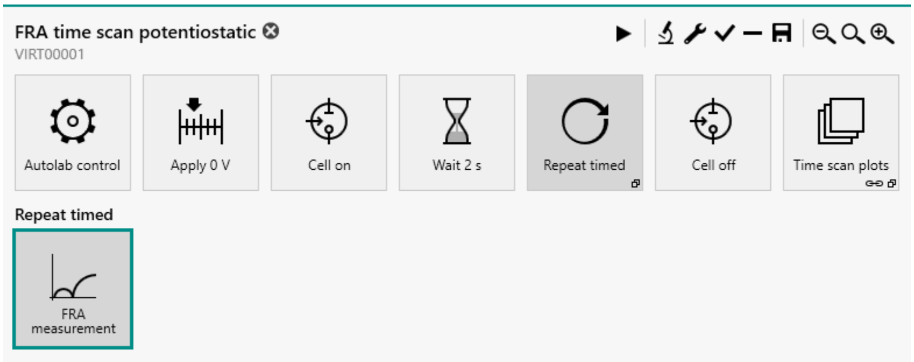


Figure 724 The default FRA time scan potentiostatic procedure

NOTE

The potentiostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure uses a **Repeat** command, used in *Timed repeat* mode, for a pre-defined duration of 200 s and interval time of 20 s *(see Figure 725, page 585)*.

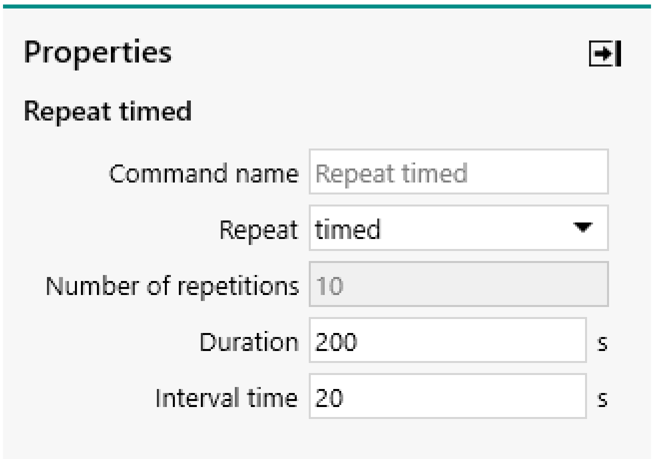


Figure 725 The properties of the Repeat command

The procedure has the following measurement properties, specified for the **FRA measurement** command *(see Figure 726, page 585)*:

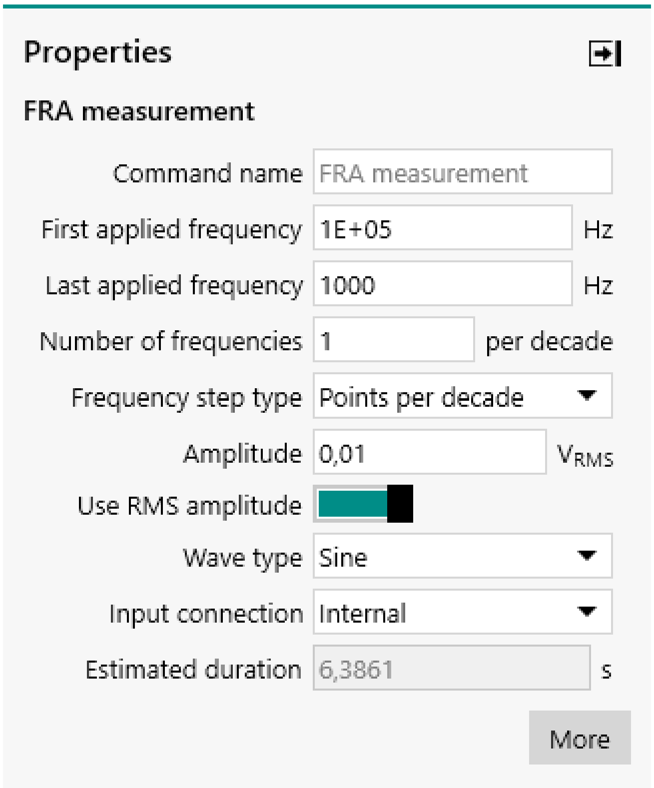


Figure 726 The measurement properties of the FRA measurement command

- **FRA measurement**
  - Start frequency: 100 kHz
  - Stop frequency: 1 kHz
  - Number of frequencies per decade: 1
  - Amplitude: 0.010 V
  - Use RMS amplitude: yes
  - Wave type: sine
  - Input connection: internal

The procedure samples the following signals *(see Figure 727, page 586)*:

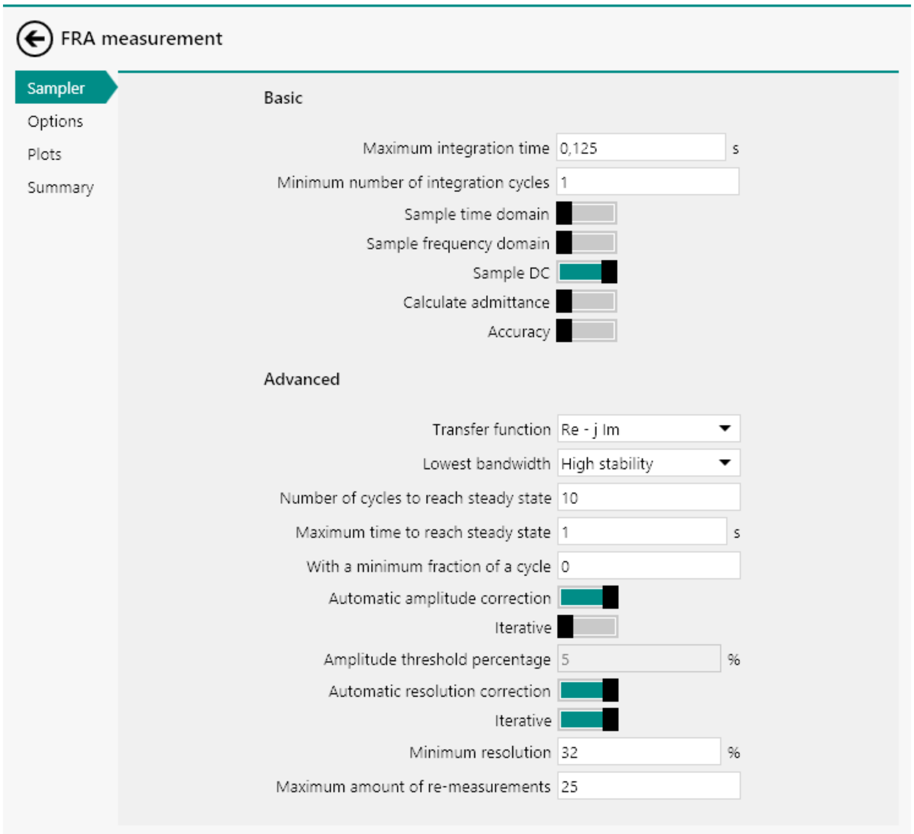


Figure 727 The sampler of the FRA measurement command

- DC signals

The procedure uses the following options *(see Figure 728, page 586)*:

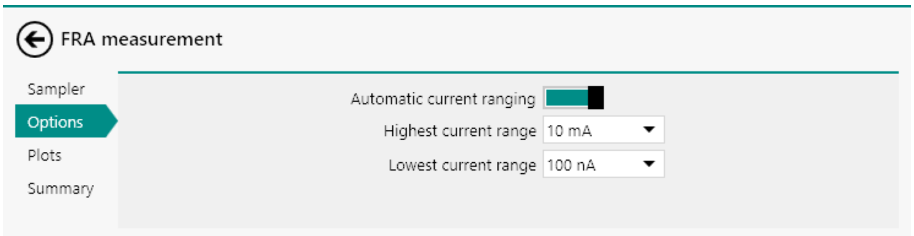


Figure 728 The options of the FRA measurement command

- Automatic current ranging
  - Highest current range: 10 mA
  - Lowest current range: 100 nA

The procedure plots the following data *(see Figure 729, page 587)*:

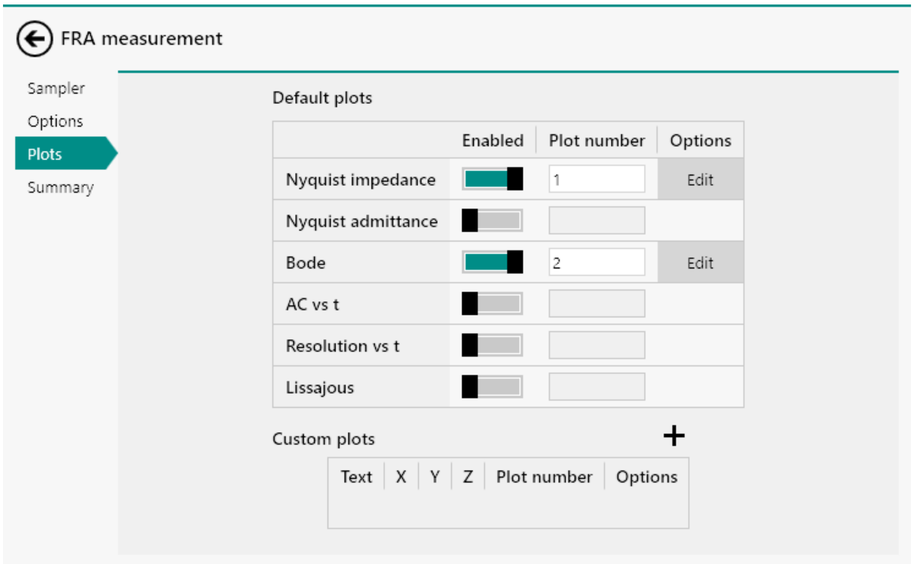


Figure 729 The plots of the FRA measurement command

- Nyquist impedance
- Bode

Additionally, the procedure gathers all the measured data points and plots the following time resolved data:

- Z vs t
- -phase vs t

8.6.6 FRA time scan galvanostatic

The default FRA time scan procedure provides an example of an electrochemical impedance spectroscopy measured at fixed time intervals, in galvanostatic mode *(see Figure 730, page 587)*.

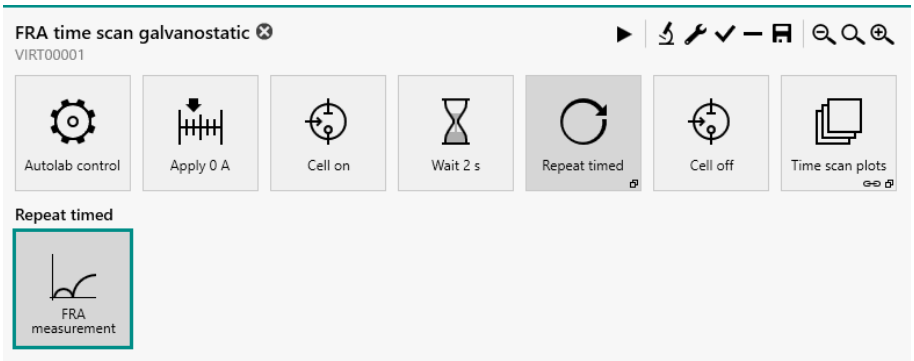


Figure 730 The default FRA time scan galvanostatic procedure

> **NOTE**
>
> The galvanostatic mode is selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure uses a **Repeat** command, used in *Timed repeat* mode, for a pre-defined duration of 200 s and interval time of 20 s *(see Figure 731, page 588)*.

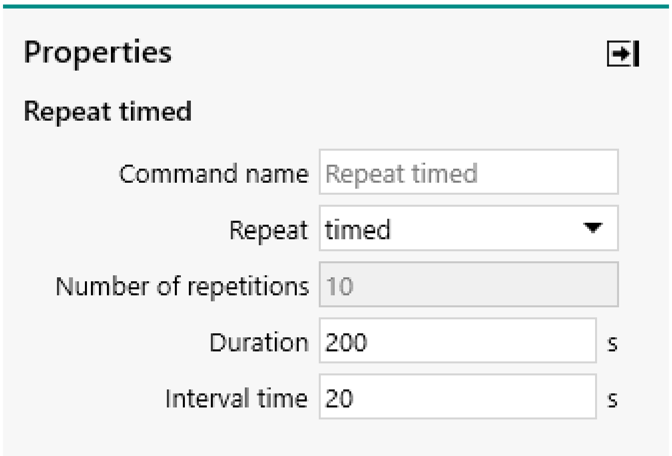


Figure 731 The properties of the Repeat command

The procedure has the following measurement properties, specified for the **FRA measurement** command *(see Figure 732, page 589)*:

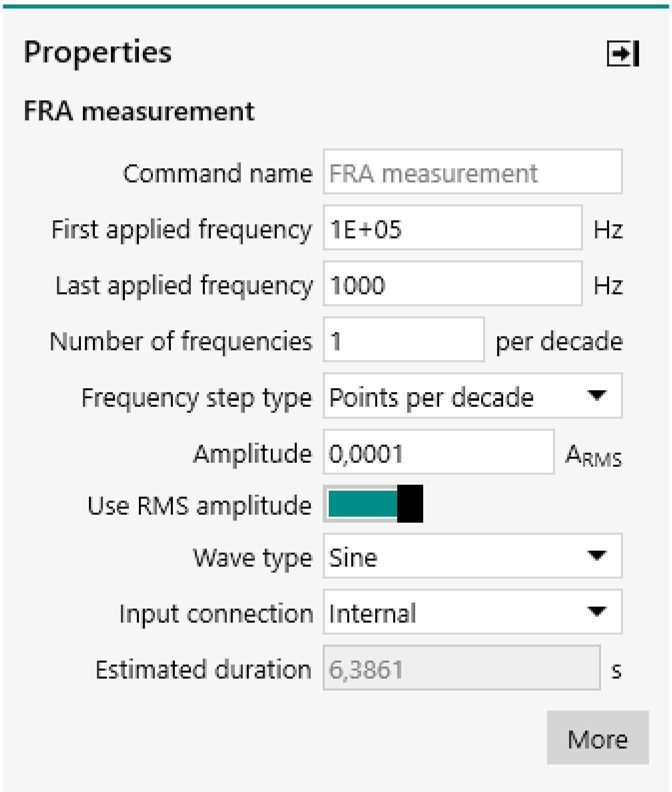


Figure 732 The measurement properties of the FRA measurement command

- **FRA measurement**
  - Start frequency: 100 kHz
  - Stop frequency: 1 kHz
  - Number of frequencies per decade: 1
  - Amplitude: 0.0001 A
  - Use RMS amplitude: yes
  - Wave type: sine
  - Input connection: internal

The procedure samples the following signals *(see Figure 733, page 590)*:

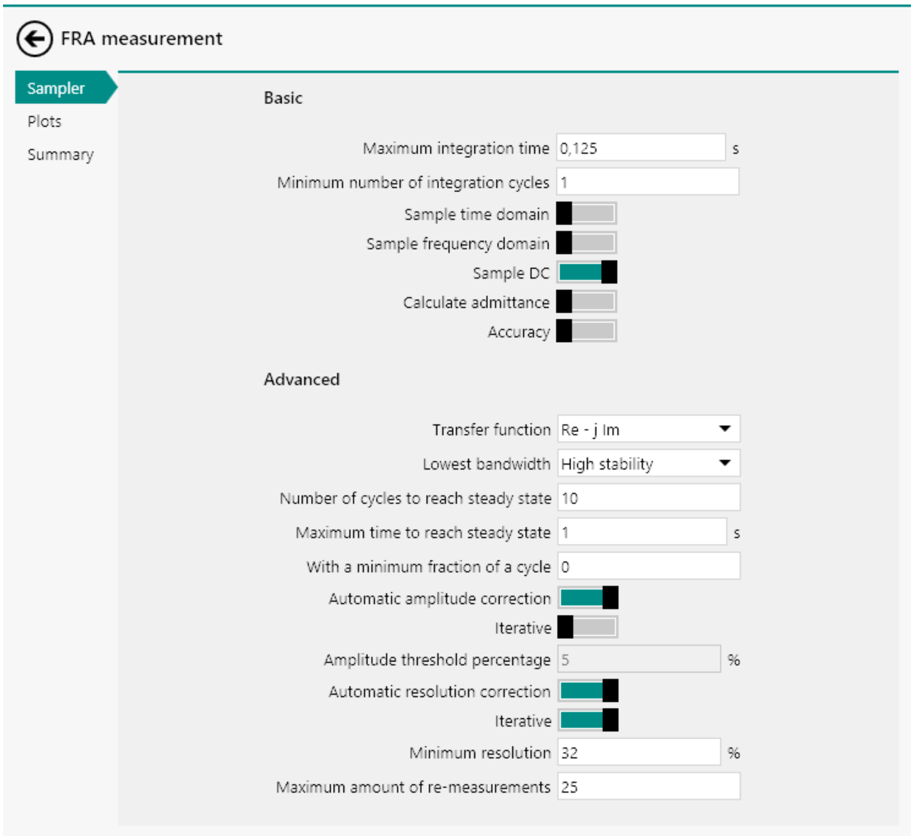


Figure 733 The sampler of the FRA measurement command

- DC signals

The procedure plots the following data *(see Figure 734, page 590)*:

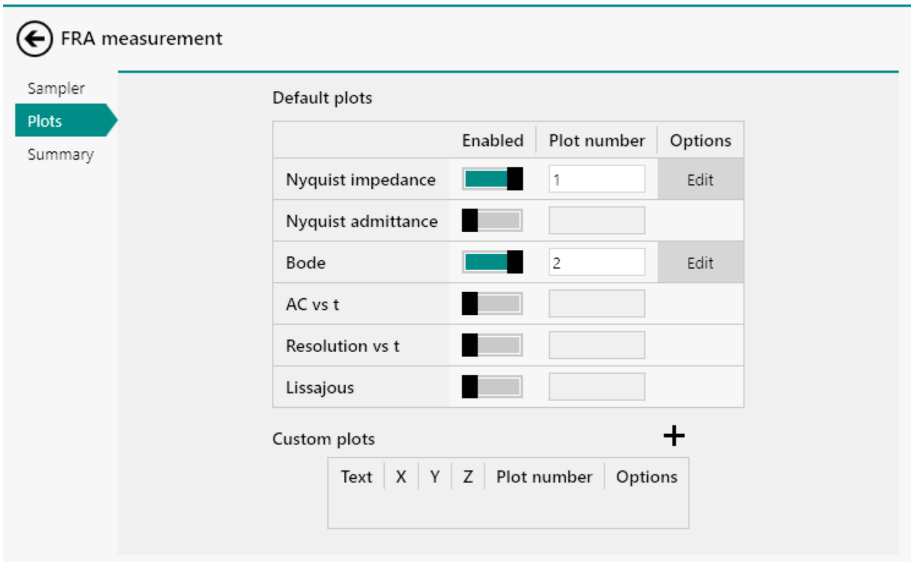


Figure 734 The plots of the FRA measurement command

- Nyquist impedance
- Bode

Additionally, the procedure gathers all the measured data points and plots the following time resolved data:

- Z vs t
- -phase vs t

8.6.7 Electrochemical Frequency Modulation

The default Electrochemical Frequency Modulation procedure provides an example of an electrochemical frequency modulation measurement (EFM) in potentiostatic conditions *(see Figure 735, page 591)*.

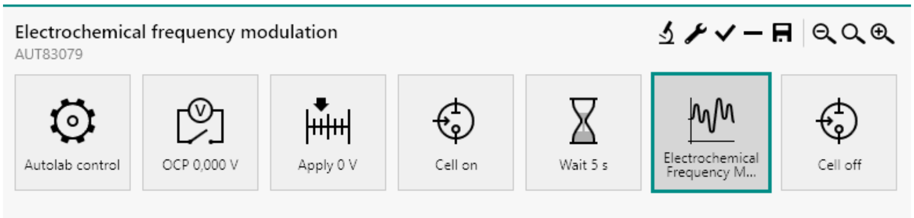


Figure 735 The default Electrochemical Frequency Modulation procedure

> **NOTE**
>
> The potentiostatic mode and current range are selected at the beginning of the procedure using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

The procedure has the following measurement properties, specified for the **Electrochemical Frequency Modulation** command :

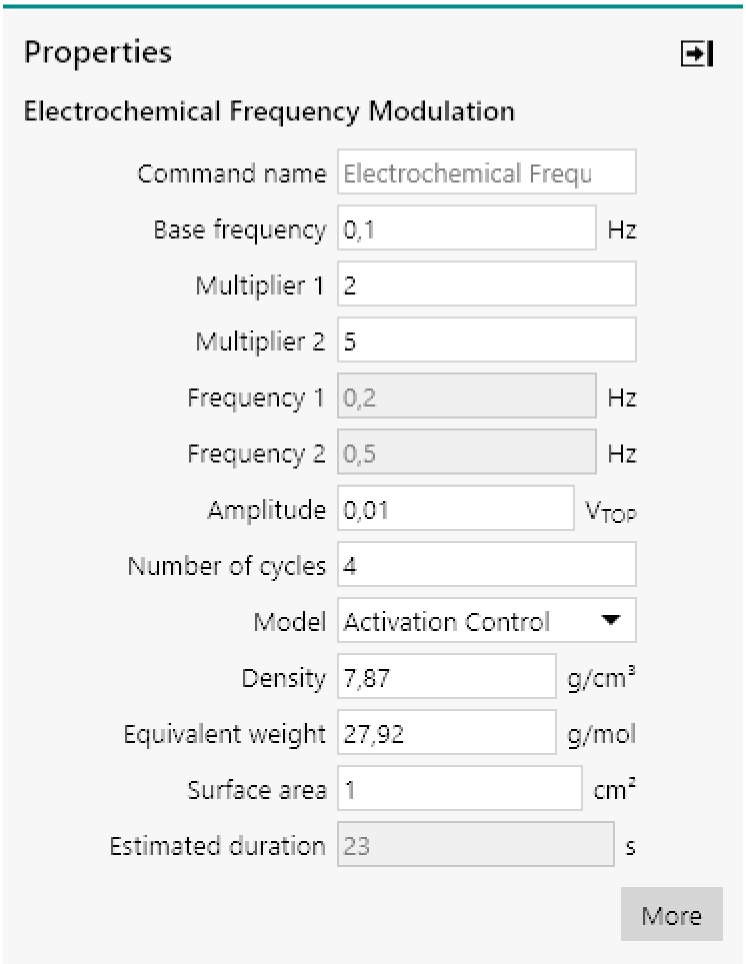


Figure 736 The measurement properties of the Electrochemical Frequency Modulation command

- **Electrochemical Frequency Modulation**
  - Base frequency: 0,1 kHz
  - Multiplier 1: 2
  - Multiplier 2: 5
  - Amplitude: 0.010 V
  - Number of cycles: 4
  - Model: Activation Control
  - Density: 7,87 g/cm^3
  - Equivalent weight: 27,92 g/mol
  - Surface area: 1 cm^2

The procedure plots the following data *(see Figure 737, page 593)*:

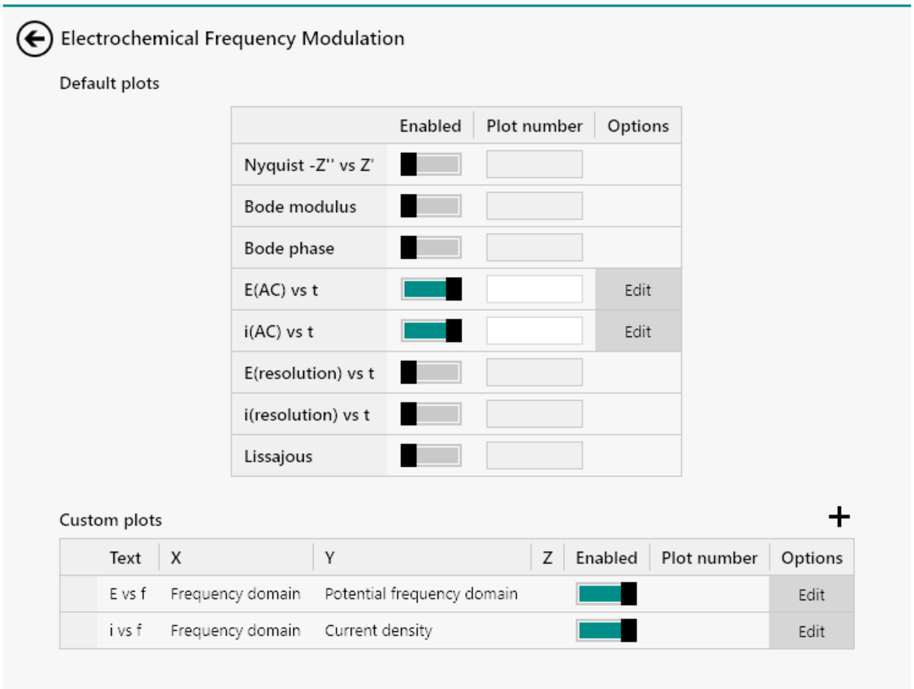


Figure 737 The plots of the Electrochemical Frequency Modulation command

- E(AC) vs t
- i(AC) vs t
- E vs f
- i vs f

9 Additional measurement command properties

Most of the measurement commands in NOVA have additional properties which can be accessed through the More button, as shown *Figure 738*.

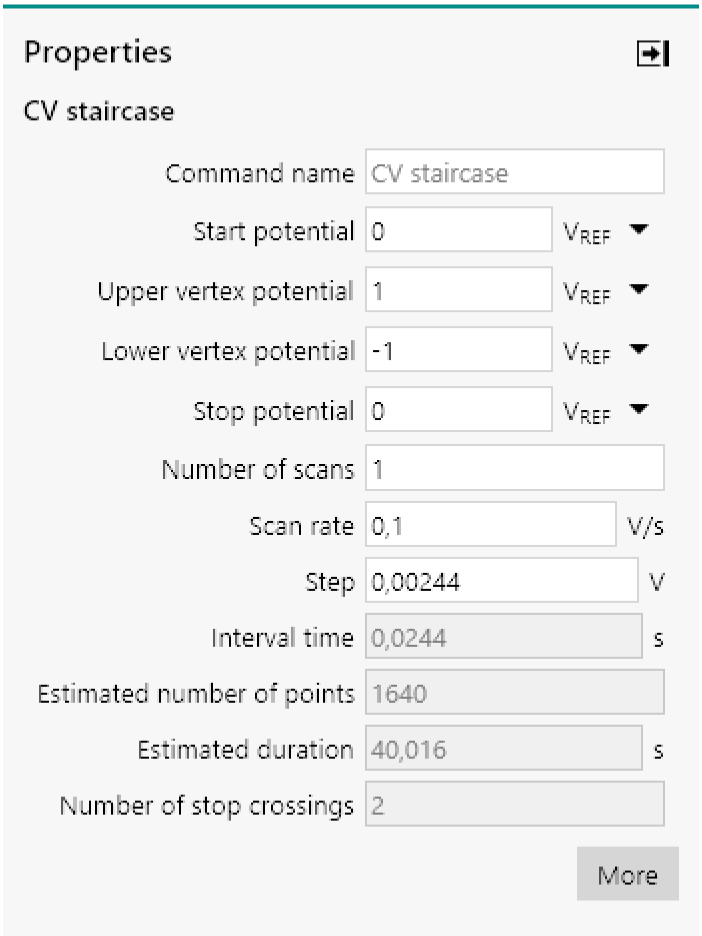


Figure 738 Additional properties are provided by most measurement commands

NOTE

Not all measurement command provide additional options and the provided option may change, depending on the measurement command.

This section provides information on the following additional properties:

- **Sampler:** the sampler defines which signals to sample during the measurement.

- **Options:** the options are additional measurement settings that affect how the data is measured.
- **Plots:** the plots define how the measured data should be plotted.
- **Advanced:** advanced acquisition properties used during the measurement.

> **NOTE**
>
> The **Advanced** properties are only available for the **CV staircase** and **LSV staircase** commands.

9.1 Sampler

The sampler defines which signals are measured or calculated by the command and how these signals should be measured. For each available signal,  toggles are provided to control the sampler settings *(see Figure 739, page 595)*.

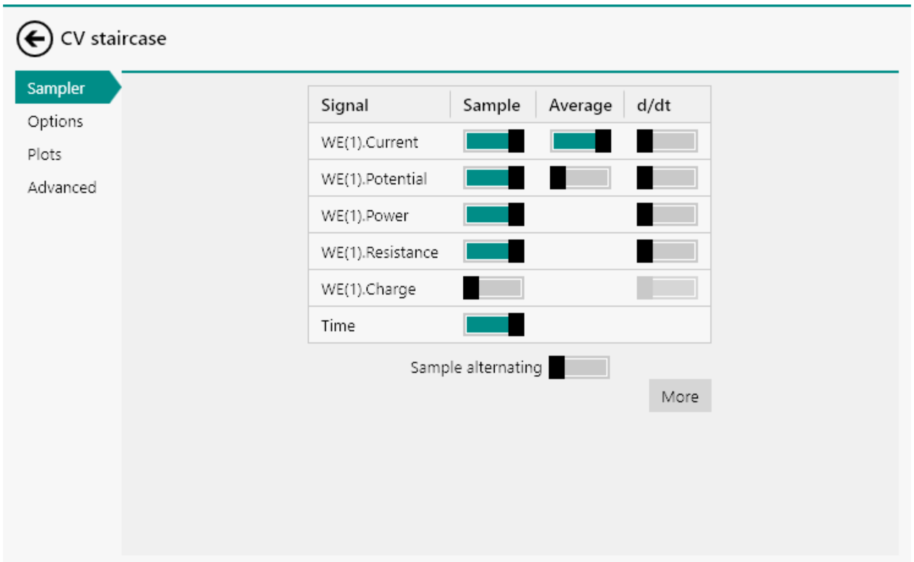


Figure 739 The sampler defines which signals are measured or calculated by the command

> **NOTE**
>
> The list of signals displayed in the sampler depends on the hardware setup.

Depending on the type of signal, the following settings can be defined in the sampler:

- **Sample:** this setting defines that the signal is sampled by the command. A single analog-to-digital conversion is performed for a sampled signal.
- **Average:** this setting defines that a sampled signal must be averaged. When a sampled signal is averaged, as many analog-to-digital conversions are performed and an averaged value is stored. Averaging a signal significantly improves the signal-to-noise ratio.
- **d/dt:** this setting defines that the time derivative of a sampled signal must be calculated.

NOTE

The **average** setting is only available for signals that can be sampled. Some of the signals provided in the sampler are calculated (Power, Resistance and Charge) while other signals are digitized by a dedicated optional module (EQCM signals).

NOTE

Up to six signals can be averaged during a measurement.

Additionally, a toggle is provided for the **Sample alternating** setting, below the sampler table *(see Figure 739, page 595)*. This setting defines how averaged signals are sampled by the command:

- **Sample alternating off:** when this setting is off, all averaged signals are sampled, sequentially. The WE(1).Current signal is always sampled last.
- **Sample alternating on:** when this setting is on, all averaged signals are sampled at the same time, alongside the WE(1).Current signal.

Clicking the More button opens a new screen that provides additional information on the exact timing of the sampler, in µs. The signals are provided in a table, as shown in *Figure 740*.

Sampler

Signal	Start time (µs)	Duration (µs)
Time	18000	0
WE(1).Potential	18000	200
WE(1).Current	18200	6100
WE(1).Power WE(1).Resistance	24300	100
Total duration		6400

Figure 740 Detailed view of timing used by the sampler

The signals are listed in the table in chronological order to sampling. The Start time column provides the time, in µs, after which the sampling of the signal starts, with respect to the beginning of the interval time. The Duration column provides the duration, in µs, during which each signal is sampled. Depending on the type of signal and on the sampling method, the following durations are used:

- **Time:** the duration of the sampling of the Time signal is always 0 µs.
- **Sampled signals:** the duration of the sampling of signals that are not averaged is at most 200 µs.
- **Calculated signals:** the duration of the calculations carried out for the determination of calculated signals is at most 100 µs.

9.2 Automatic current ranging

The **Automatic current ranging** option specifies which of the available current ranges can be used by the measurement command. When this option is used, the instrument will automatically select the most suitable current range available. The instrument will also change the current range in the following cases:

- **Current overload:** the measured current exceeds the current overload threshold. The active current range is adjusted to the next available higher range.
- **Current underload:** the measured current exceeds the current underload threshold. The active current range is adjusted to the next available lower range.

NOTE

Five consecutive overload or underload detections are required to trigger a change in current range.

Automatic current ranging settings are defined in the dedicated table, in the **Automatic current ranging** sub-panel *(see Figure 741, page 598)*.

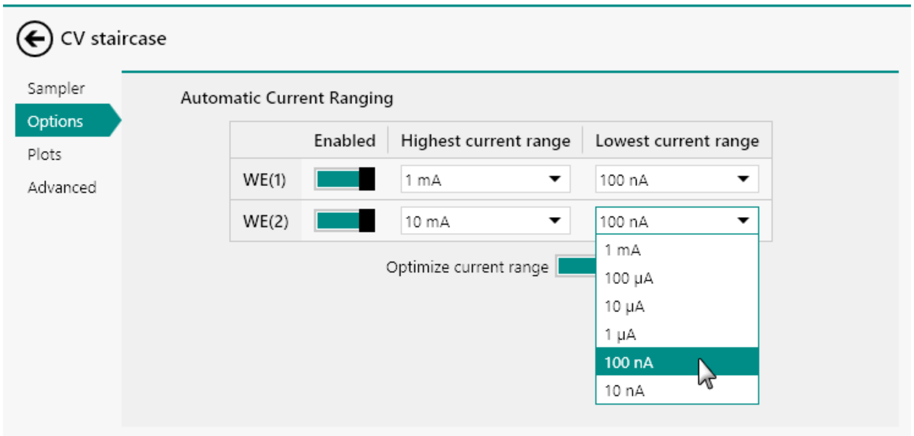


Figure 741 The Automatic current ranging option is defined in a dedicated sub-panel

Three properties can be specified for each working electrode:

- **Enabled:** a toggle used to set the Automatic current ranging on or off.
- **Highest current range:** defines the highest possible current range, using the provided drop-down list.
- **Lowest current range:** defines the lowest possible current range, using the provided drop-down list.

Additionally, the **Optimize current range** toggle is located below the table. When this setting is on, the instrument will automatically adjust the current range of each electrode for which the **Automatic current ranging** option is enabled to the most suitable current range **before** the command starts measuring.

NOTE

It is highly recommended to use the **Optimize current range** option whenever using the **Automatic current ranging** option in order to ensure that each measurement starts in the most suitable current range.

9.3 Cutoffs

Cutoffs are convenient tools which can be used to control the experimental conditions when a signal exceeds a user-defined threshold. Cutoffs can be defined for any signal available in the Sampler and can be defined for any measurement command that uses the Sampler.

Cutoffs are defined in the dedicated table, in the **Cutoffs** sub-panel *(see Figure 742, page 599)*.

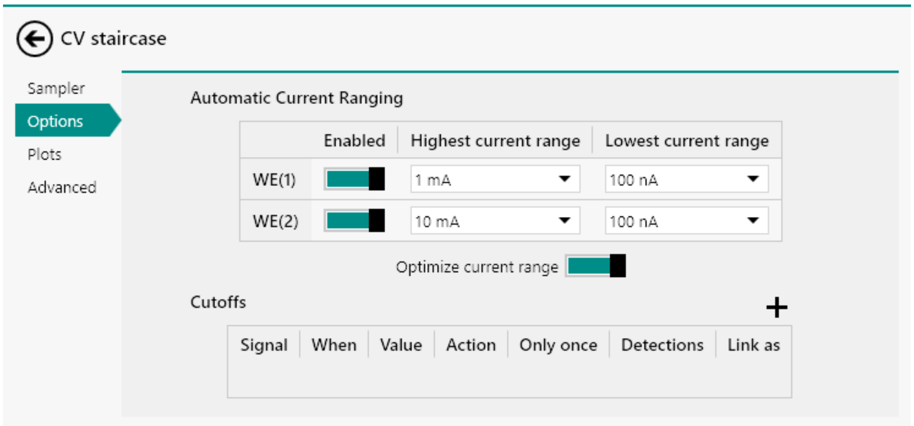


Figure 742 The Cutoffs are defined in a dedicated sub-panel

Seven properties are defined per cutoff:

- **Signal:** the signal on which the cutoff is applied.
- **When:** defines the inequality needed to trigger the cutoff.
- **Value:** defines the threshold value of the signal.
- **Action:** defines what should happen when the cutoff condition is met. Four or five actions are available:
  - **Stop command:** the current command is stopped as soon as the cutoff is triggered and the procedure continues.
  - **Stop measurement:** the current command, as well as all consecutive measurement commands are stopped as soon as the cutoff is triggered and the procedure proceeds from the first non-measurement command in the sequence.
  - **Stop complete procedure:** the complete procedure is stopped as soon as the cutoff is triggered.
  - **Reverse scan direction:** the scan direction is reversed as soon as the cutoff is triggered.
  - **And:** no action is taken when the cutoff is triggered. Instead, this cutoff is joined to one or more cutoff conditions. When all the cutoffs joined with the And action are triggered, the collective action is executed.

- **Only once:** specifies if the cutoff action should be executed only once or each time the cutoff condition is met, using the provided toggle .
- **Detections:** defines the number of consecutive detections required to trigger the cutoff.
- **Link as:** defines a unique name for the cutoff **Value** that can be used to link to other command parameters in the procedure editor.

NOTE

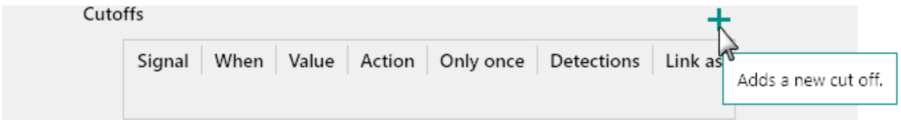
The **Reverse scan** direction action is only available for the **LSV staircase** and the **CV staircase** commands.

9.3.1 Cutoff configuration

The following steps describe how to add and configure a cutoff.

1 Add a cutoff to the list

Click on the + button to add a cutoff to the table.



NOTE

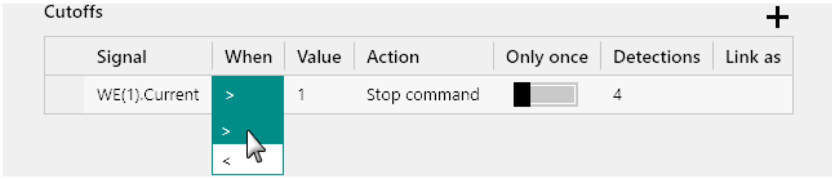
A cutoff on the WE(1).Current is automatically generated.

2 Specify the signal

Click on the cell of the **Signal** column and select the signal to use in the cutoff using the provided drop-down list.

3 Specify the inequality

Click on the cell of the **When** column and select the inequality to use in the cutoff using the provided drop-down list (< or >).

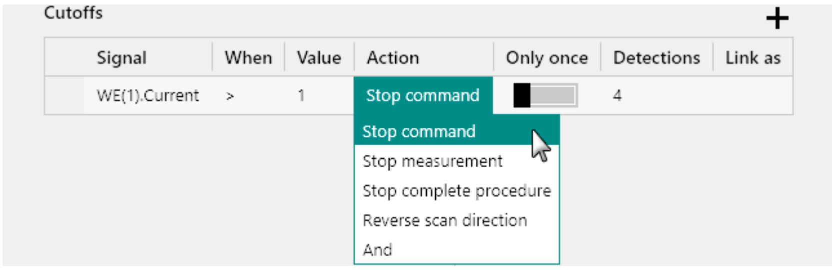


4 Specify the value

Specify the threshold value for the signal used in the cutoff in the corresponding cell of the **Value** column.

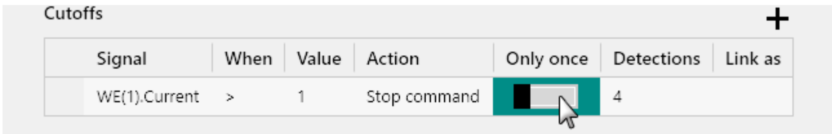
5 Specify the action

Click on the cell of the **Action** column and select the action to use in the cutoff using the provided drop-down list.



6 Set the only once property

Use the provided  toggle to define if the cutoff should be triggered only once or continuously.



7 Set the number of detections

Specify the number of detections value for the signal used in the cutoff in the corresponding cell of the **Detections** column.

Cutoffs +

Signal	When	Value	Action	Only once	Detections	Link as
WE(1).Current	>	1	Stop command		10	

8 **Specify a unique linkable name**

If required, a *unique* linkable name can be specified in the **Link as** column. If a name is specified, the threshold specified in the Value column can be linked to another command parameter in the procedure. Using this link, the actual threshold value can be modified during the execution of the procedure.

Cutoffs +

Signal	When	Value	Action	Only once	Detections	Link as
WE(1).Current	>	1	Stop command		10	Link

NOTE

The **Link as** property is optional and can be left empty if no link is required.

NOTE

To remove a cutoff from the table, select the row of the cutoff and click the – button above the **Cutoffs** table.

9.3.2 Combining cutoffs

It is possible to define more than one cutoff condition in the **Cutoffs** table. Depending on how the cutoffs conditions are defined, it is possible to arrange two or more cutoffs in two different ways:

- **OR arrangement:** each cutoff condition is defined as a standalone cutoff. The action defined for each of them is triggered whenever the corresponding threshold value is reached. This corresponds to a *OR* logical operator. The measurement command will be affected by each individual cutoff separately.
- **AND arrangement:** the two or more cutoff conditions can be joined with a *AND* action in order to trigger a single action when each of the involved cutoffs is triggered.

Figure 743 shows an example of three cutoff conditions. The first cutoff monitors the value of the WE(1).Current signal and forces the command to stop if this signal exceeds 1 mA. The second cutoff monitors the WE(1).Potential signal. When the value of this signal exceeds 1.2 V, the third cutoff will be monitored. When the third cutoff, specified on the WE(1).Charge signal is triggered, the complete procedure will be stopped.

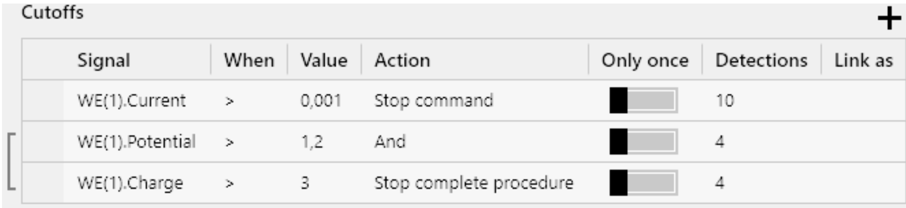


Figure 743 Multiple cutoffs

NOTE

The second and third cutoff shown in *Figure 743* are connected by a grey line on the left-hand side of the table, indicating that both cutoffs have a **AND** relationship.

9.4 Counters

Counters can be used during a measurement to perform dedicated actions whenever a condition associated with the counter is triggered. Each counter accumulates during a measurement, and it is possible to assign a specific instrumental action when a counter reaches a user defined value.

Since the counters are intrinsically linked to the measured data, the events triggered by the counters are directly correlated to the data points.

Counters are defined in the dedicated table, in the **Counters** sub-panel *(see Figure 744, page 603)*.

Figure 744 The Counters are defined in a dedicated sub-panel

Five properties are defined per counter:

- **When:** defines the equality or inequality for the counter.
- **Value:** defines the counter threshold value.

- **Action:** defines the action taken when the counter is triggered. Three actions are available:
  - **And:** no action is taken when the counter is triggered. Instead, this counter is joined to one or more counters conditions. When all the counters joined with the **And** action are triggered, the collective action is executed.
  - **Pulse:** a user-defined TTL pulse is generated at the DIO connector.
  - **Autolab control:** an instance of the Autolab control command is executed.
  - **Shutter control:** defines the state of the shutter of a connected Autolab or Avantes light source with TTL control.
  - **Get spectrum:** triggers the acquisition of a spectrum on a connected Autolab or Avantes spectrophotometer.
- **Reset:** specifies if the counter should be reset when it is triggered, using the provided toggle.
- **Properties:** defines the properties of the Action defined in the Action column.

9.4.1 Counter configuration

The following steps describe how to configure a counter.

1 Add a counter to the list

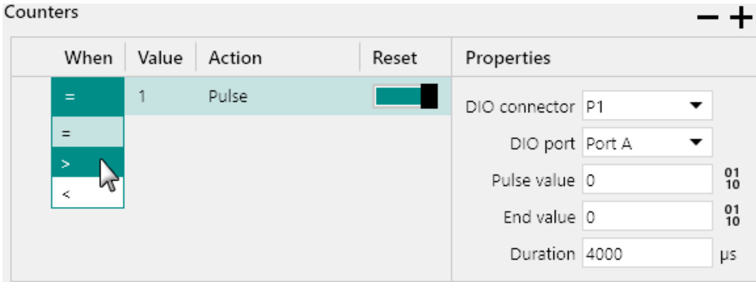
Click on the + button to add a counter to the table.

NOTE

A counter is automatically generated.

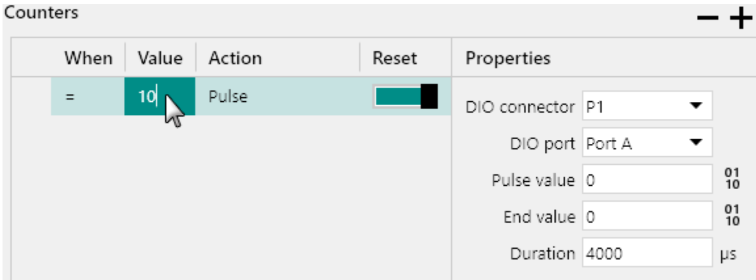
2 Specify the counter (in)equality

Click on the cell of the **When** column and select the equality or inequality to use in the counter using the provided drop-down list (< , = or >).



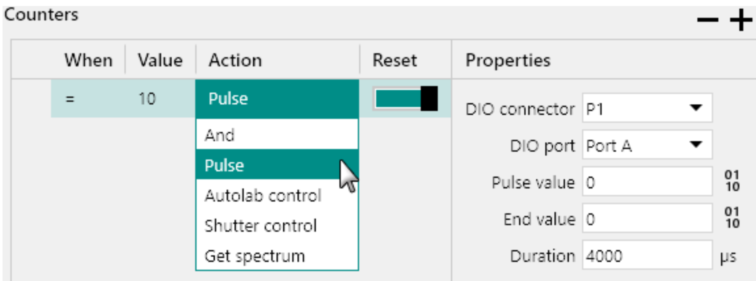
3 Specify the value

Specify the threshold value for the counter in the corresponding cell of the **Value** column.



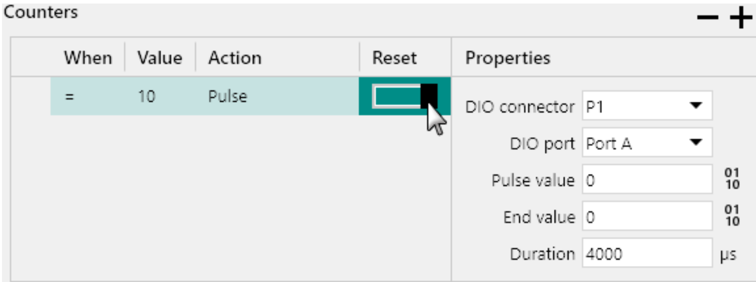
4 Specify the action

Click on the cell of the **Action** column and select the action to use in the counter using the provided drop-down list.



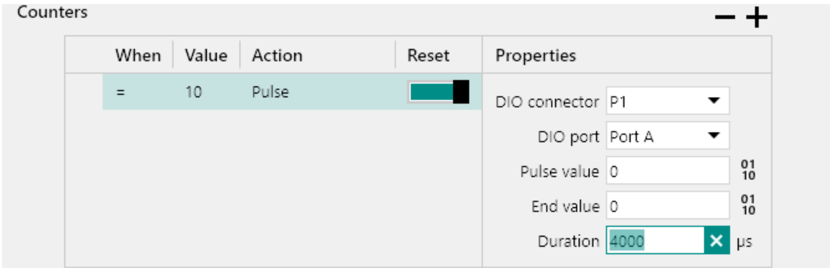
5 Set the reset property

Use the provided toggle to define if the counter should be reset after it is triggered.



6 Define the properties of the specified action

Use the provided properties frame to define the properties of the Action defined for the counter.



NOTE

To remove a counter from the table, select the row of the counter and click the **−** button above the **Counters** table.

9.4.2 Counter action - Pulse

The **Pulse** action can be used to send a TTL (Transistor-Transistor Logic) pulse to an external device when the condition defined for the counter is met. This pulse can be used to trigger the external device to perform a specific action.

The properties of the Pulse are defined in the dedicated frame, on the right-hand side of the **Counters** table *(see Figure 745, page 607)*.

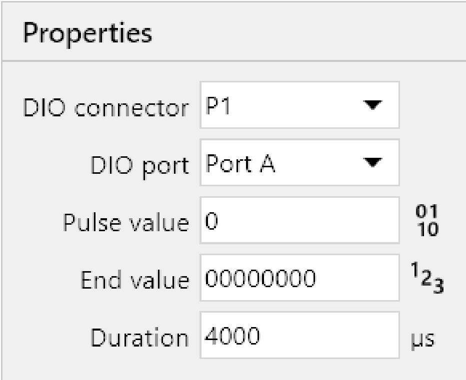


Figure 745 The Pulse properties are defined in the frame on the right-hand side of the Counters table

The following properties are available:

- **DIO connector (P1 or P2):** defines the DIO connector used to send the pulse.
- **Port (A, B or C):** defines the DIO port used to send the pulse.
- **Pulse value:** the decimal or binary expression of the 8 bit pulse state of the specified DIO port.
- **End value:** the decimal or binary expression of the 8 bit end state of the specified DIO port.
- **Duration (µs):** the duration of the pulse, in µs.

NOTE

It is possible to switch from binary expression to decimal expression and from decimal expression to binary expression by clicking the and buttons located next to the Pulse value and End value fields, respectively.

NOTE

More information on the DIO ports and connectors can be found in *Chapter 16.3.1.3*.

For example, using the settings specified in *Figure 746*, the following pulse will be generated from DIO connector P1, port B:

1. **From initial state to Pulse value:** the pulse will start from the initial state of the DIO port. It will then go to the **Pulse** value defined in the **Properties** frame. In this example, the **Pulse** value is *10000000* (or 128 in decimal).

2. **From Pulse value to End value:** after 10 ms, the DIO port will transition from the **Pulse** value to the **End** value. In this example, the **End** value is *00000000*.

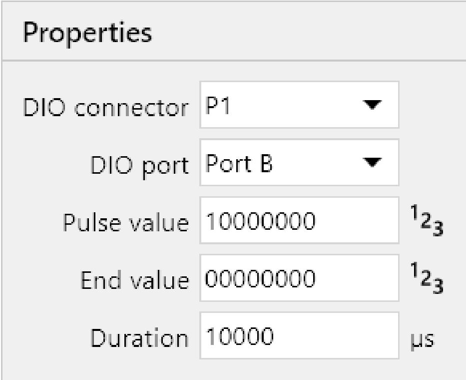


Figure 746 Example of a Pulse

9.4.3 Counter action - Autolab control

The **Autolab control** action can be used to set the properties of the instrument using an instance of the **Autolab control** command.

The properties of the Pulse are defined in the dedicated frame, on the right-hand side of the **Counters** table *(see Figure 747, page 608)*.

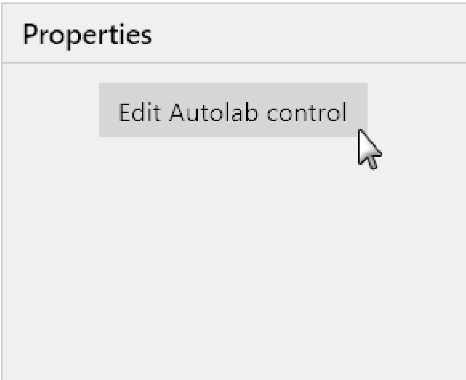


Figure 747 The Autolab control properties are defined in the frame on the right-hand side of the Counters table

Clicking the Edit Autolab control button opens the Autolab control editor *(see Figure 748, page 609)*.

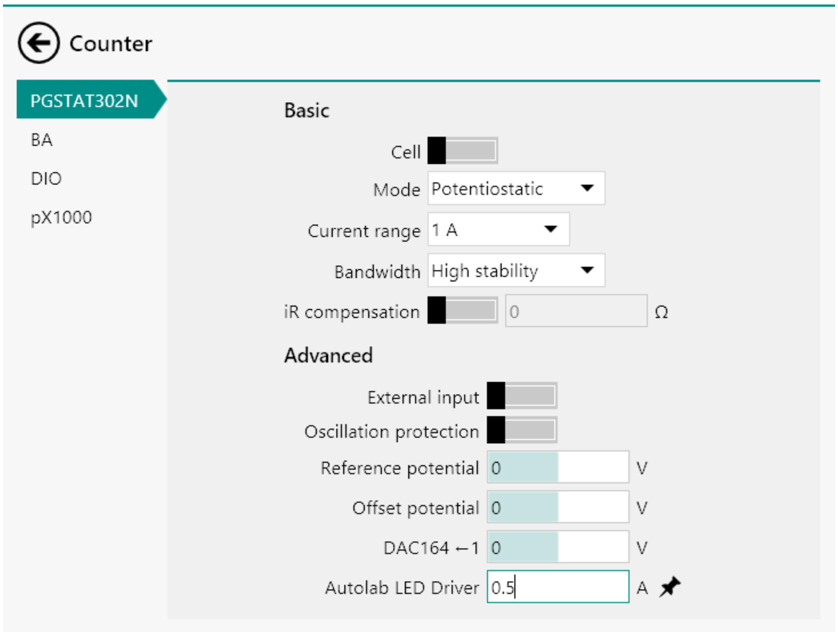


Figure 748 The Autolab control editor

NOTE

For more information on the **Autolab control** command, please refer to *Chapter 7.2.1*.

9.4.4 Counter action - Shutter control

The **Shutter control** action can be used to open or close the shutter of a connected Autolab or Avantes light source by setting the required DIO value on the specified connector.

The properties of the **Shutter control** action are defined in the dedicated frame, on the right-hand side of the **Counters** table *(see Figure 749, page 609)*.

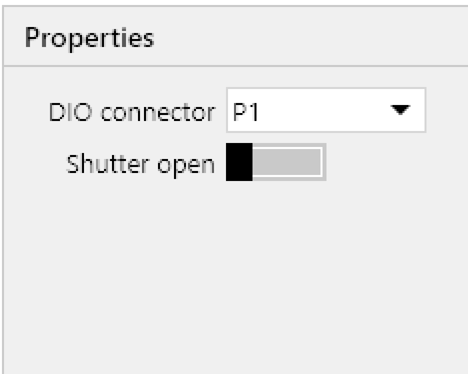


Figure 749 The properties of the Shutter control action

The following properties are available:

- **DIO connector (P1 or P2):** defines the DIO connector used to control the light source shutter.
- **Shutter open:** defined the state of the shutter, using the provided toggle. When the shutter is off, no light comes out of the light source. When the shutter is on, light can come out of the light source.

NOTE

The light source shutter will remains in the specified state until changed.

CAUTION

The **Shutter control** action only works with light sources that support TTL control that are used with this mode enabled.

9.4.5 Counter action - Get spectrum

The **Get spectrum** action can be used to synchronize the acquisition of a spectrum on a connected Autolab or Avantes spectrophotometer by sending a TTL pulse of required length.

The properties of the **Get spectrum** action are defined in the dedicated frame, on the right-hand side of the **Counters** table *(see Figure 750, page 610)*.

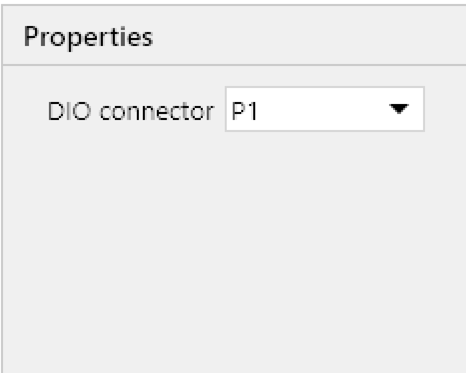


Figure 750 The property of the Get spectrum action

The following property are available:

- **DIO connector (P1 or P2):** defines the DIO connector used to send the trigger.

CAUTION

The **Get spectrum** action will also trigger the opening of the shutter of the light source connected to the same DIO connector, if this light source support TTL control and if this mode is enabled.

9.4.6 Combining counters

It is possible to define more than one counter in the Counters table. Depending on how the counters are defined, it is possible to arrange two or more counters in two different ways:

- **OR arrangement:** each counter is defined as a standalone counter. The action defined for each of them is triggered whenever the corresponding threshold value is reached. This corresponds to a *OR* logical operator. Each counter will trigger a specific action separately.
- **AND arrangement:** the two or more counters can be joined with a *AND* action in order to trigger a single action when each of the involved counters reaches its corresponding threshold value.

Figure 751 shows an example of four counters. The first counter is executed once, and it changes instrumental properties at the fifth point. The second counter is executed every 10 points. When this happens, the counter is reset and the instrumental properties are adjusted again. The third counter triggers the fourth counter after the fifth point. The action defined for the fourth counter is executed every 10 points. An Autolab control event is used for this counter.

Using this combination, the second counter and the fourth counter are used to change instrumental properties every 10 points. However, both counters are offset by five points.

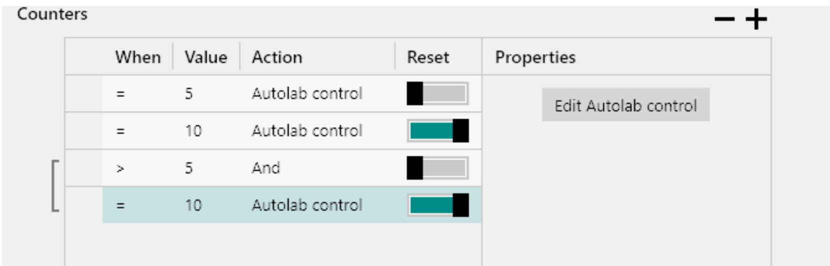


Figure 751 Multiple counters

> **NOTE**
>
> The third and fourth counter shown in *Figure 751* are connected by a grey line on the left-hand side of the table, indicating that both counters have a **AND** relationship.

9.5 Plots

The plots define which how measured or calculated signals are plotted during a measurement. Two plots groups are shown in this section *(see Figure 752, page 612)*:

- **Default plots:** a table containing a list of preconfigured plots.
- **Custom plots:** a table that can be used to define custom plots.

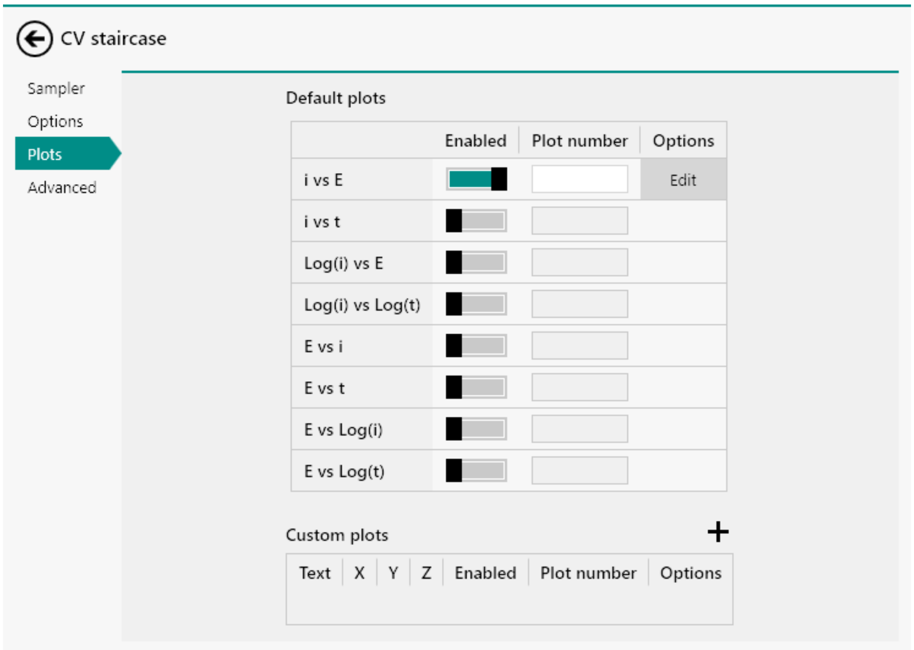


Figure 752 The Plots define how the measured data is displayed during a measurement

> **NOTE**
>
> The plots listed in the **Default** plots table depend on the measurement command and on the signal defined in the **Sampler**.

NOTE

Click the Edit button will open the plot Properties screen *(see Chapter 9.5.3, page 615)*.

9.5.1 Default plots

Default plots are defined in the dedicated table, in the **Default** plots subpanel *(see Figure 753, page 613)*.

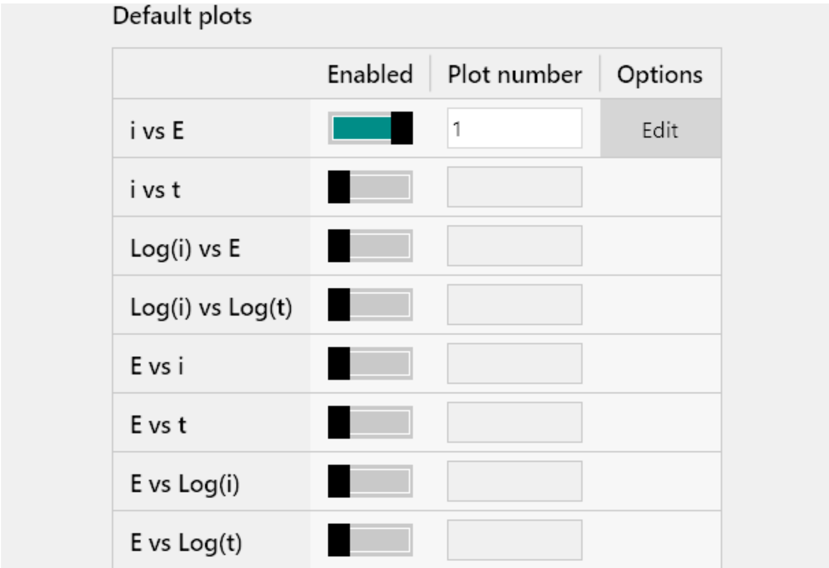


Figure 753 The Default plots table

Four properties are defined per plot:

- **Name:** the name of the default plot.
- **Enabled:** specifies if the default plot should be used during the measurement, using the provided toggle.
- **Plot number:** defines the plot number. This value is an integer. Plots that have the same plot number will be displayed as an overlay during the measurement. If this value is unspecified, the plot will be assigned a number during the measurement.
- **Options:** defines the plot options for this plot. These options are defined in a dedicated editor.

9.5.2 Custom plots

Custom plots are defined in the dedicated table, in the **Custom** plots subpanel *(see Figure 754, page 614)*.

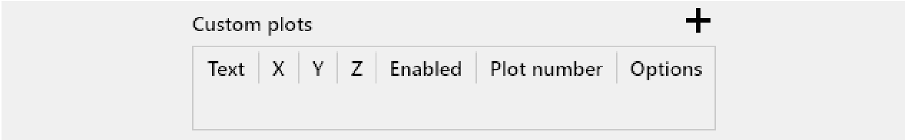
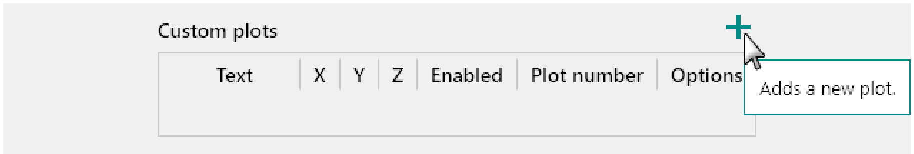


Figure 754 The Custom plots table

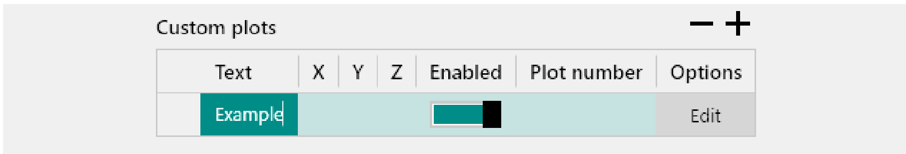
1 **Add a plot to the list**

Click on the **+** button to add a plot to the table.



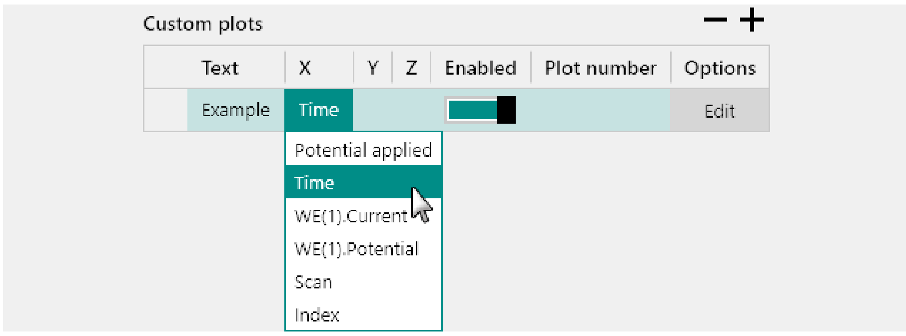
2 **Specify the name of the plot**

Specify the name of the custom plot by typing the name in the first cell of the **Text** column.



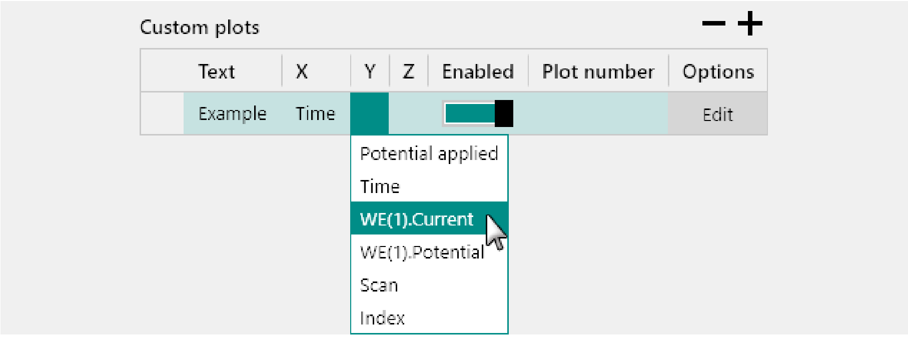
3 **Specify the signal for the X axis**

Click the first available cell in the **X** column and select the signal to plot on the X axis using the provided drop-down list.



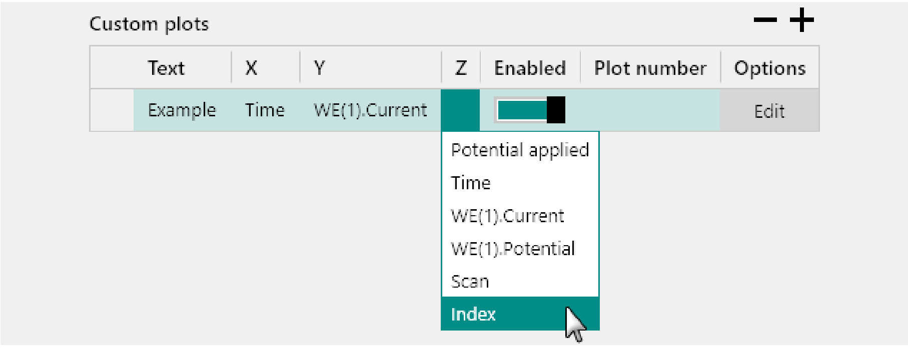
4 **Specify the signal for the Y axis**

Click the first available cell in the **Y** column and select the signal to plot on the Y axis using the provided drop-down list.



5 **Specify the signal for the Z axis**

Click the first available cell in the **Z** column and select the signal to plot on the Z axis using the provided drop-down list.



NOTE

To remove a plot from the table, select the row of the plot and click the ━ button above the **Custom plots** table.

9.5.3 Plot options

To edit the plot options, the button Edit located next to each enabled plot or each custom plot is provided *(see Figure 755, page 616)*.

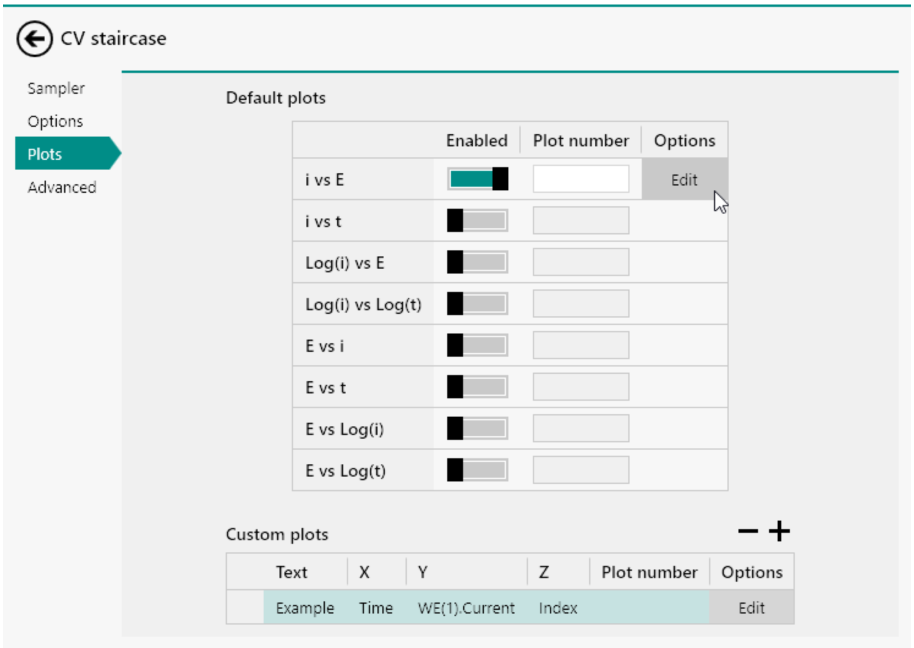


Figure 755 Editing the plot options

Clicking this button displays the plot options editor screen. The controls on this screen can be used to define the plot settings of the corresponding plot *(see Figure 756, page 617)*.

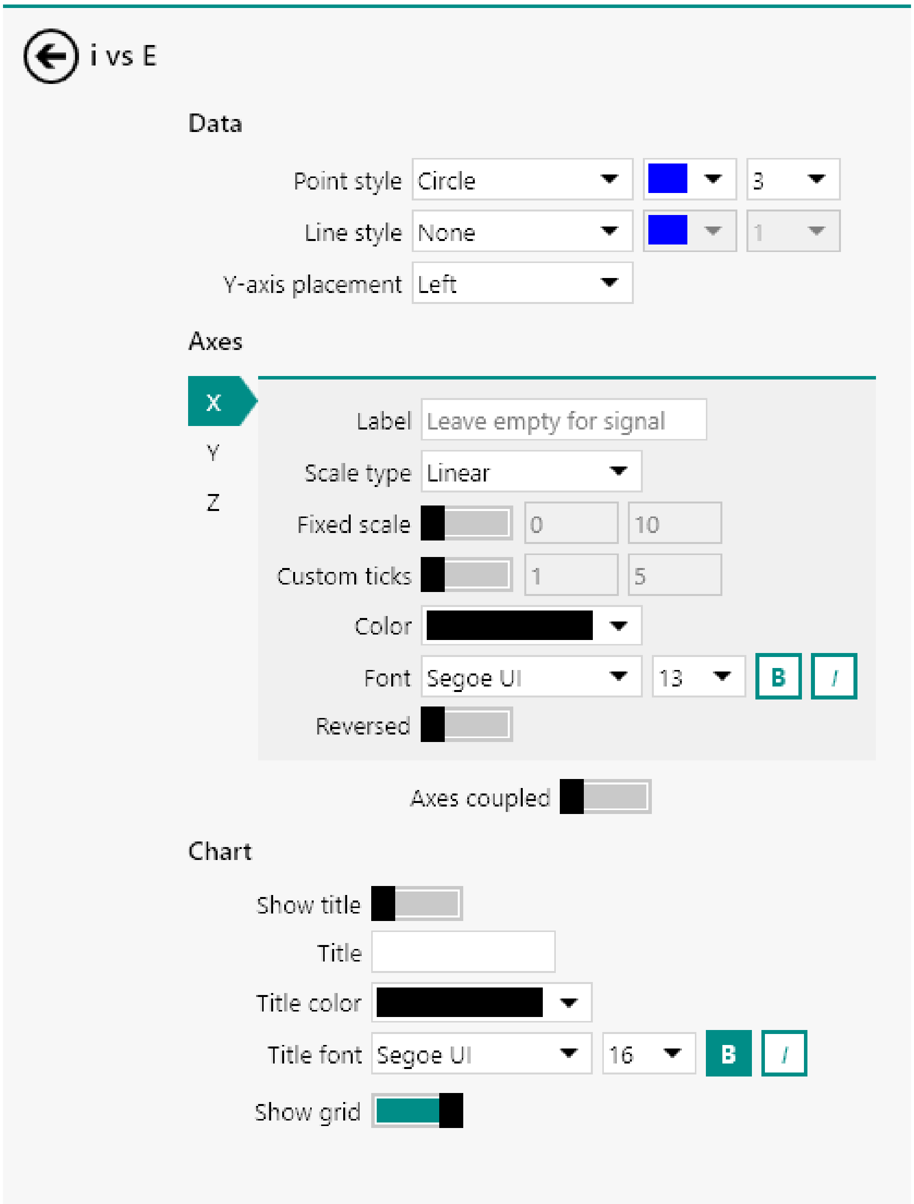


Figure 756 The plot Properties screen

The plot options are defined in three sub-panels:

- **Data:** these are properties associated with the data points.
- **Axes:** these are properties associated with the plot axes.
- **Chart:** these are properties of the whole chart not specifically associated with data or axes.

Clicking the ⊖ button closes the screen and returns to the procedure editor.

9.5.3.1 Data option

The **Data** sub-panel can be used to defined general properties of the data *(see Figure 757, page 618)*.

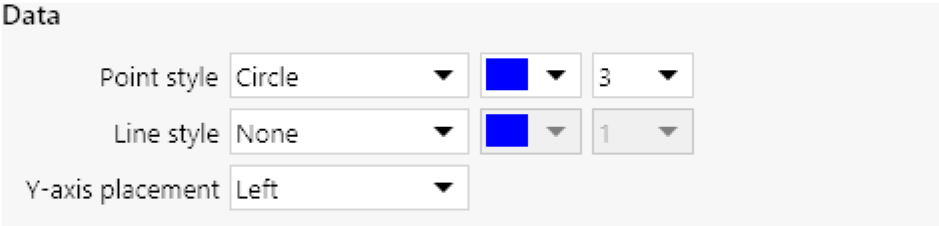


Figure 757 The Data sub-panel

The following properties can be edited in the **Data** sub-panel:

- **Point style:** defines the point style, color and size used by the plot, using dedicated drop-down lists.
- **Line style:** defines the line style, color and size used by the plot, using dedicated drop-down lists.
- **Y-axis placement:** specify the location of the Y axis using the provided drop-down list. The choice is provided between left and right.

9.5.3.2 Axes option

The **Axes** sub-panel can be used to defined general properties of the plot axes *(see Figure 758, page 618)*.

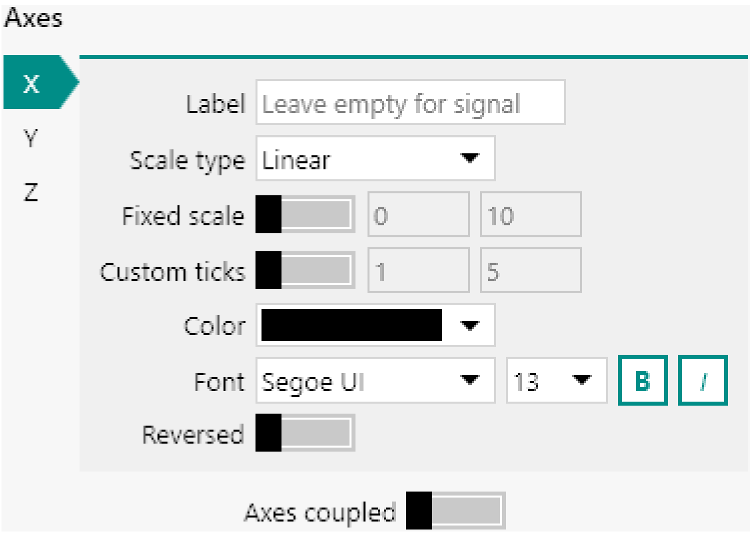


Figure 758 The Axes sub-panel

The following properties can be edited in the **Axes** sub-panel for each individual axis:

- **Label:** defines the label of the axis. When this field is left empty, the name of the signal plotted on this axis will be used instead.
- **Scale type:** defines the scale type of the axis, using the provided drop-down list. The choice is provide between linear and logarithmic.

- **Fixed scale:** defines if an automatic or fixed scaling should be used for the axis, using the provided toggle. When a fixed scale is used, the minimum and maximum value for the axis can be specified in the provided field.
- **Custom ticks:** defines if major and minor ticks should be automatically plotted or if major and minor ticks should be defined manually, using the provided toggle. When custom ticks are used, the distribution for major and minor ticks can be specified in the provided field.
- **Color:** the color for the axis. The color can be specified using the provided drop-down list.
- **Font:** the font used for the axis. The font type and size can be specified using dedicated drop-down lists. The format of the title can be edited by toggling the bold formatting or italic formatting on or off using the dedicated buttons.
- **Reversed:** defines if the axis is reversed or not, using the provided toggle.

A common property is available for all the axes:

- **Axes coupled:** defines if the scaling used on the X, Y and Z axes should be the same using the provided toggle.

9.5.3.3 Chart option

The **Chart** sub-panel can be used to defined general properties of a plot that are not directly associated to the data points or the plot axes *(see Figure 759, page 619)*.

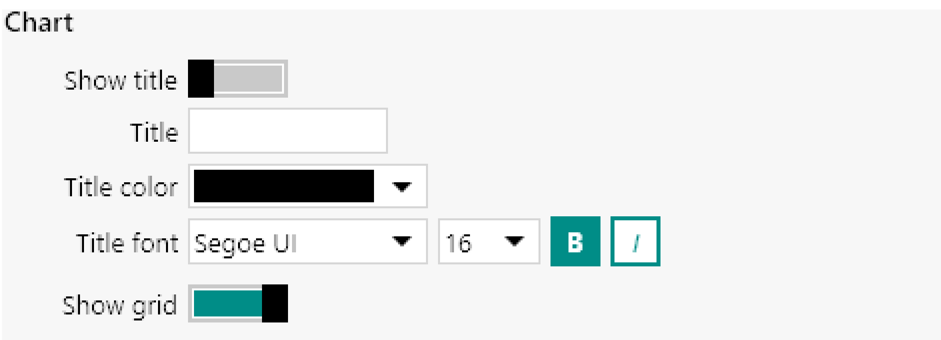


Figure 759 The Chart sub-panel

The following properties can be edited in the **Chart** sub-panel:

- **Show title:** a toggle which can be used to show or hide the title.
- **Title:** the title of the plot. This title is displayed if the **Show title** property is on.
- **Title color:** the color for the title. The color can be specified using the provided drop-down list.

- **Title font:** the font used for the title. The font type and size can be specified using dedicated drop-down lists. The format of the title can be edited by toggling the bold formatting or italic formatting on or off using the dedicated buttons.
- **Show grid:** enables or disables the chart grid, using the provided toggle.

9.6 Automatic integration time

The **Automatic integration time** option specifies which of the available integration time constants can be used by the measurement command.

NOTE

This option is only available for instruments that are fitted with the optional **FI20** module or the **on-board integrator**. For more information, please refer to *Chapter 16.3.2.11*.

When this option is used, the instrument will automatically select the most suitable integration time constant.

Automatic integration time settings are defined in the dedicated table, in the **Automatic integration time** sub-panel *(see Figure 760, page 620)*.

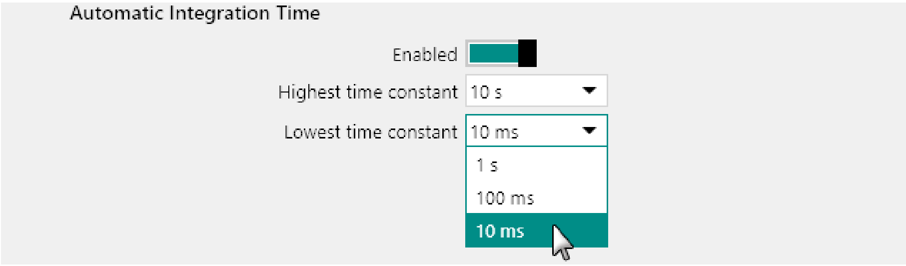


Figure 760 The Automatic integration time option is defined in a dedicated sub-panel

Three properties can be specified for each working electrode:

- **Enabled:** a toggle used to set the Automatic integration time on or off.
- **Highest time constant:** defines the highest possible time constant, using the provided drop-down list.
- **Lowest time constant:** defines the lowest possible time constant, using the provided drop-down list.

9.7 Value of Alpha

For the **CV staircase** command and the **LSV staircase** command, the *Alpha value* advanced property is available *(see Figure 761, page 621)*.

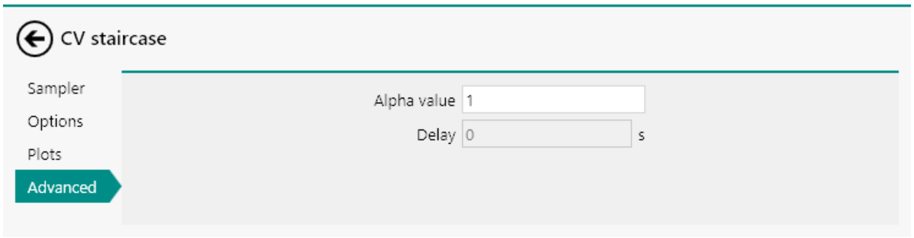


Figure 761 The Alpha value property is available for the CV staircase and LSV staircase command

The *Alpha value* can have a value between 1 and 0.

The *Alpha value* represents the fraction of the interval time, between two consecutive potential steps, at which the WE(1).Current signal is sampled. Its default value is 1, which means that the current is measure in the last quarter of the interval time. Through a careful specification of the Alpha value, the response recorded during a staircase cyclic voltammetry measurement or a linear sweep voltammetry measurement can be compared, in first approximation, to the response measured using a linear scan. For a reversible system, a value of 0.3 is suitable for comparing a staircase measurement with a linear scan measurement.

The difference between the normal sampling procedure (using a *Alpha value* of 1) and a sampling procedure using a *Alpha value* smaller than 1 is represented in *Figure 762* and *Figure 763*, schematically.

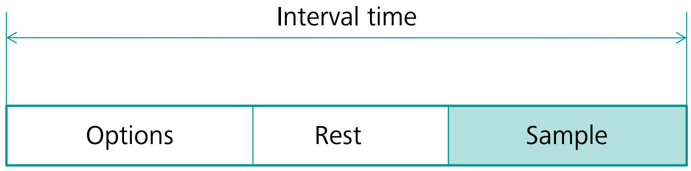


Figure 762 The normal sampling procedure

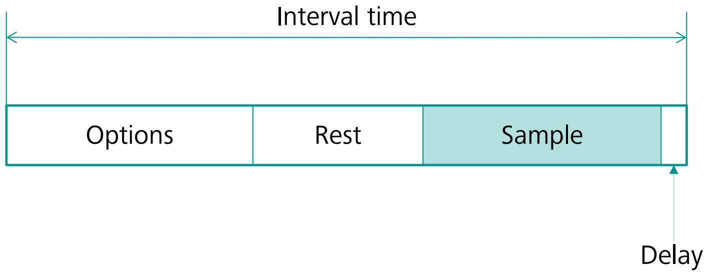


Figure 763 The sampling procedure with a Alpha value < 1

When a *Alpha value* smaller than 1 is used, a delay is added to the end of the interval time, in order to shift the sampling segment towards the front of the interval time. The value of the applied delay, in s, in updated in the field below the input field for the *Alpha value* *(see Figure 764, page 622)*.

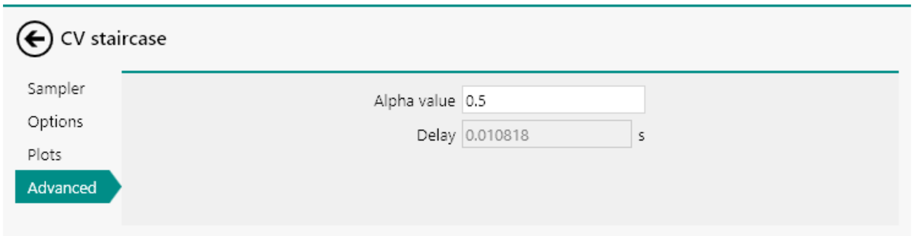


Figure 764 The Delay value is automatically updated when the Alpha value is modified

NOTE

The actual delay depends on the interval time.

NOTE

For more information on the *Alpha value* property, please refer to M. Saralthan, R.A. Osteryoung, J. Electroanal. Chem. 222, 69 (1987).

10 Procedure editor

The **Procedure editor** is the main frame in NOVA. This part of the interface provides the tools required to edit, modify or create procedures. New commands can be added to a procedure, commands can be removed or disabled and links or groups can be created or removed in order to further customize the procedure setup.

This chapter explains the different tools provided in the **Procedure editor** frame and how these tools can be used to build procedures in NOVA. The following concepts are explained in this chapter:

1. New procedure *(see Chapter 10.1, page 624)*
2. Global options *(see Chapter 10.2, page 626)*
3. End status Autolab *(see Chapter 10.3, page 630)*
4. Command tracks *(see Chapter 10.4, page 631)*
5. Procedure wrapping *(see Chapter 10.5, page 632)*
6. Procedure zooming *(see Chapter 10.6, page 633)*
7. Command groups *(see Chapter 10.7, page 634)*
8. Enabling and disabling commands *(see Chapter 10.8, page 637)*
9. Adding and removing commands *(see Chapter 10.9, page 639)*
10. Moving commands *(see Chapter 10.10, page 647)*
11. Stacking commands *(see Chapter 10.12, page 653)*
12. Linking commands *(see Chapter 10.13, page 657)*
13. My commands *(see Chapter 10.14, page 671)*

NOTE

Most of the tools provided in the **Procedure editor** are reserved for advanced users. Before using this tools, it is recommended to carefully read this chapter.

10.1 Creating a new procedure

To create a new procedure, click the New procedure button in the **Actions** panel on the dashboard *(see Figure 765, page 624)*.

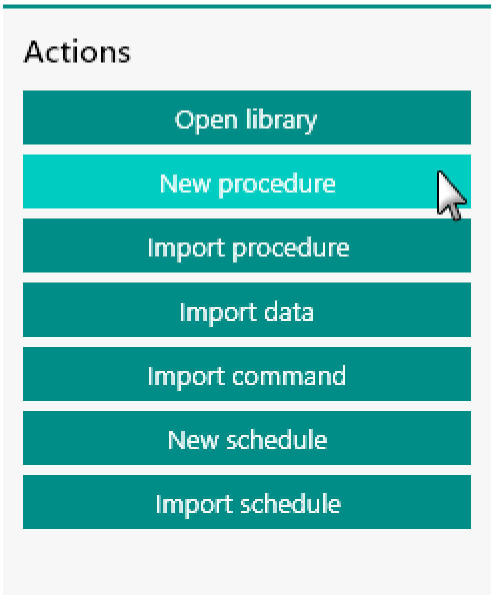


Figure 765 Click the New procedure button to create a new procedure

A new tab will be created, providing an empty procedure editor that can be used to create a customized procedure *(see Figure 766, page 624)*.

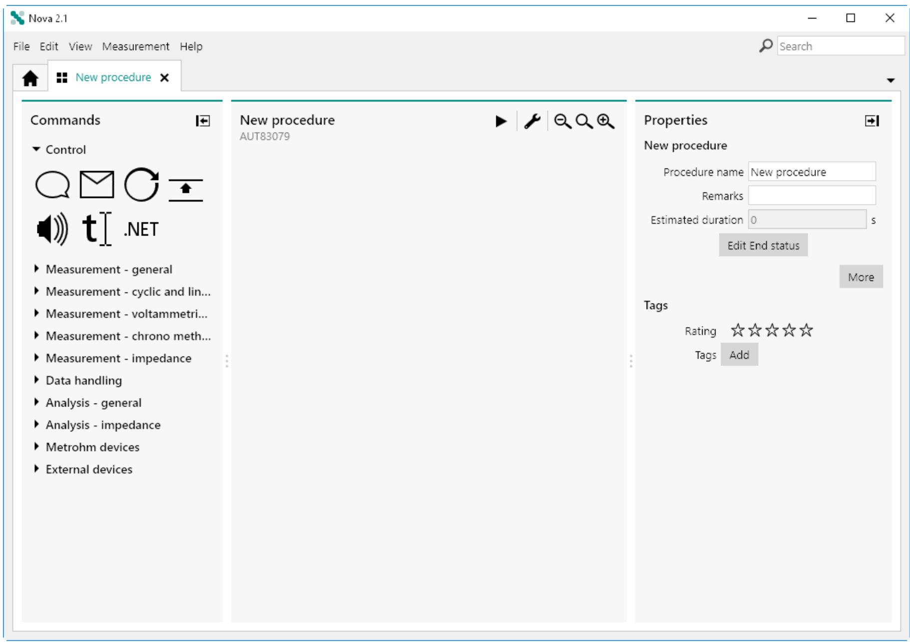


Figure 766 A new tab is created

The new procedure editor displays the three main panels:

- **Commands panel:** this panel shows all the available commands that can be used to create a procedure.
- **New procedure panel:** this empty panel provides the environment to create a new procedure.
- Properties panel: this panel shows the properties of the new procedure or the properties of a selected command in the procedure.

The **Properties** panel shows the properties of the procedure. These properties are displayed when no command is located in the procedure or when no command is selected if commands are located in the procedure *(see Figure 767, page 625)*.

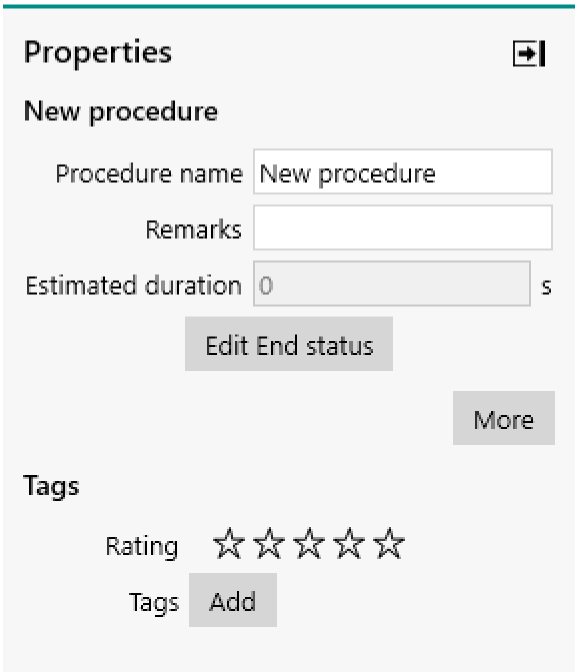


Figure 767 The properties of the new procedure

The following properties are available in the **Properties** panel:

- **Procedure name:** the name of the procedure (default: New procedure)
- **Remarks:** a remarks field that can be used to add comments to the procedure.
- **Estimated duration:** this read-only value shows the estimated duration of the procedure. This value is updated whenever commands are added to the procedure editor.

These properties can be edited for bookkeeping purposes, and when commands are added to the procedure, the **Estimated duration** is updated. *Figure 768* shows the properties of the default Chrono amperometry (Δt > 1 ms) procedure.

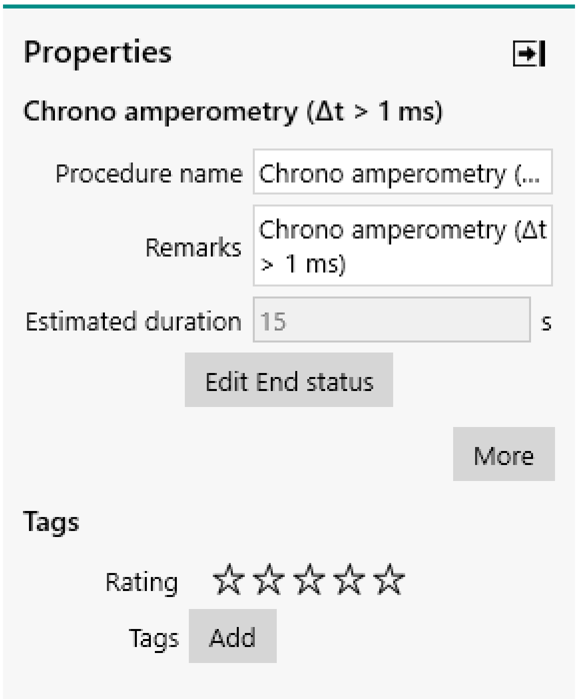


Figure 768 Procedure properties of the default Chrono amperometry procedure

Additional buttons are available to edit the of the procedure options or the End status of the instrument. Please refer to *Chapter 10.2* and *Chapter 10.3* for more information.

The controls provided in the **Tags** sub-panel can be used to assign a rating and tags to the procedure. This provides further options for bookkeeping purposes. More information on the use of the rating and tags can be found in *Chapter 6.8*.

10.2 Global options and global sampler

For all procedures, it is possible to define global options and global sampler settings. If defined, these settings will be used for all the commands unless overruled for a specific command in the procedure. To define these settings, click the More button in the **Properties** panel *(see Figure 769, page 627)*.

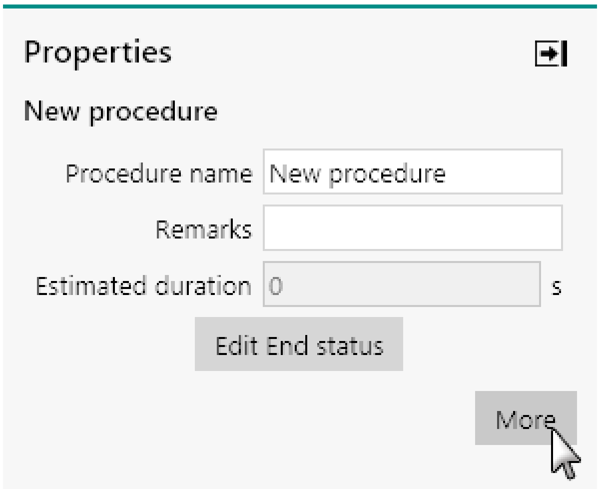


Figure 769 Click the More button to open define the global options and global sampler

A new screen will be displayed, as shown in *Figure 770*, showing two different sections:

- **Sampler:** the settings in this section define the global sampler settings *(see Figure 770, page 627)*.
- **Options:** the settings in this section define the global options settings *(see Figure 771, page 628)*.

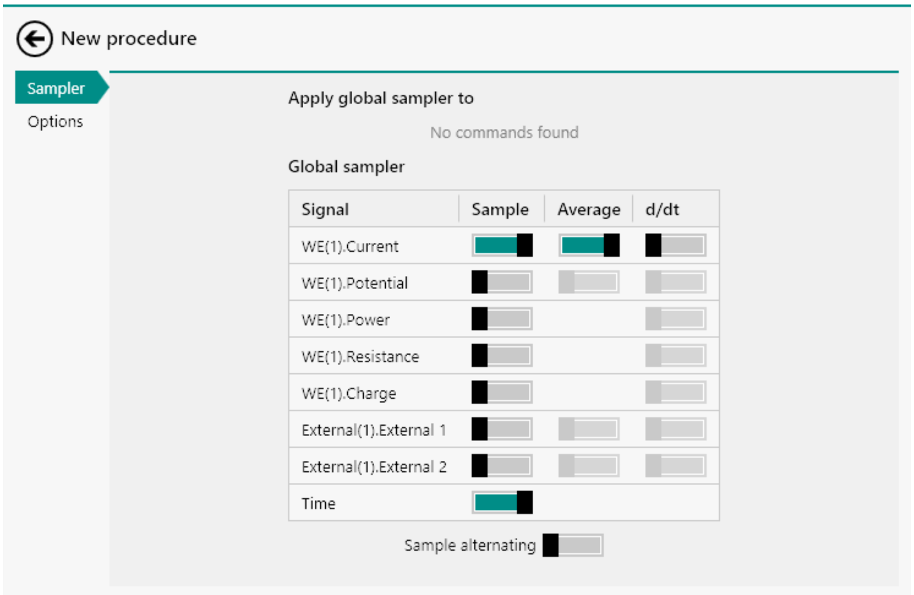


Figure 770 The global sampler settings

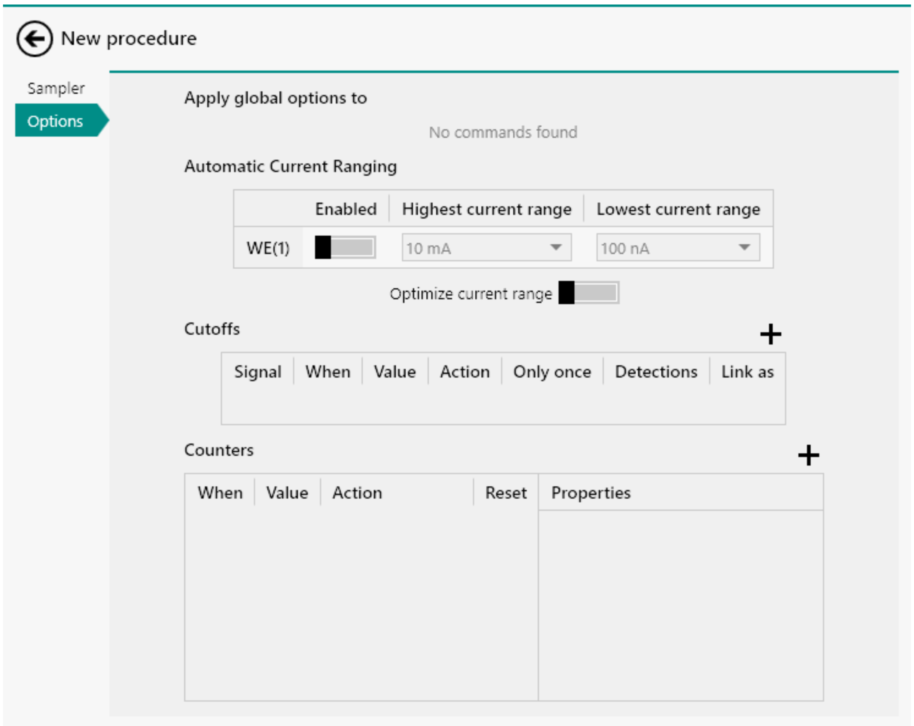


Figure 771 The global options settings

The settings in **Sampler** section can be used to specify which signals have to be sampled during the procedure. More information about the **Sampler** can be found in *Chapter 9.1*.

If commands are already located in the procedure, these commands will be displayed in the Apply global sampler to subsection. Using the provided checkboxes, it is possible to define on which the global sampler settings need to be applied *(see Figure 772, page 629)*.

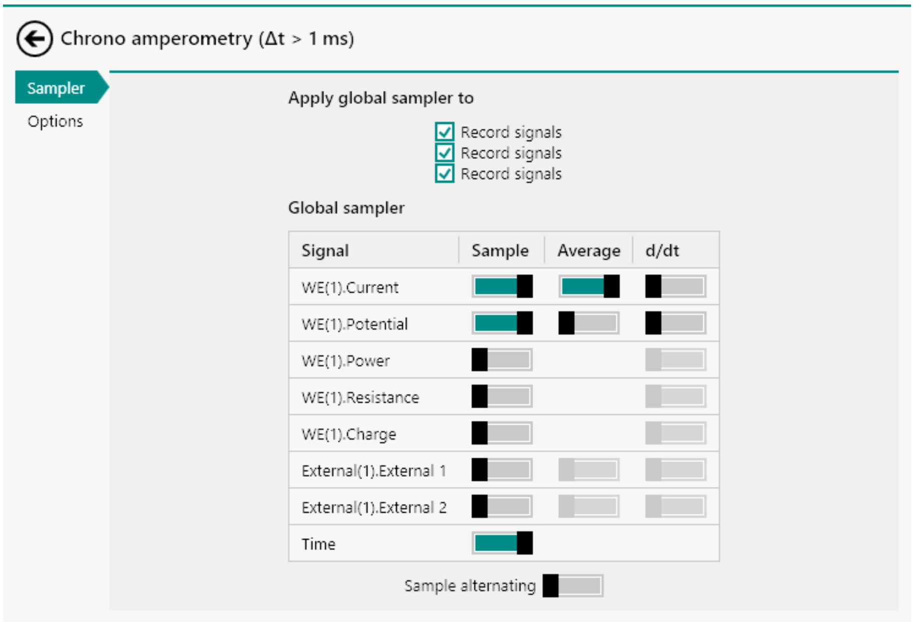


Figure 772 It is possible to define on which commands the global sampler needs to be applied

The same applies to the global options. Using the provided checkboxes, it is possible to define on which the global options settings need to be applied *(see Figure 773, page 629)*.

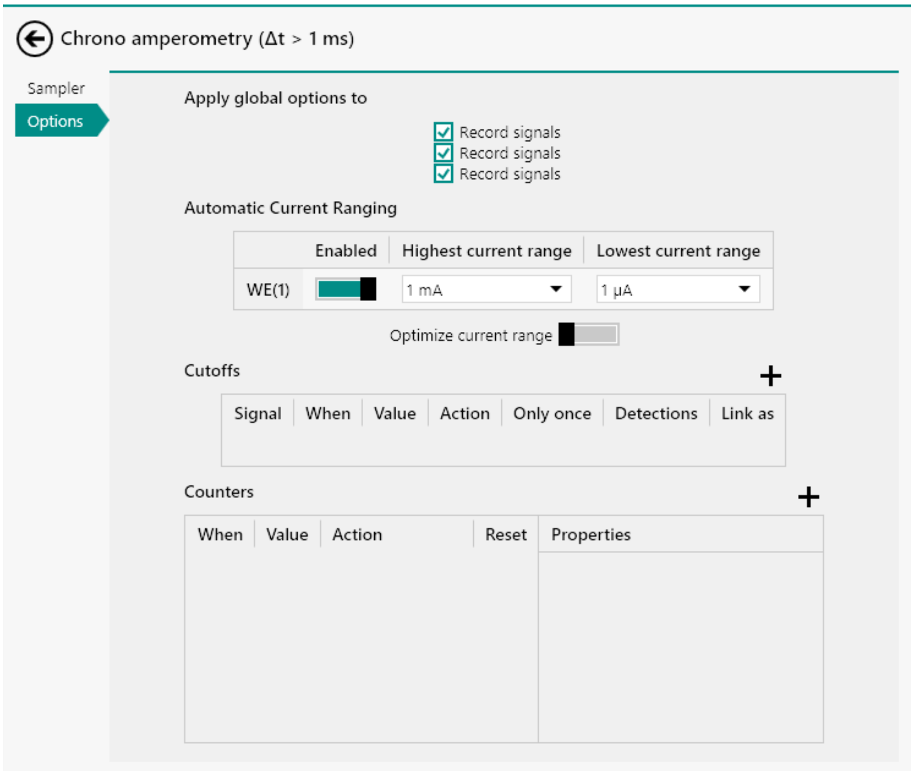


Figure 773 It is possible to define on which commands the global options needs to be applied

10.3 End status Autolab

The End status Autolab is an option that can be defined for all procedures to define the how the Autolab system should behave when the procedure stops. The settings defined in the End status Autolab will be activated when the following events occur:

- The procedure stops normally
- The procedure is stopped by the user
- The procedure is stopped by a cutoff

To edit the End status Autolab, click the button *(see Figure 774, page 630)*.

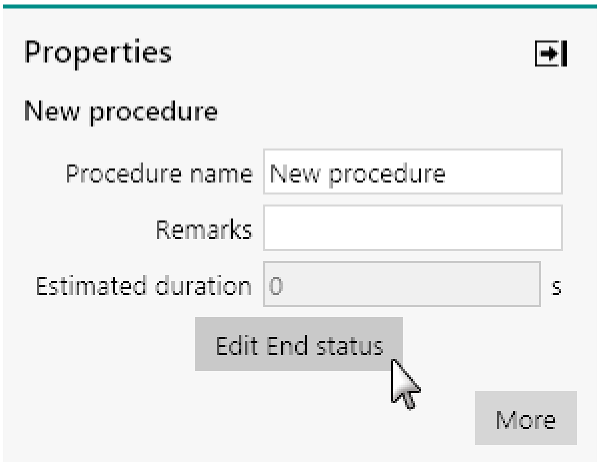


Figure 774 Click the Edit End status button to open the End status Autolab editor

The End status Autolab screen will be displayed. In this screen all of the instrumental settings and module settings can be specified *(see Figure 775, page 631)*.

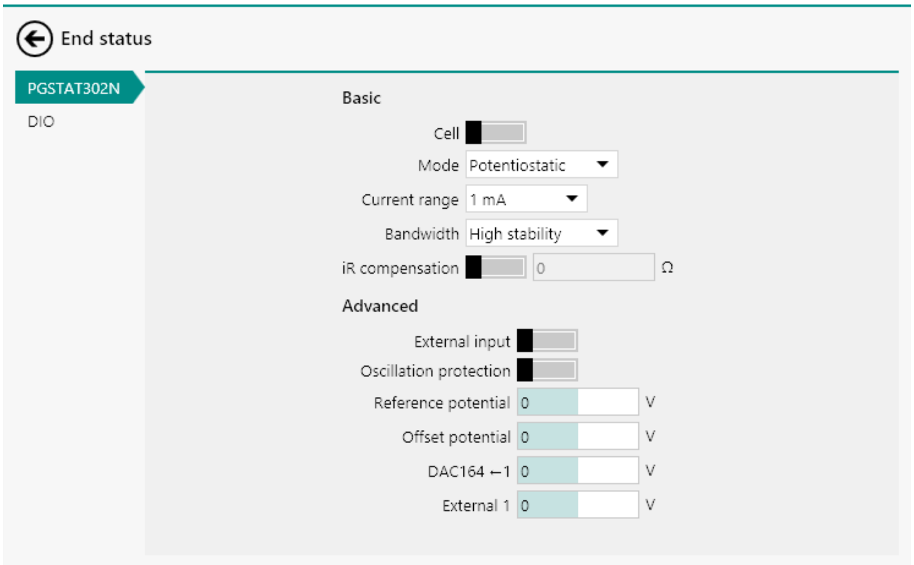


Figure 775 The End status Autolab editor

NOTE

The End status Autolab editor is the same as the one for the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

10.4 Procedure tracks

Procedures in NOVA consist of at least one main **Track** of commands, which are executed in sequence. Each command can be used to create a sub-track in which additional commands can be located. Commands located in each sub-track are executed sequentially when the parent command located in the main track is executed.

A simple example is provided by the default *Hydrodynamic linear sweep* procedure, available from the **Default** procedures, in the **Library** (please refer to *Chapter 8.2.4* for a complete description of this procedure). This procedure contains one main track in which a **Repeat** command is located. This command is configured in the *Repeat for multiple values* mode. The rotation rates required for this measurement are pre-defined in the **Repeat** command *(see Figure 776, page 632)*.

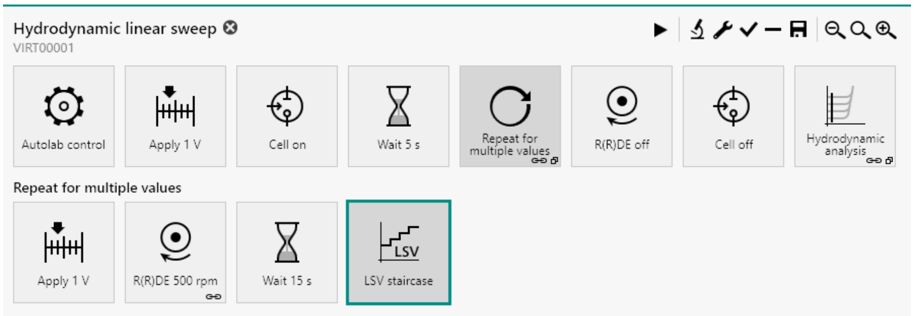


Figure 776 The Hydrodynamic linear sweep voltammetry procedure

Additional command are located in the sub-track of the **Repeat** command. These commands are visible when the **Repeat** command is selected *(see Figure 776, page 632)*.

When the procedure is executed, the four first commands located in the main track are executed in sequence. When the **Repeat** command is executed, the four commands located in the sub-track are executed in sequence and repeated six times as defined by the **Repeat** command. When all six repetitions are completed, the procedure resumes the main track of the procedure.

10.5 Procedure wrapping

The procedure editor frame has a limited width. When a procedure track has more commands than can be displayed in a single line, the software will wrap the track and display the commands on multiple lines. In the example below, the procedure has a single track, wrapped on two lines. The last **Cell** command is located on the second line *(see Figure 777, page 632)*.

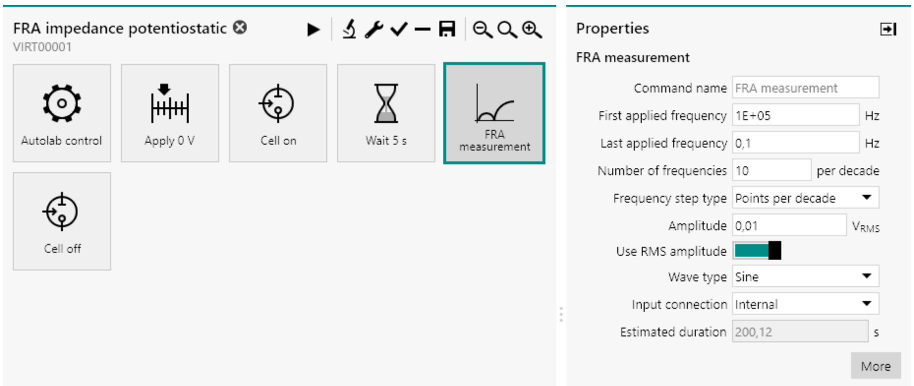


Figure 777 Long tracks are wrapped on several lines if needed

If the NOVA window is resized, or if the **Properties** panel on the right-hand side or the **Command** panel on the left hand side are resized or collapsed, the procedure editor will be readjusted, and if possible, the commands will be displayed on a single line *(see Figure 778, page 633)*.

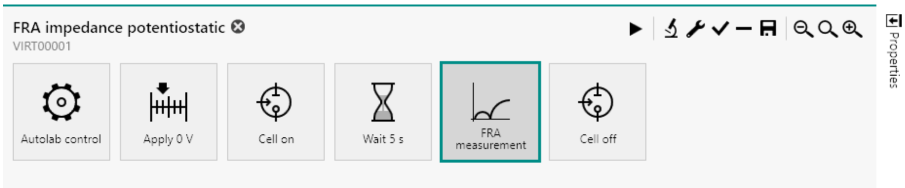


Figure 778 Commands are relocated to a single line when enough room is available

10.6 Procedure zooming

The procedure editor frame has a limited width. If needed, the size of the items in the procedure editor frame can be adjusted with the controls located in the top right corner of the frame *(see Figure 779, page 633)*.

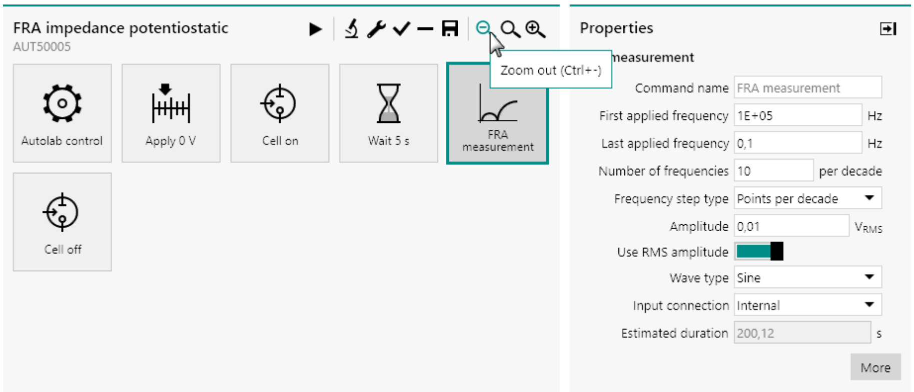


Figure 779 Zoom controls are provided in the procedure editor

Using this function will either scale the size of the items and the text up or down (between 200 % and 50 % of the original size), as shown in *Figure 780*.

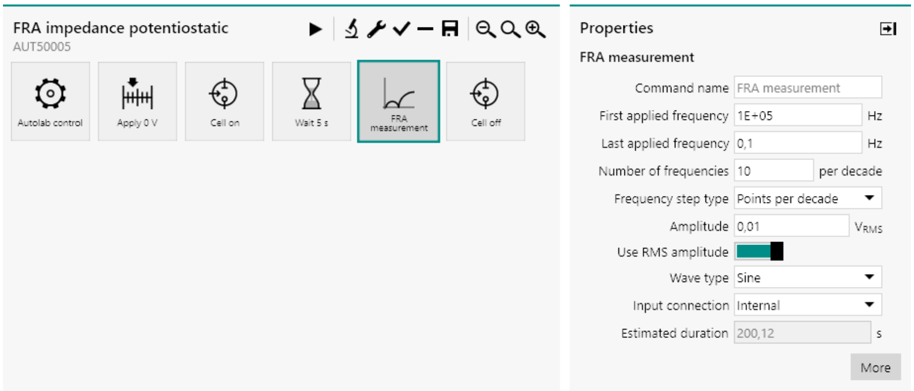


Figure 780 Zooming the procedure editor out

The following zooming controls are available:

- **Zoom out:** decreases the scaling of the items and text shown on screen. The button or **[CTRL]** + **[-]** keyboard shortcut can be used to do this.
- **Zoom to 100%:** resets the scaling of the items and text shown on screen to the default size. The button or **[F4]** keyboard shortcut can be used to do this.
- **Zoom in:** increases the scaling of the items and text shown on screen. The button or **[CTRL]** + **[=]** keyboard shortcut can be used to do this.

10.7 Command groups

Command groups can be created in a procedure in order to structure a complex procedure. Grouping commands allows to group commands in a container command in the main track (or a sub-track). This is useful for hiding commands that are not directly important to the measurement, or it can simply be used to group commands that have similar functionality. A group also provide the possibility to relocate several commands in the procedure at once.

10.7.1 Grouping commands

It is possible to group commands located in a procedure. Creating groups provides the benefit of locating command into groups which can then be moved in the procedure editor. Grouping commands can also be used to create a clear procedure structure, which is especially useful for complex procedures. Grouped commands are replaced by a **Group** item in the procedure editor in which the grouped commands are relocated in the original order.

To group commands, select the commands in the procedure editor and click the button located in the top right corner of the procedure editor. It is also possible to use the keyboard shortcut consisting of the **[CTRL]** + **[G]** combination *(see Figure 781, page 634)*.

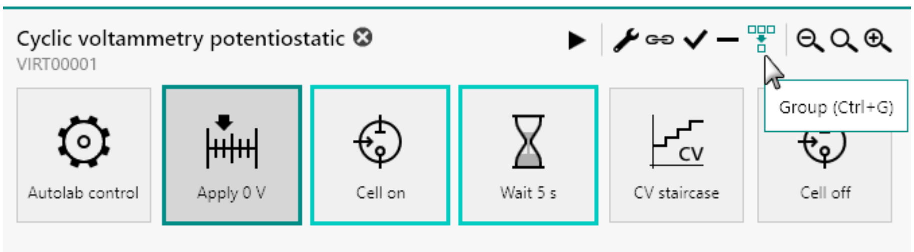


Figure 781 Grouping commands

The grouped commands will be replaced in the procedure track by the Group item. Clicking this item in the procedure editor reveals the grouped command in the track below *(see Figure 782, page 635)*.

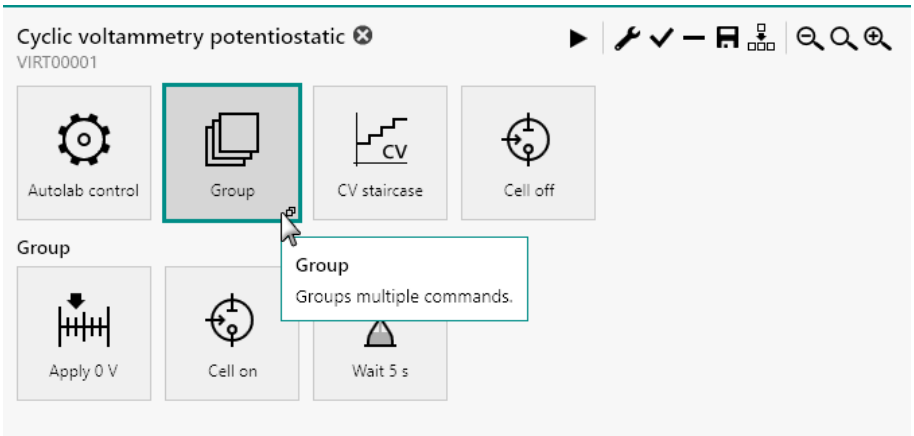
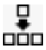


Figure 782 The grouped commands are visible when the group is selected

10.7.2 Ungrouping commands

It is possible to ungroup commands that are located in a group. Ungrouping commands removes the group from the procedure editor and promotes the involved commands to the next available procedure track above the group.

To ungroup grouped commands, select the group in the procedure editor and click the  button located in the top right corner of the procedure editor. It is also possible to use the keyboard shortcut consisting of the **[SHIFT]** + **[CTRL]** + **[G]** combination *(see Figure 783, page 635)*.

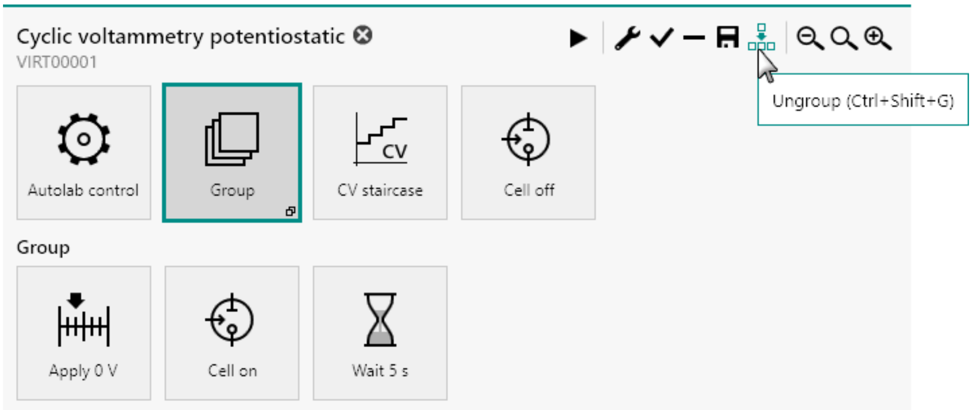


Figure 783 Ungrouping grouped commands

The group will be removed from the procedure editor and the grouped commands will be restored in the track above the former group in the order in which these commands were located in the group *(see Figure 784, page 636)*.

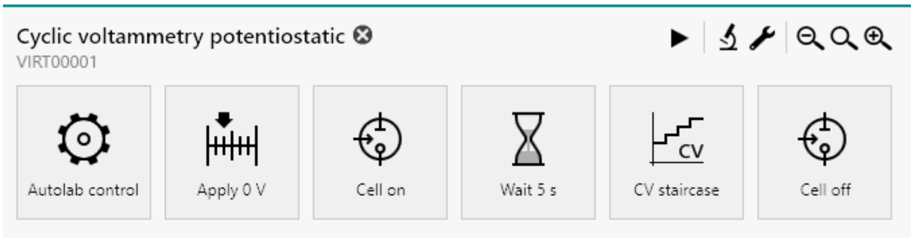


Figure 784 The ungrouped commands are restored in the procedure editor

10.7.3 Renaming groups

For bookkeeping purposes, it is possible to change the name of a group of commands. This is useful when creating complex procedures as each group can be given a relevant name. To change the name of a group in the procedure, select the group and change the name in the **Properties** panel located on the right-hand side of the screen *(see Figure 785, page 636)*.

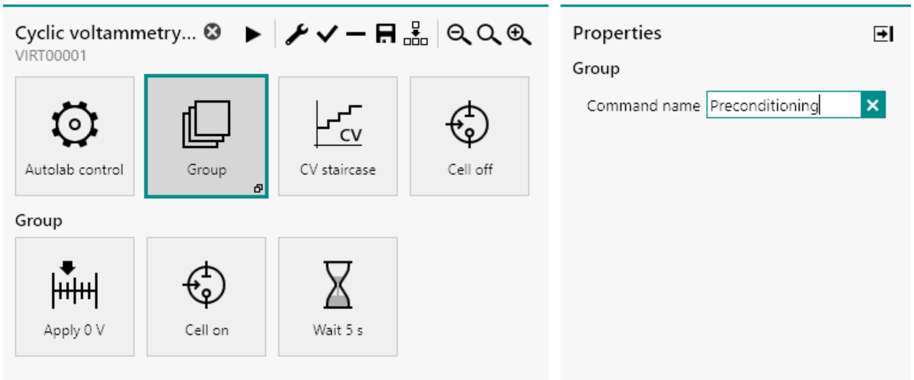


Figure 785 Changing the name of a group

After validation of the new name, the procedure editor will be updated, displaying the name of the group *(see Figure 786, page 636)*.

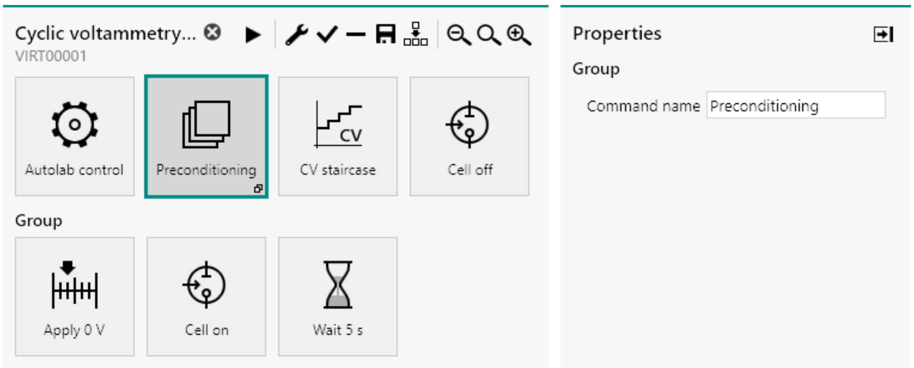


Figure 786 The group is renamed

10.8 Enabling and disabling commands

It is possible to enable or disable a commands or several commands at once in the procedure editor.

Disabled commands are shown in the procedure editor greyed out. These commands are still part of the procedure but are not executed during a measurement.

Disabled commands can be enabled again.

NOTE

It is not possible to enable or disable commands in a procedure while the procedure is running.

10.8.1 Disabling commands

To disable one or more commands in the procedure editor, select the command or commands and click the ✓ button located in the top-right corner of the procedure editor *(see Figure 787, page 637)*.

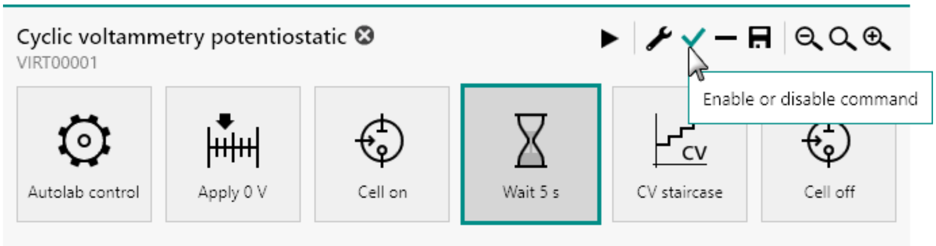


Figure 787 Commands in the procedure can be disabled

The disabled commands will be greyed out, indicating that they are disabled *(see Figure 788, page 637)*.

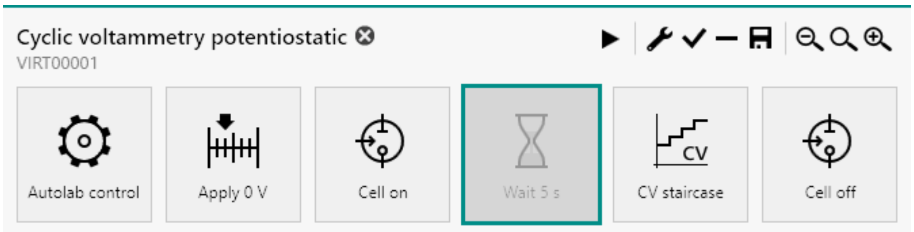


Figure 788 The Wait command is greyed out

Disabled commands are not executed during a measurement.

NOTE

It is not possible to change the status of a disabled command while the procedure is running.

If the disabled command is a **Group** command or if the disabled command has additional commands stacked below it, all the commands located in this group or stack are disabled *(see Figure 789, page 638)*.

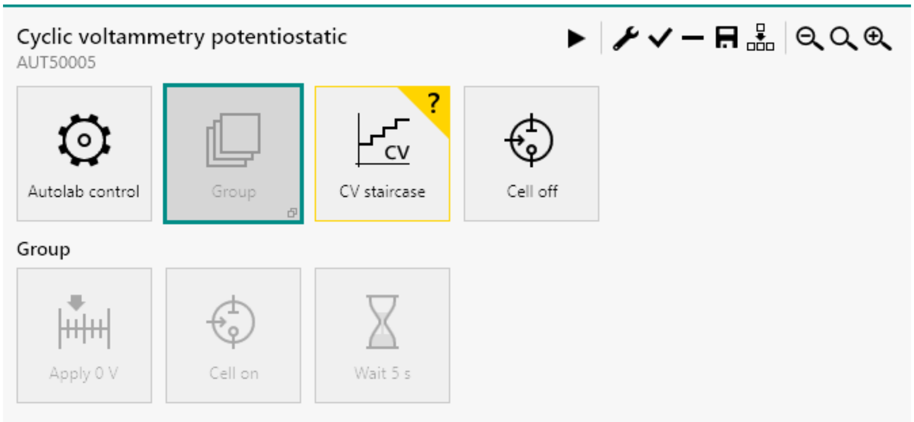


Figure 789 Disabling a Group command in the procedure editor

10.8.2 Enabling commands

Disabled commands can be enabled again in the procedure editor. Enabling a disabled command restores this command to its previous state in the procedure.

To enable one or more *disabled* commands in the procedure editor, select the command or commands and click the ✓ button located in the top-right corner of the procedure editor *(see Figure 790, page 638)*.

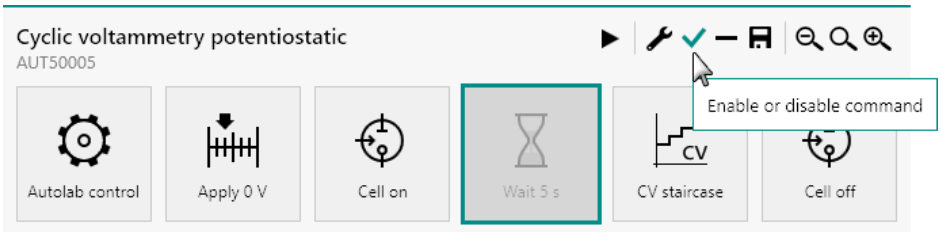


Figure 790 Enabling disabled commands

When a disabled **Group** command is enabled, all the commands contained in the group are enabled. This also applies to command located in a disabled stack.

10.9 Adding and removing commands

Commands available in the commands browser can be added to a NOVA procedure. This way, existing or new procedures can be completely customized to fit the experimental requirements. Commands located in any procedure can also be removed.

10.9.1 Adding commands

Two different methods can be used to add commands to a NOVA procedure:

- **Drag and Drop method:** by selecting the required command and dragging it in the procedure editor. This method can be used to add the required command anywhere in the procedure and to any available track in the procedure editor.
- **Double click method:** by double clicking the command to the add to the procedure. This method adds the selected command to the end position of the active procedure track in the procedure editor.

10.9.1.1 Adding commands using the drag and drop method

The drag and drop method can be used to add new commands to a procedure. Any command provided in the command browser, located on the left-hand side of the procedure editor frame can be added to the procedure.

NOTE

To drag and drop an item on screen, click the item and while holding the mouse button, move the item to a new location. Release the mouse button to validate the new position of the item.

NOTE

It is only possible to add one command at a time.

10.9.1.1.1 Using the drag and drop method to add commands to the main track

The following steps illustrate how to use the drag and drop method for adding commands to the main track of a procedure. This method is used to add a **Wait** command to the following procedure *(see Figure 791, page 640)*.

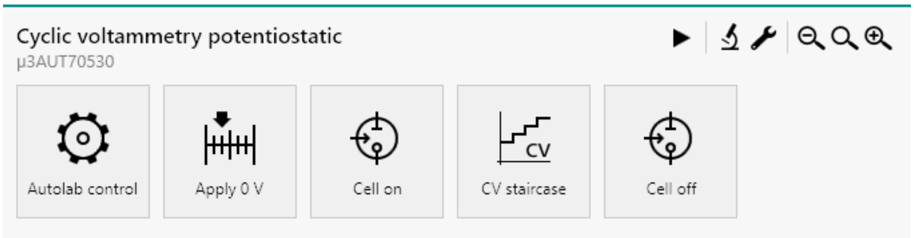
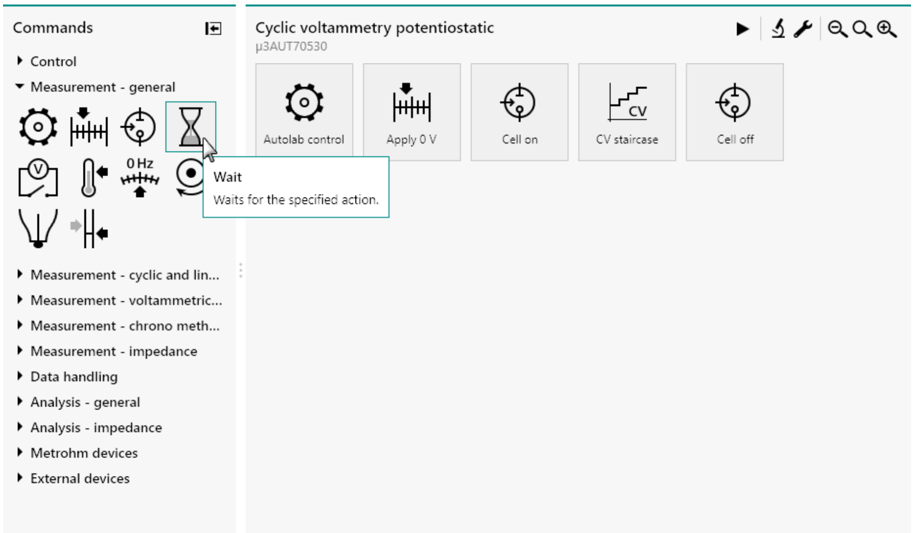


Figure 791 The original procedure

The **Wait** command will be added between the **Cell** command and the **CV staircase** command.

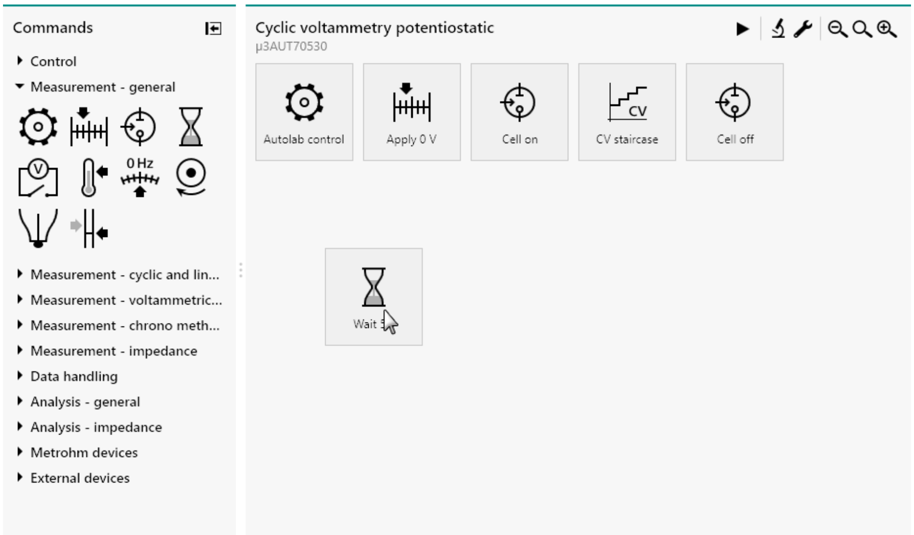
1 Select the command to add

Click the command to add to the procedure.



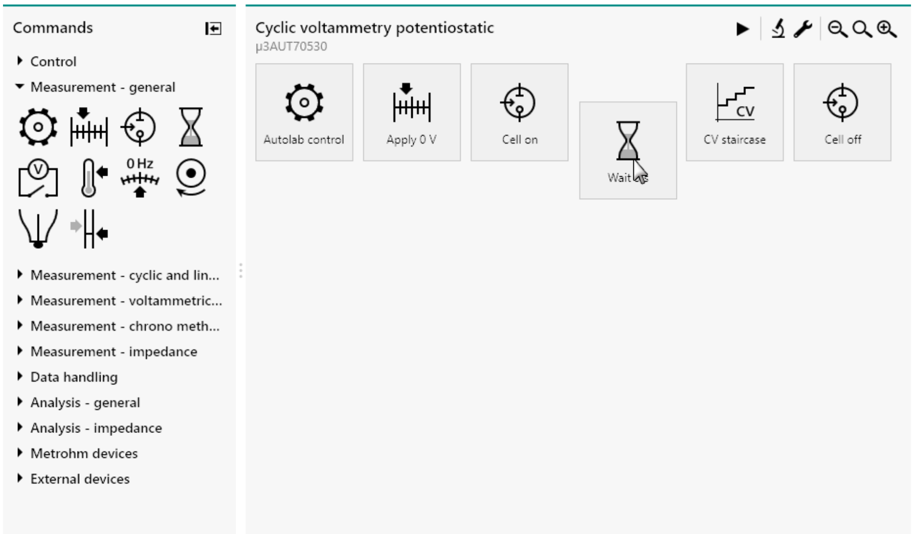
2 Drag the command in the procedure editor

While holding the mouse button, drag the command to the procedure editor frame.



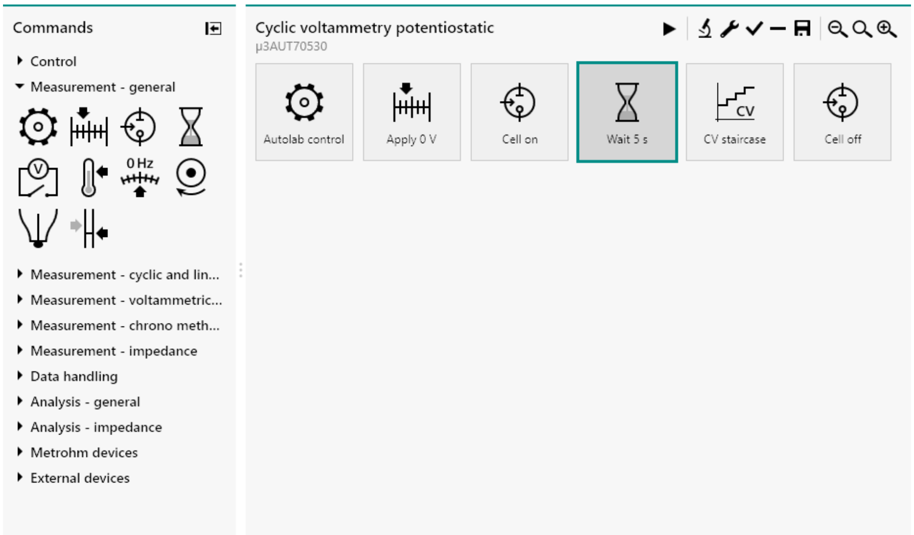
3 Place the command in the procedure

Place the new command in the procedure.



4 Finalize the insertion of the command

Release the mouse button to validate the position of the new command in the procedure.



10.9.1.1.2 Using the drag and drop method to add commands to a command group or a sub-track

The following steps illustrate how to use the drag and drop method for adding commands to a command group or to a sub-track . This method is used to add a **Message** command to the following procedure *(see Figure 792, page 642)*.

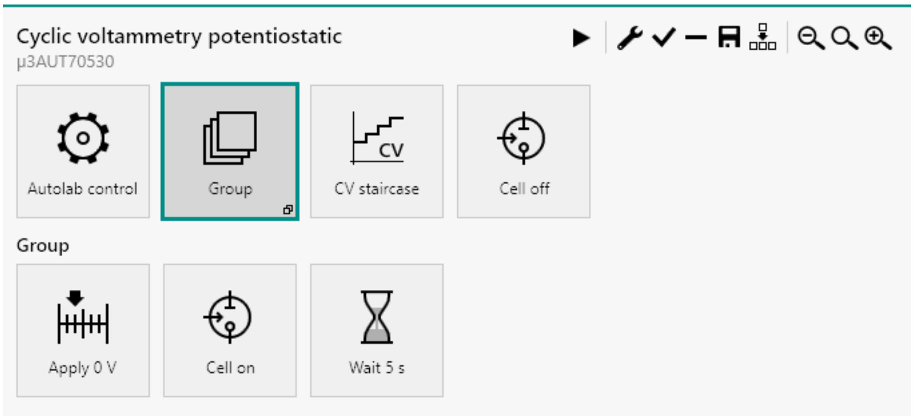
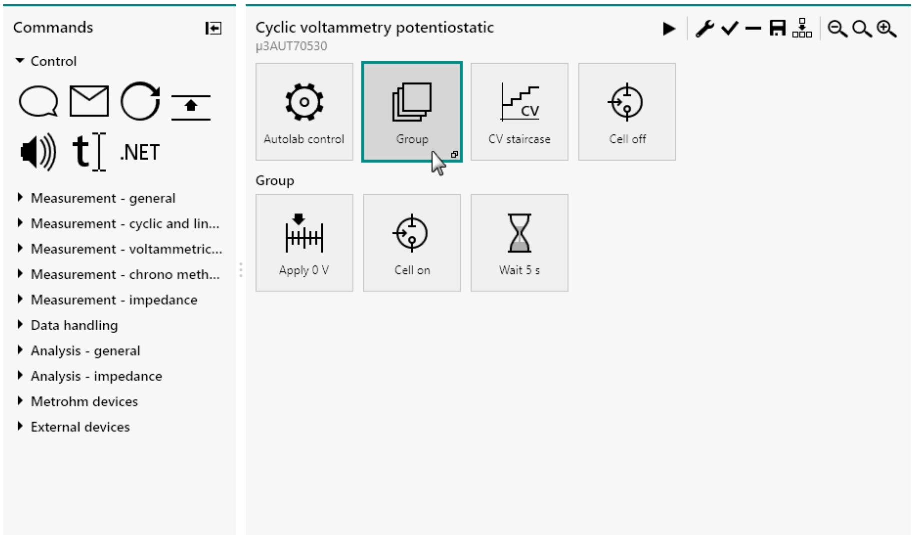


Figure 792 The original procedure

The **Message** command will be added at the beginning of the Group track, before the **Apply** command.

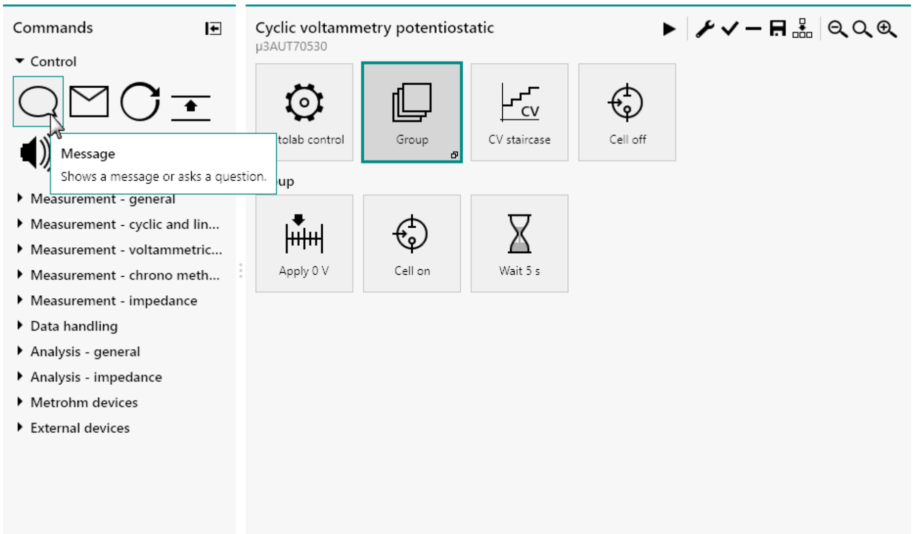
1 Select the Group command

Click the **Group** command in the procedure editor to display the commands located in the group below the main procedure track.



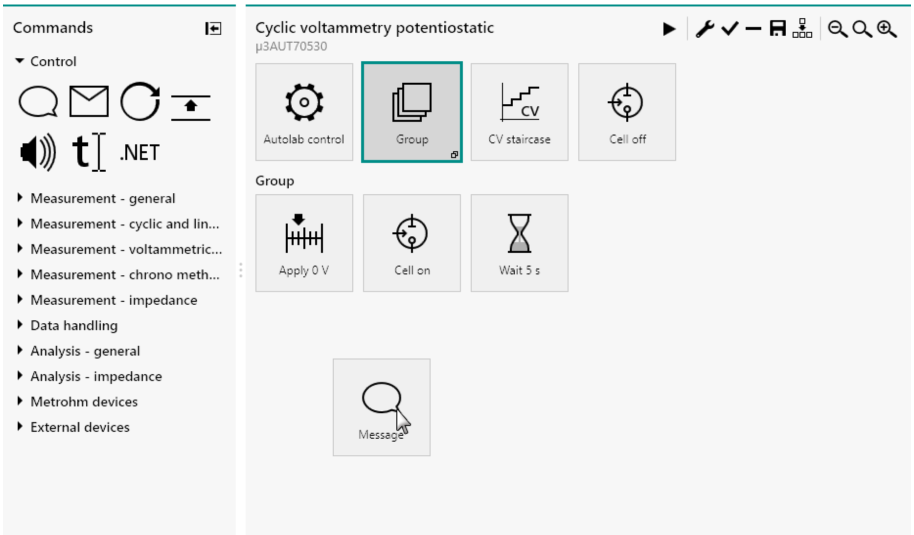
2 Select the command to add

Click the command to add to the procedure.



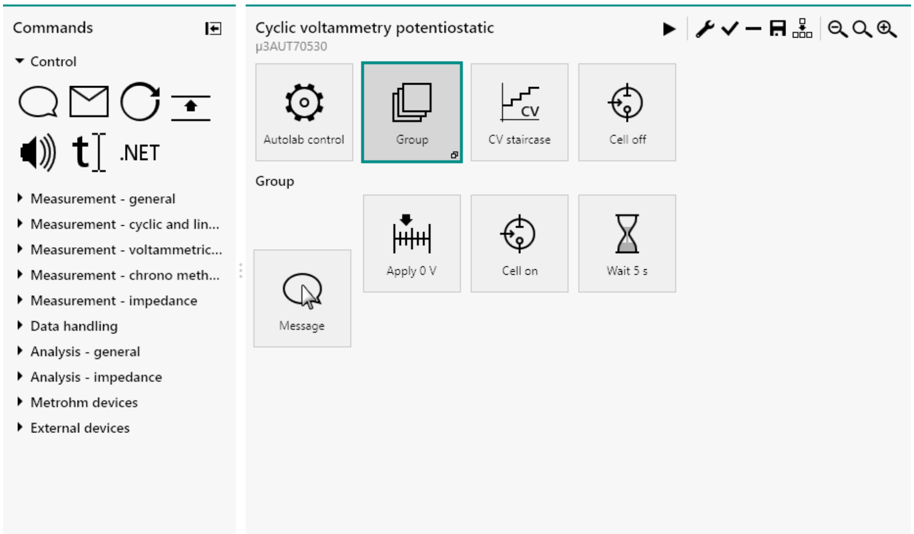
3 Drag the command in the procedure editor

While holding the mouse button, drag the command to the procedure editor frame.



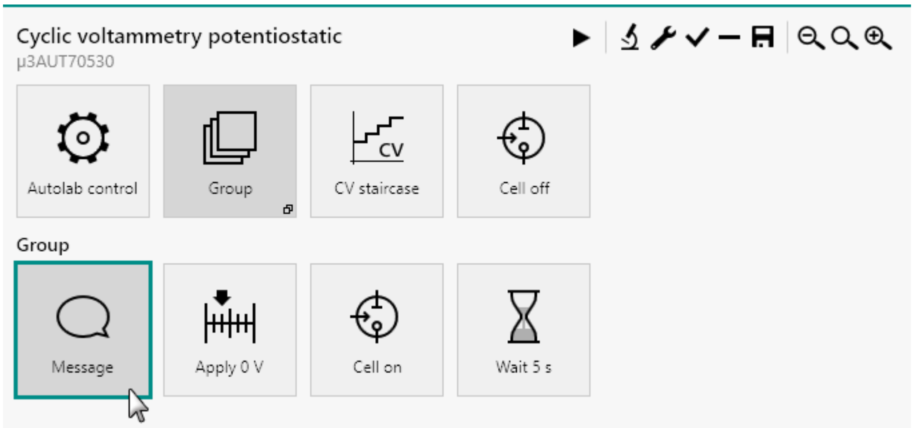
4 Place the command in the procedure

Place the new command in the procedure.



5 Finalize the insertion of the command

Release the mouse button to validate the position of the new command in the procedure.



10.9.1.2 Adding commands using the double click method

The *double click* method provides the means to quickly add commands to a procedure. Double clicking a command in the commands browser adds the command to the end of the active track in the procedure editor.

For example, double clicking the **Play sound** command adds this command after the **Cell** command in the *Cyclic voltammetry potentiostatic* procedure *(see Figure 793, page 645)*.

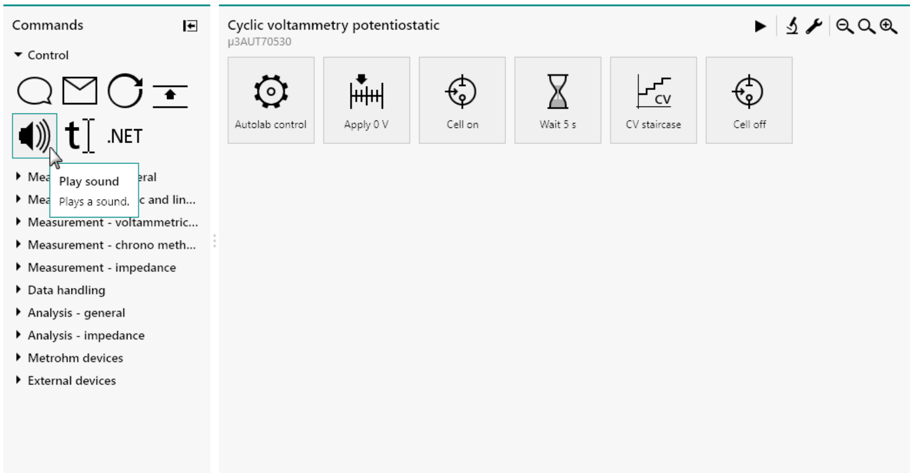


Figure 793 Adding a command to a procedure using the double click method

The command is added to the procedure editor *(see Figure 794, page 646)*.

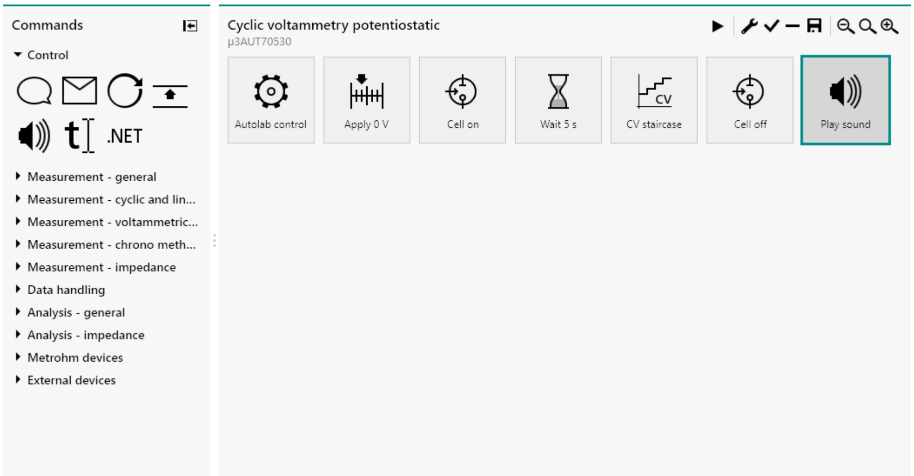


Figure 794 The command is added to the procedure

If the procedure contains more than one track, the double-clicked command is added at the very end of the active track. In the example shown below, when the contents of the **Group** are visible, double-clicking the **Play sound** command adds the command at the end of the sequence in the **Group** *(see Figure 795, page 646)*.

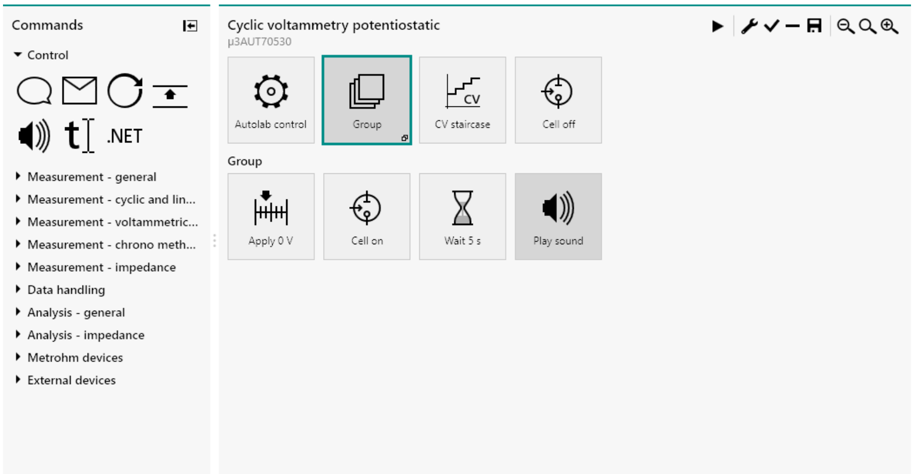


Figure 795 The command is added to the active track

If the contents of the **Group** command are not displayed in the procedure editor, double-clicking the **Play sound** command leads to the same result as in *Figure 794*.

10.9.2 Removing commands

It is possible to remove commands from a NOVA procedure, by selecting the command or commands and clicking the ━ button or pressing the **[Delete]** key *(see Figure 796, page 647)*.

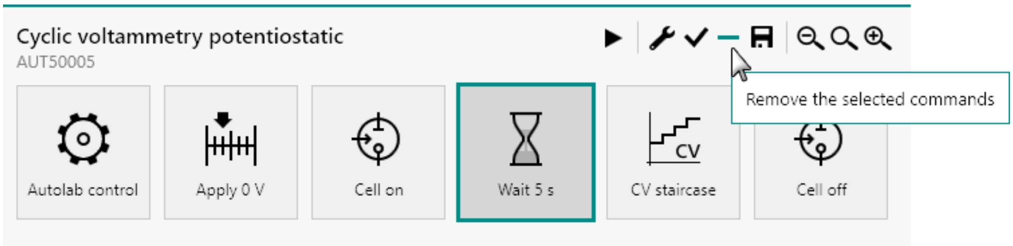


Figure 796 Select the command or commands to delete

It is also possible to use the *Delete* option available from the **Edit** menu.

Deleted commands are removed from the procedure and all commands located after the deleted commands are shifted leftwards to replace the deleted commands *(see Figure 797, page 647)*.

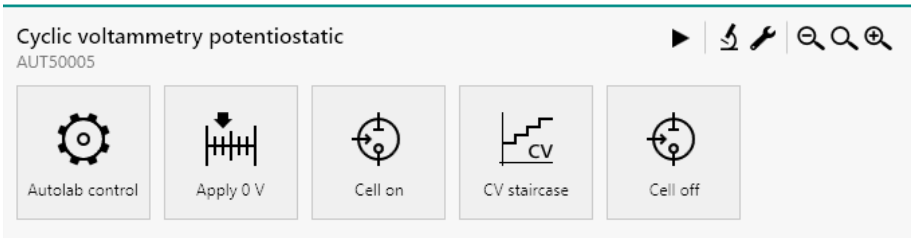


Figure 797 The commands located on the right of the deleted command are shifted leftwards

NOTE

Removing a **Group** command or a command stack removes all the commands in the group or in the stack from the procedure.

10.10 Moving commands

Commands located in a procedure can be moved and relocated anywhere in the procedure using the drag and drop method. Whenever a command is moved, the other commands located in the procedure are shifted leftwards or rightwards in order to create room for the moved command.

NOTE

It is only possible to move one command at a time.

10.10.1 Moving commands using the drag and drop method

The following steps illustrate how to use the drag and drop method for moving commands in the procedure editor. This method is used to move a **Message** command, located in a **Group** command, to the beginning of the following procedure *(see Figure 798, page 648)*.

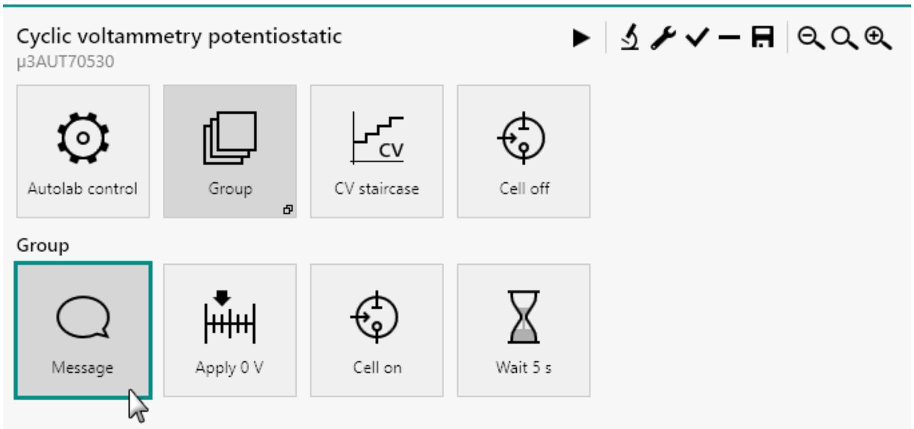
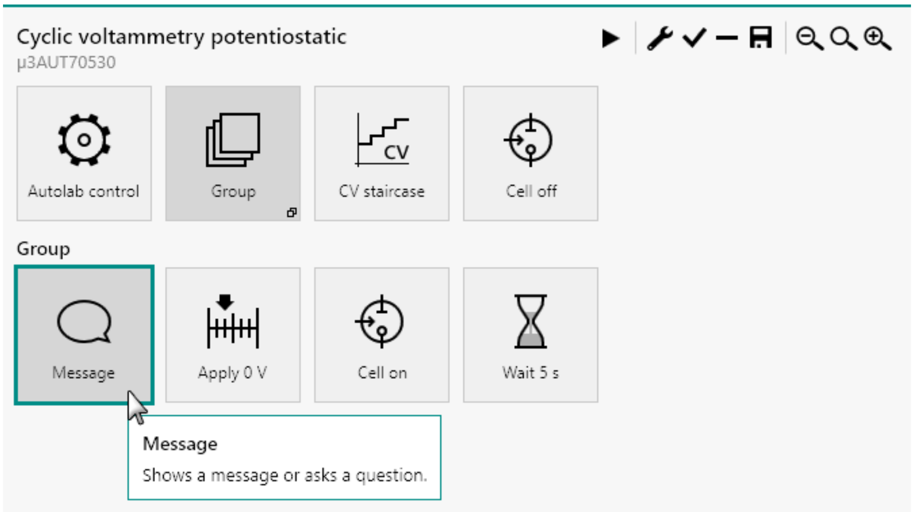


Figure 798 The original procedure

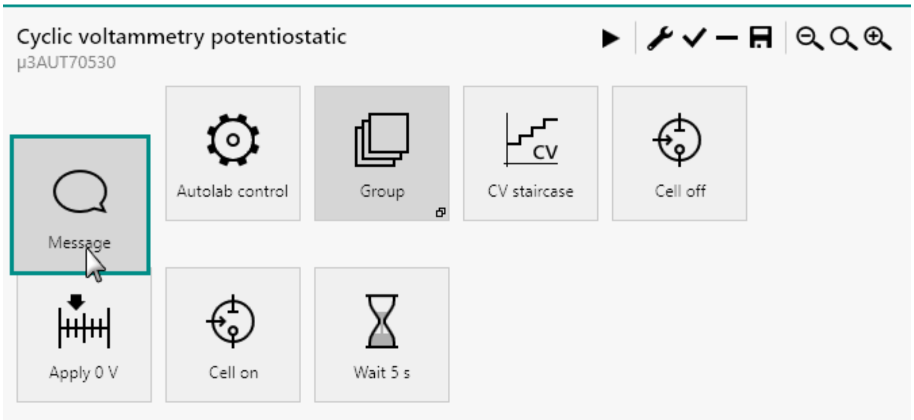
1 Select the command to move

Click the command to move in the procedure editor.



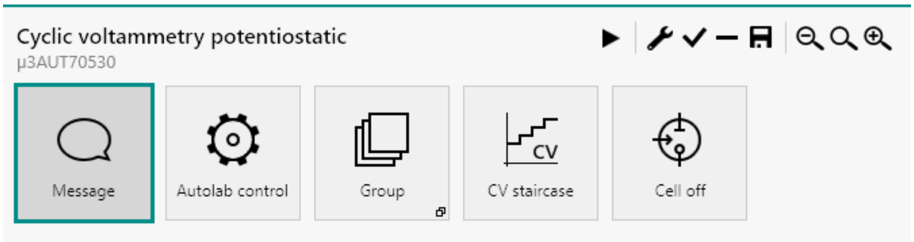
2 Drag the command to move

While holding the mouse button, move the selected command to a new location in the procedure editor. The other commands located in the procedure editor will be shifted in order to make room for the moved command.



3 **Release the mouse button**

Release the mouse button to validate the new position of the command in the procedure.



10.10.2 Using the drag and drop method to move commands to a command group or a sub-track

The following steps illustrate how to use the drag and drop method for moving commands to a command group or to a sub-track . This method is used to move a **Message** command, located at the beginning of the procedure, to the **Group** command *(see Figure 799, page 649)*.

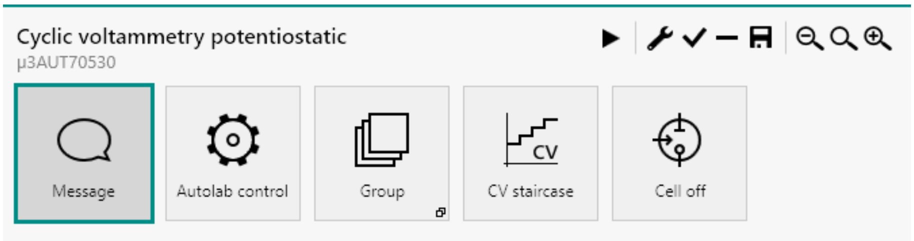
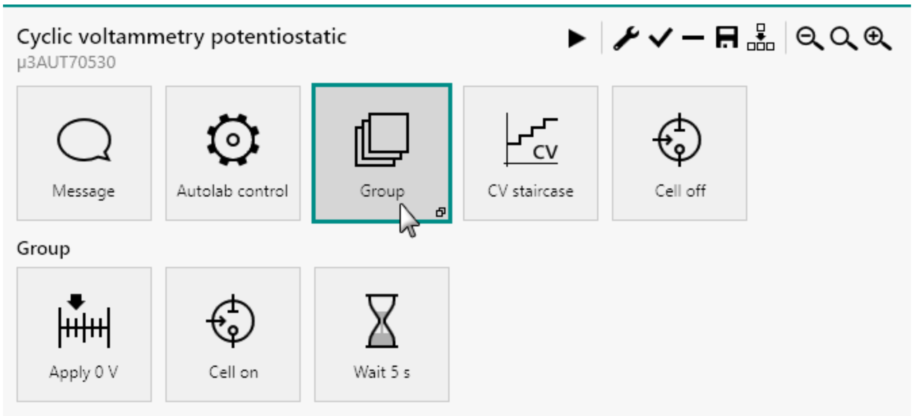


Figure 799 The original procedure

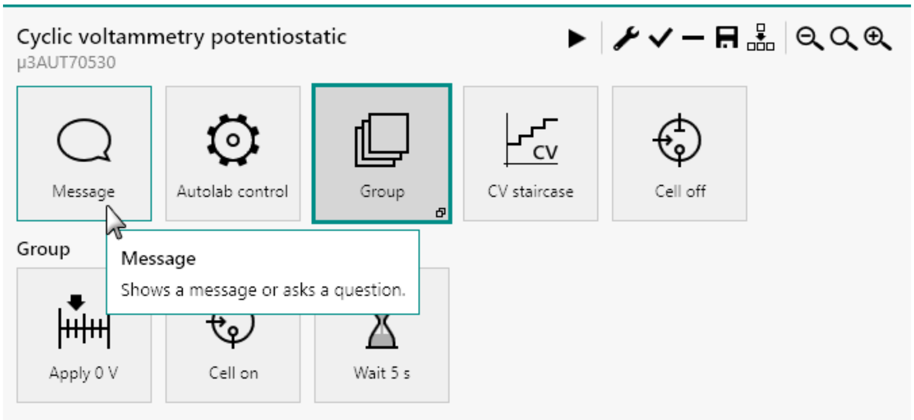
1 **Select the destination Group command**

Click the **Group** command in the procedure editor to display the commands located in the group below the main procedure track.



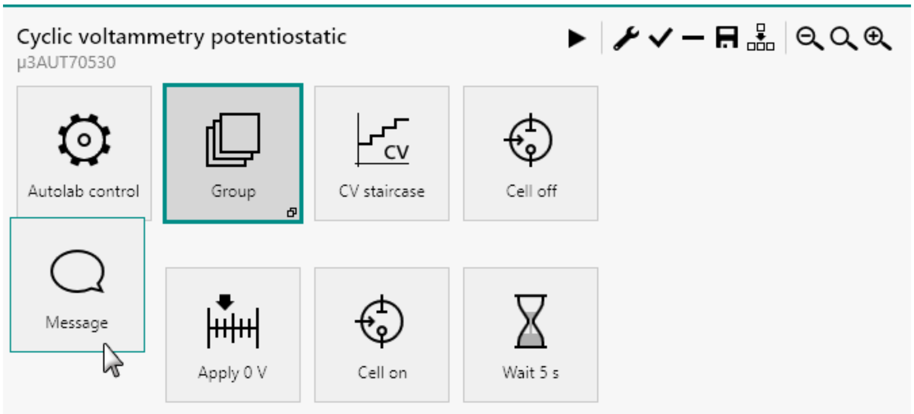
2 Select the command to move

Click the command to move in the procedure editor.



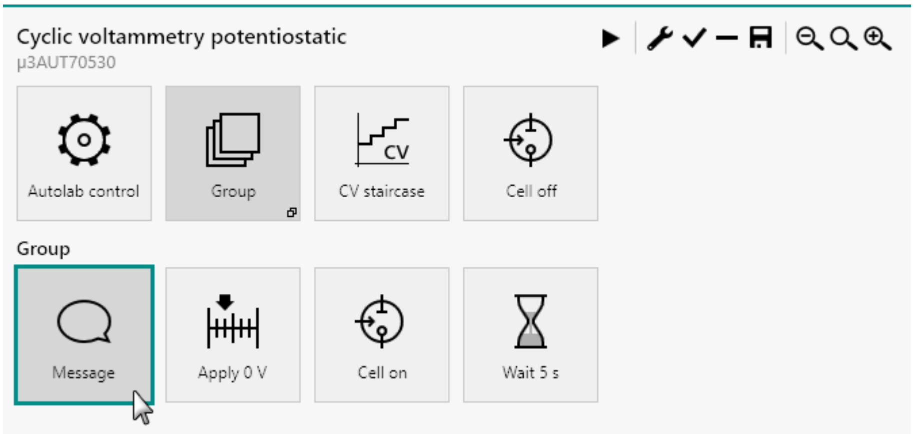
3 Drag the command to move

While holding the mouse button, move the selected command to a new location in the procedure editor. The other commands located in the procedure editor will be shifted in order to make room for the moved command.



4 **Release the mouse button**

Release the mouse button to validate the new position of the command in the procedure.



10.11 Moving multiple commands

It is possible to select multiple commands and move them in the procedure editor. All the selected commands will be moved using this method. To select multiple commands in the procedure, two methods can be used:

- **Selecting commands with the [CTRL] key:** holding the **[CTRL]** key, multiple command can be selected. This method allows the selection of non-adjacent commands.
- **Selecting commands with the [SHIFT] key:** holding the **[SHIFT]** key, multiple command can be selected. This selection method automatically selects all the commands located between the two outermost selected command. This method can only be used to select adjacent commands.

NOTE

The order in which the commands are selected is stored in the selection.

The selected command are highlighted *(see Figure 800, page 652)*.

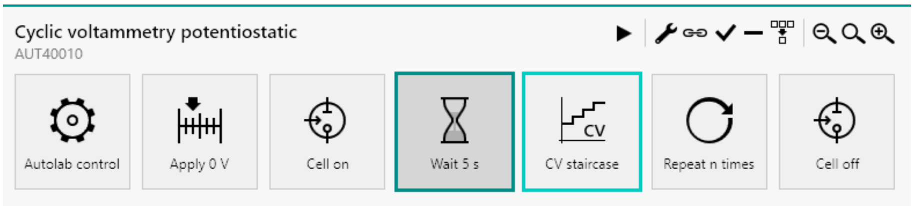


Figure 800 Selecting multiple commands in the procedure editor

Once the commands are selected, it is possible to use the editing tools like cut or copy and paste or drag and drop to edit the procedure. When the one of the commands in the selection is dragged through the procedure editor, an indicator will be shown in the top left corner of the command, indicating the number of additional commands moved at the same time *(see Figure 801, page 652)*.

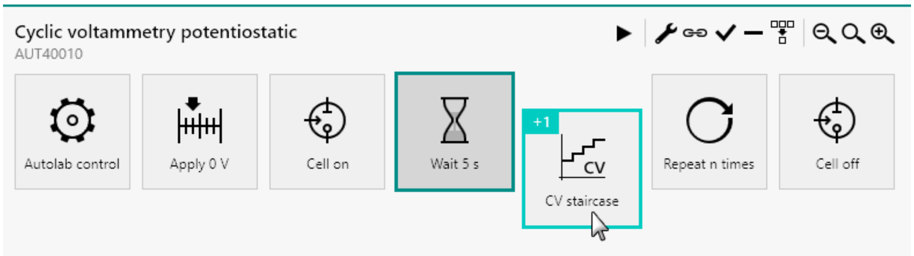


Figure 801 The number of commands dragged at the same time is indicated in the procedure editor

In the case of *Figure 801*, one additional command in included in the selection. A **+1** indicator is therefore shown in the top left corner of the **CV staircase** command.

The selected commands can be repositioned in the procedure editor using the grad and drop method *(see Figure 802, page 652)*.

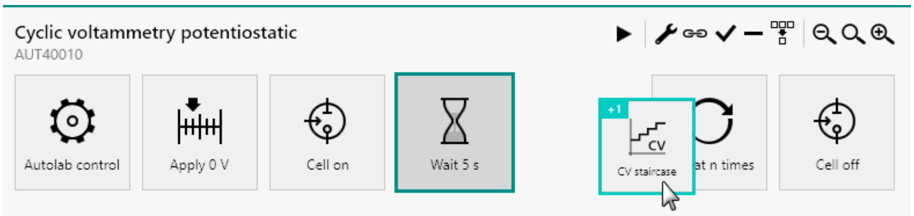


Figure 802 Relocating commands using the drag and drop method

The releasing the mouse confirms the new position of the commands in the procedure editor *(see Figure 803, page 653)*.

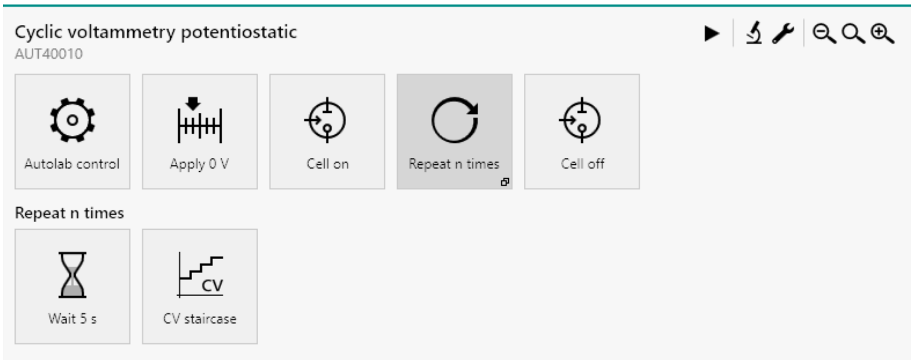


Figure 803 The commands are repositioned in the procedure editor

NOTE

The commands are repositioned in the order in which they are selected!

10.12 Stacking commands

The procedure editor of NOVA can be used to *stack* command onto one another. Stacking works in the similar way in grouping commands *(see Chapter 10.7, page 634)*. The difference with a command group is that in a command stack, one command is used as a *parent* command and the other commands, stacked below the parent command, are acting as *child* commands.

The most commonly used command stack used in NOVA consists of a **Repeat** command and the command added to the **Repeat** command *(see Figure 804, page 653)*.

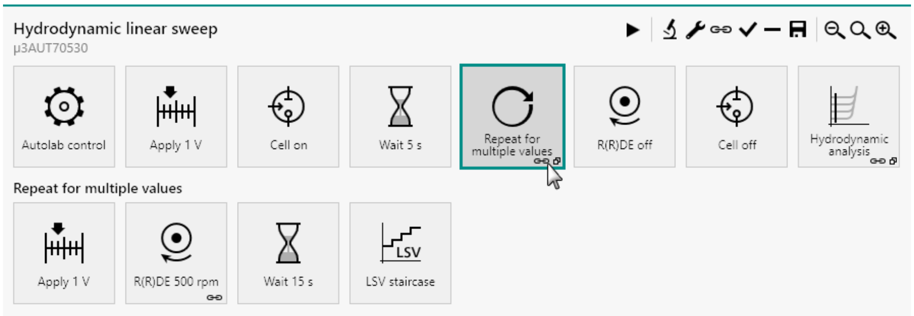


Figure 804 Example of stacked commands

10.12.1 Creating command stacks

The following steps illustrate how to create a command stack using the drag and drop method. This method is used to add a **Calculate signal** command to the following procedure *(see Figure 805, page 654)*.

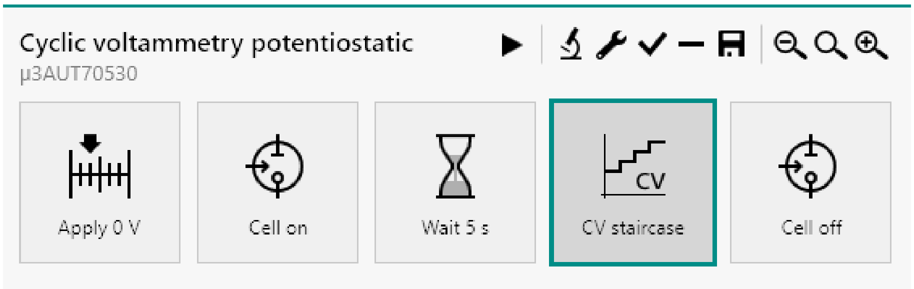
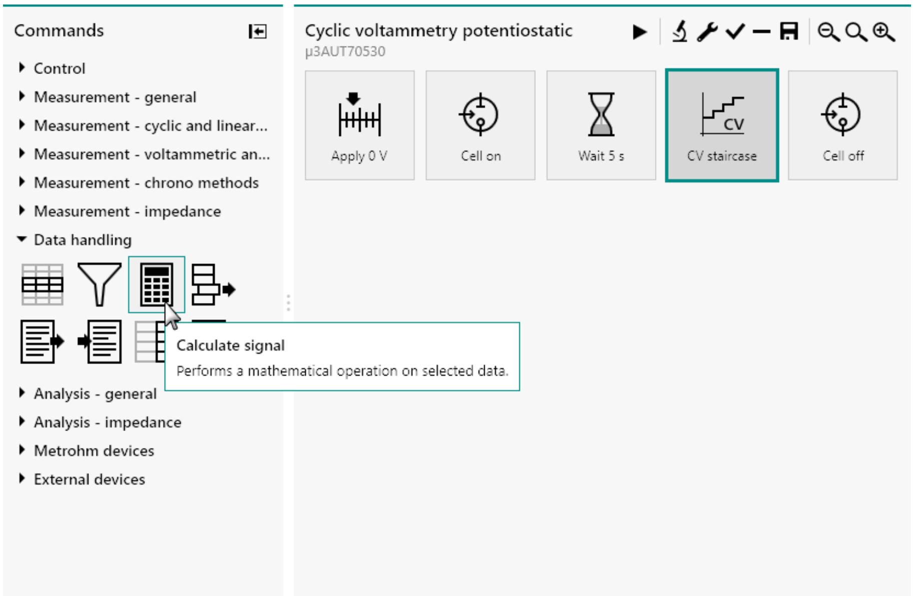


Figure 805 The original procedure

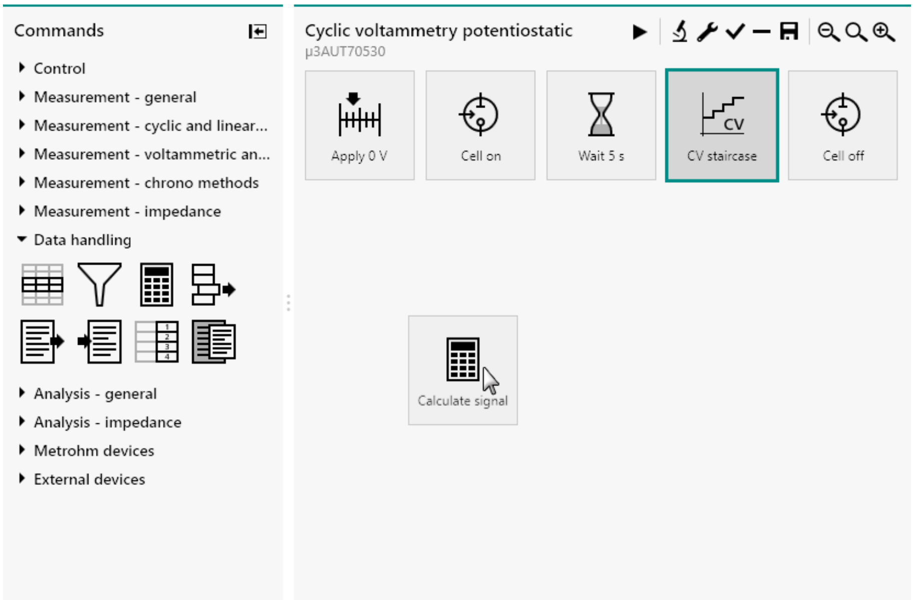
1 Select the command to add

Click the command to add to the procedure.



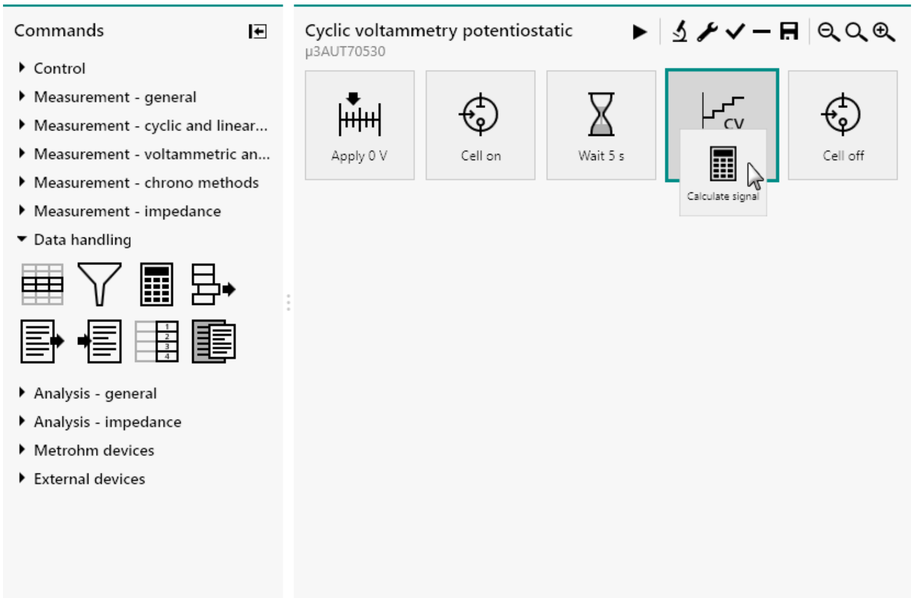
2 Drag the command in the procedure editor

While holding the mouse button, drag the command to the procedure editor frame.



3 Drag the command onto the parent command

Drag the command onto the parent command.

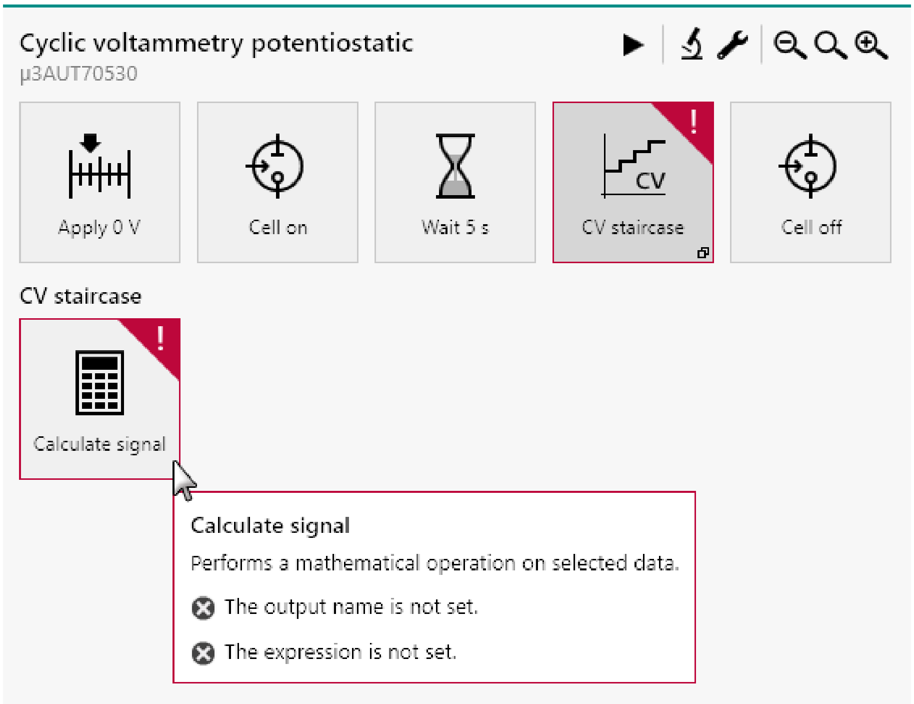


NOTE

The command shrinks when located onto the parent command.

4 Finalize the command stack

Release the mouse button, validating the location of the command and the command stack.



10.12.2 Remove commands from stacks

To remove a command from a stack, simply select the command in the stack and click the ━ button or press the **[Delete]** key. The command will be removed from the stack *(see Figure 806, page 656)*.

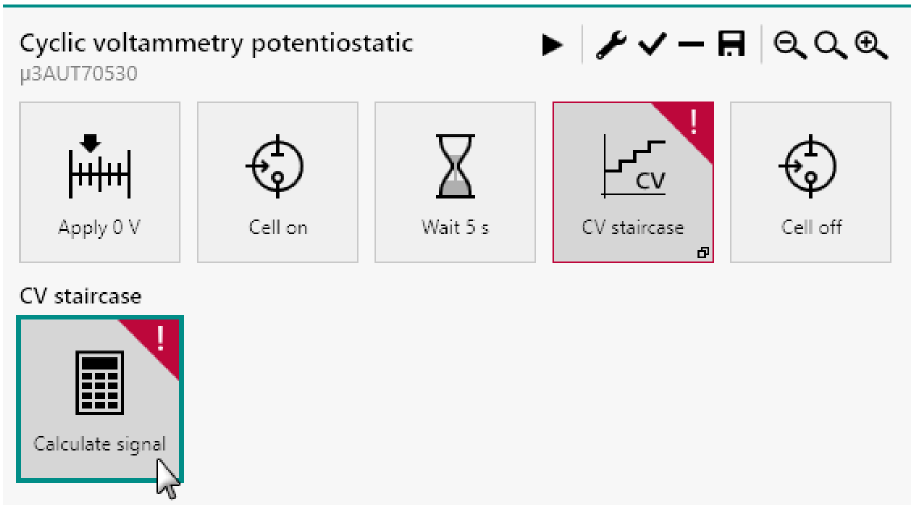


Figure 806 Select the command to remove from the stack

The command will be removed. It there are no more commands in the stack, the stack is removed from the procedure *(see Figure 807, page 657)*.

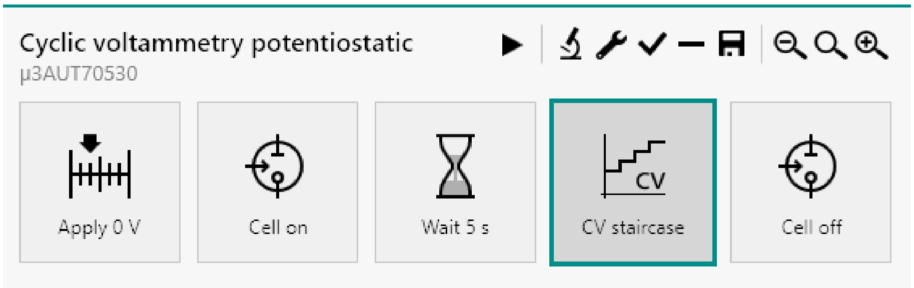


Figure 807 The command is removed from the stack

10.13 Links

Links are essential programming tools provided by NOVA. A link creates a relationship between two or more properties in a procedure. Using links, it is possible to create procedures in which command properties are adjusted in function of command properties these are linked to.

Some of the default procedures, supplied with NOVA and available in the **Default procedures** group in the **Library** contain pre-configured links. This is the case for the *Hydrodynamic linear sweep* procedure *(see Figure 808, page 657)*.

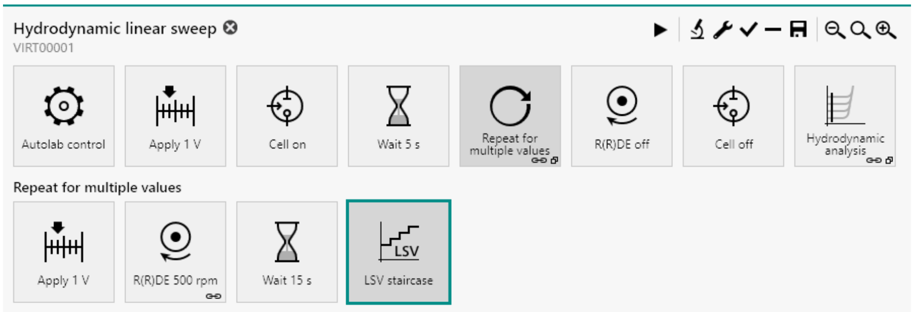


Figure 808 The default Hydrodynamic linear sweep procedure

This procedure contains a **Repeat** command, configured in the *Repeat for multiple values* mode. The rotation rates required for this measurement are pre-defined in the **Repeat** command *(see Figure 808, page 657)*.

The small chain drawing symbol in the bottom right corner of the **Repeat** command indicates that this command is linked to another command. This means that the properties of the **Repeat** command will be used during the measurement to adjust the properties of another command in the procedure.

The **Repeat** command has a list of predefined rotation rates *(see Figure 809, page 658)*.

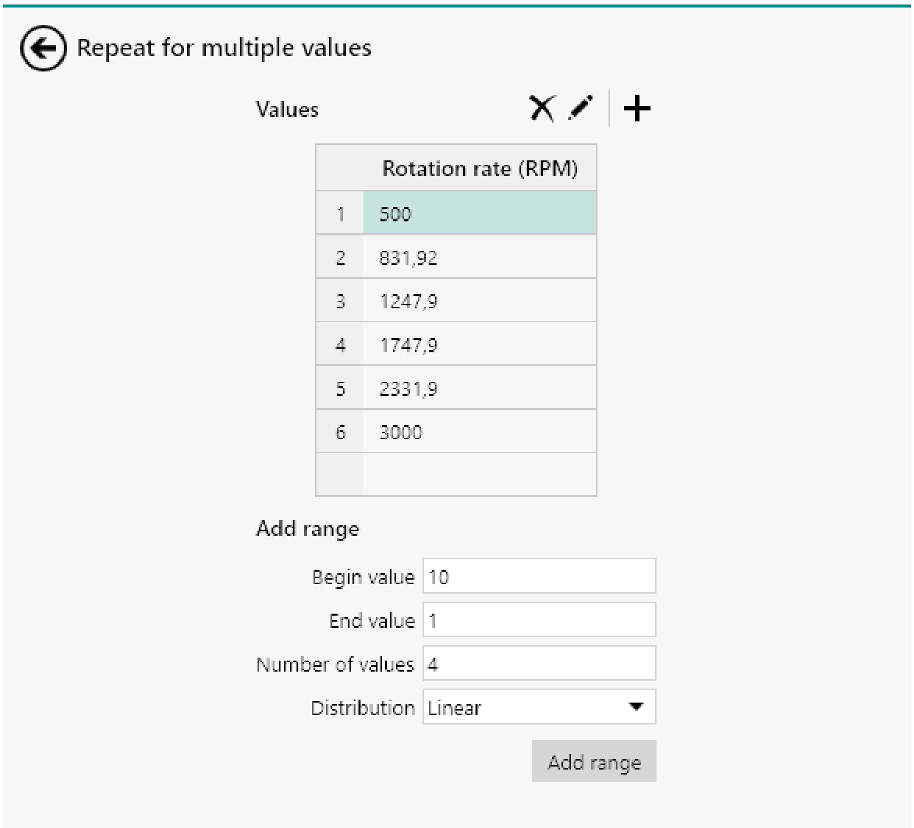


Figure 809 The Repeat command has a list of predefined rotation rates

The rotation rate of a rotating disk electrode (RDE) or rotating ring-disk electrode (RRDE) can therefore be automatically adjusted during a measurement. In this procedure, the **Repeat** command is linked to a R(R)DE command in the repeat loop.

This chapter explains how the linking mechanism works in NOVA and how to perform the following link-related actions:

- View links
- Create links
- Edit links

10.13.1 Viewing links

It is possible to show the links of any linked command in NOVA by selecting such a command anywhere in the procedure and clicking the button located in the top right corner of the procedure editor *(see Figure 810, page 659)* or by using the keyboard shortcut **[CTRL]** + **[L]**.

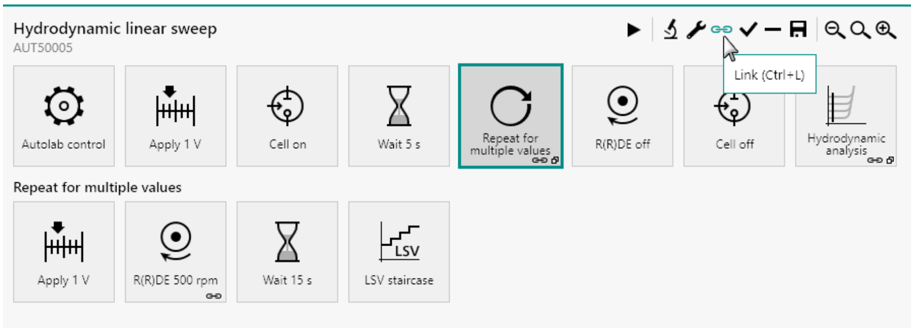


Figure 810 Viewing procedure links

The links are displayed in the dedicated **Edit link** screen. All the links involving the properties of the selected command are shown *(see Figure 811, page 659)*.

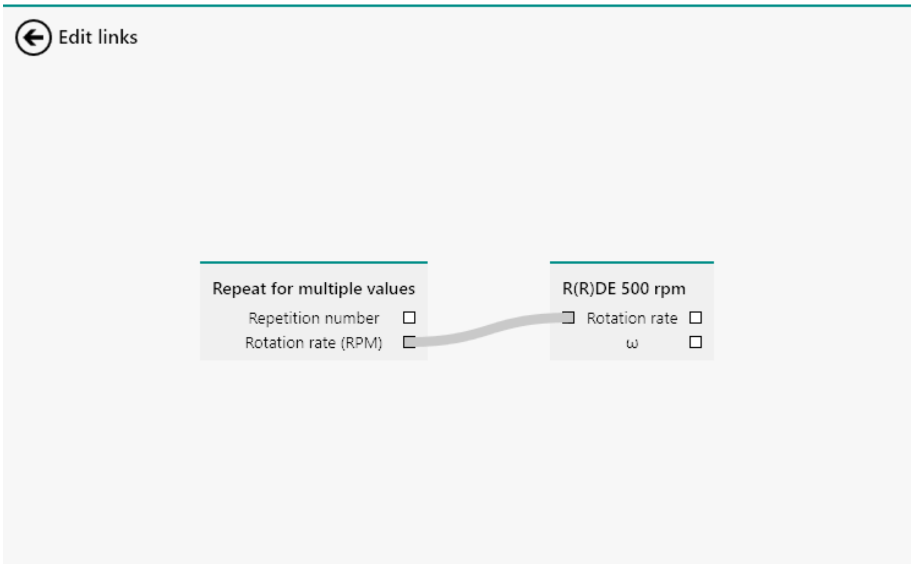


Figure 811 The Edit links screen

The following information is represented in the **Edit links** screen:

- **Commands:** all commands linked to the selected command are represented in the **Edit links** screen. Commands located before the selected command are represented on the left of the screen and command located after the selected command are represented on the right of the screen.
- **Linkable properties:** all linkable properties of the commands represented in the **Edit links** screen are represented. These properties are identified with a name and one or more anchoring points.
- **Anchoring points:** one or more anchoring points, identified by a ☐ symbol, are represented for each linkable property. Anchoring points locating on the left of a linkable property are *output* points. Anchoring points located on the left of a linkable property are *input* points.
- **Link line:** one or more grey lines connecting two or more anchoring points.

The grey link line connects the anchoring points located on the right of the *Rotation rate (RPM)* property of the **Repeat** command with the anchoring point located on the left of to the *Rotation rate* property of the **R(R)DE** command.

This link indicates that all the values defined in the **Repeat** command will be used to change the *Rotation rate* property of the **R(R)DE** command during the measurement. The *Rotation rate (RPM)* property of the **Repeat** command is used as an output and the *Rotation rate* property of the **R(R)DE** command is used as an input, indicating the direction of the link.

10.13.2 Creating links

It is possible to create links in any NOVA procedure containing linkable commands. To illustrate this option, the following procedure template will be used *(see Figure 812, page 660)*.

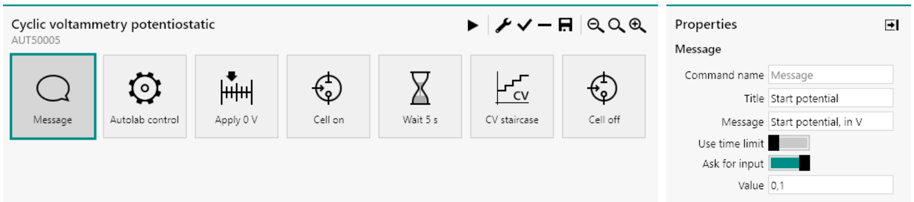


Figure 812 The procedure used to illustrate the creation of links in NOVA

NOTE

The procedure used in this example is created by adding a **Message** command, used as in Input, at the beginning of the default Cyclic voltammetry potentiostatic procedure.

Links can be created between two or more commands in the **Edit links** screen. To create links between two or more command, is it necessary to select these command in the procedure editor and click the button, located in the top right corner for the procedure editor or use the **[CTRL]** + **[L]** keyboard shortcut.

NOTE

The button is only visible when two or more commands are selected in the procedure editor.

Two procedures are available for creating links:

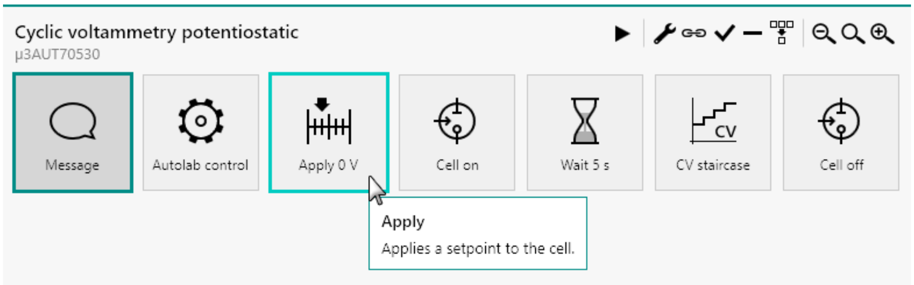
1. Creating links between two commands
2. Creating links between more than two commands

10.13.2.1 Creating a link between two commands

NOTE

The steps detailed in this section apply to an example procedure. These steps can be repeated for any procedure containing two or more linkable commands.

Select the **Message** command and the **Apply** command in the procedure editor.

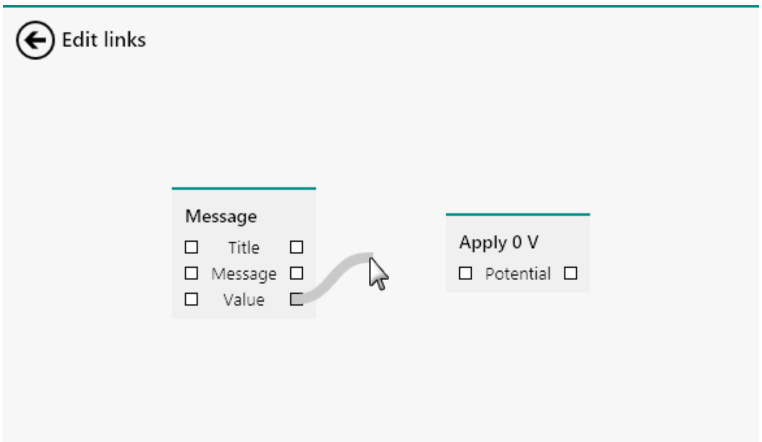


1 Open the Edit links screen

Click the button or use the **[CTRL]** + **[L]** keyboard shortcut to open the **Edit link** screen.

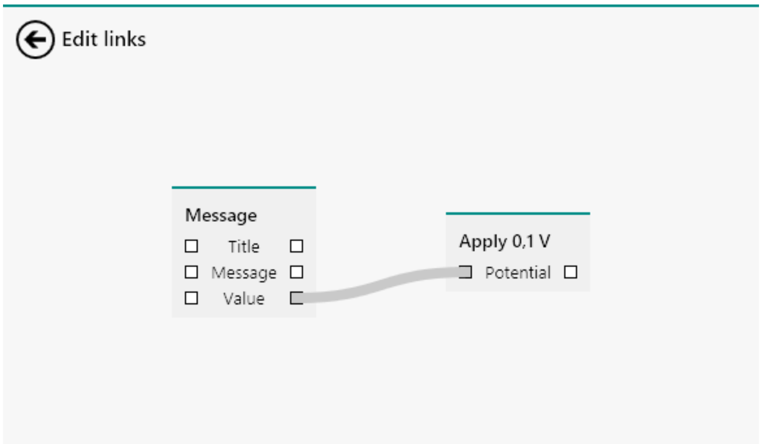
2 Set the output anchoring point

Click the output anchoring point of the *Value* property of the **Message** command and, while holding the mouse button, drag a line towards the input anchoring point of the *Potential* property of the **Apply** command.



3 **Set the input anchoring point**

While still holding the mouse button, move the line on top of the *Potential* input anchoring point of the **Apply** command and release the mouse button. The link line will be drawn between the two properties.



The link is now created.

4 **Close the Edit link screen**

Click the ⊕ button located in the top left corner to close the **Edit links** screen.

The properties are now linked. The created link will force both properties to be the same at any point during the measurement or whenever either one is modified by the user.

NOTE

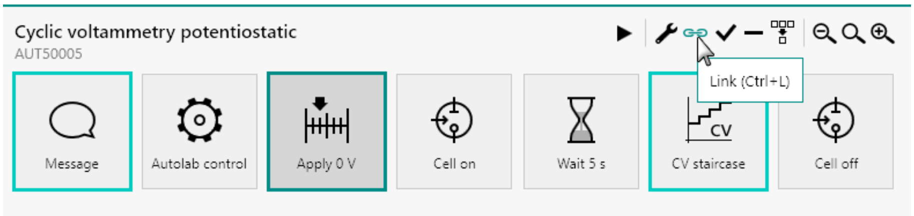
The **Apply 0 V** command is dynamically changed to **Apply 0,1 V** after the link is created.

10.13.2.2 Creating a link between more than two commands

NOTE

The steps detailed in this section apply to an example procedure. These steps can be repeated for any procedure containing two or more linkable commands.

Select the **Apply** command first, then the **Message** command and the **CV staircase** command in the procedure editor.

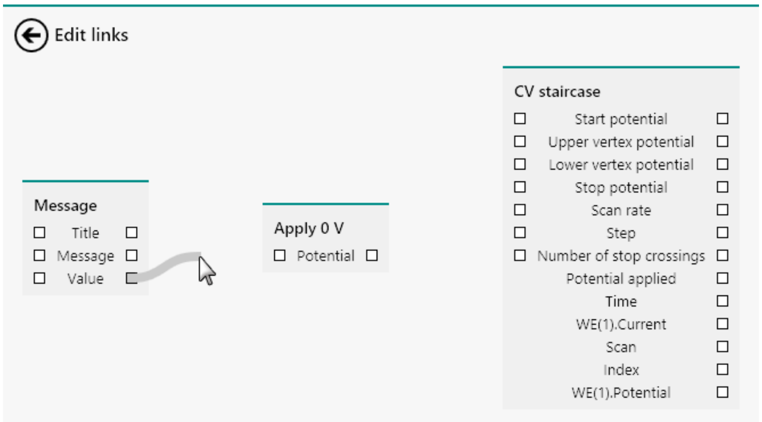


1 Open the Edit links screen

Click the button or use the **[CTRL]** + **[L]** keyboard shortcut to open the **Edit link** screen.

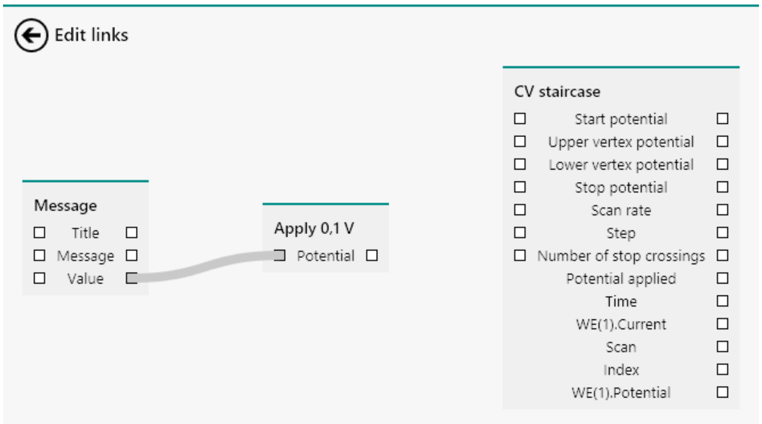
2 Set the first output anchoring point

Click the output anchoring point of the *Value* property of the **Message** command and, while holding the mouse button, drag a line towards the input anchoring point of the *Potential* property of the **Apply** command.



3 Set the first input anchoring point

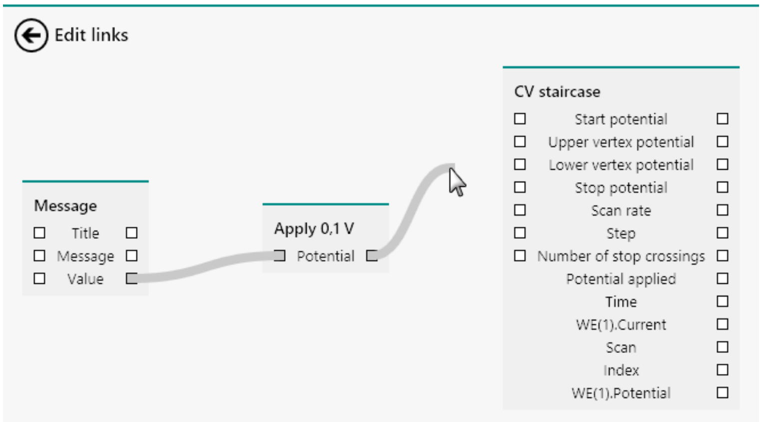
While still holding the mouse button, move the line on top of the *Potential* input anchoring point of the **Apply** command and release the mouse button. The link line will be drawn between the two properties.



The first link is now created.

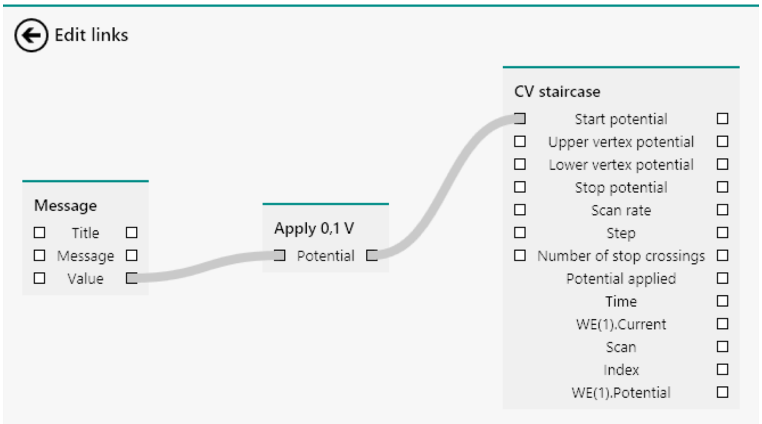
4 **Set the second output anchoring point**

Click the output anchoring point of the *Potential* property of the **Apply** command and, while holding the mouse button, drag a line towards the input anchoring point of the *Start potential* property of the **CV staircase** command.



5 **Set the second input anchoring point**

While still holding the mouse button, move the line on top of the *Start potential* input anchoring point of the **CV staircase** command and release the mouse button. The link line will be drawn between the two properties.



The second link is now created.

6 Close the Edit link screen

Click the ⊕ button located in the top left corner to close the **Edit links** screen.

The properties are now linked. The created links will force all properties to be the same at any point during the measurement or whenever either one is modified by the user.

NOTE

The **Apply 0 V** command is dynamically changed to **Apply 0,1 V** after the link is created.

10.13.2.3 Linking order

The procedure detailed in *Chapter 10.13.2.2* shows how to create links between more than two commands. Depending on the order in which the linked commands are selected in the procedure editor, the **Edit links** screen may be show in a different way.

The command selected as first item in the selection is always the main focus in the **Edit links** screen. The other commands are represented on the left and on the right, depending on their respective location in the procedure. In *Chapter 10.13.2.2*, the **Apply** command is selected first, which is why this command is located in the middle of the **Edit links** screen.

If the **Message** command is selected first, the **Apply** and **CV staircase** commands will be displayed in a different way in the **Edit links** screen *(see Figure 813, page 666)*.

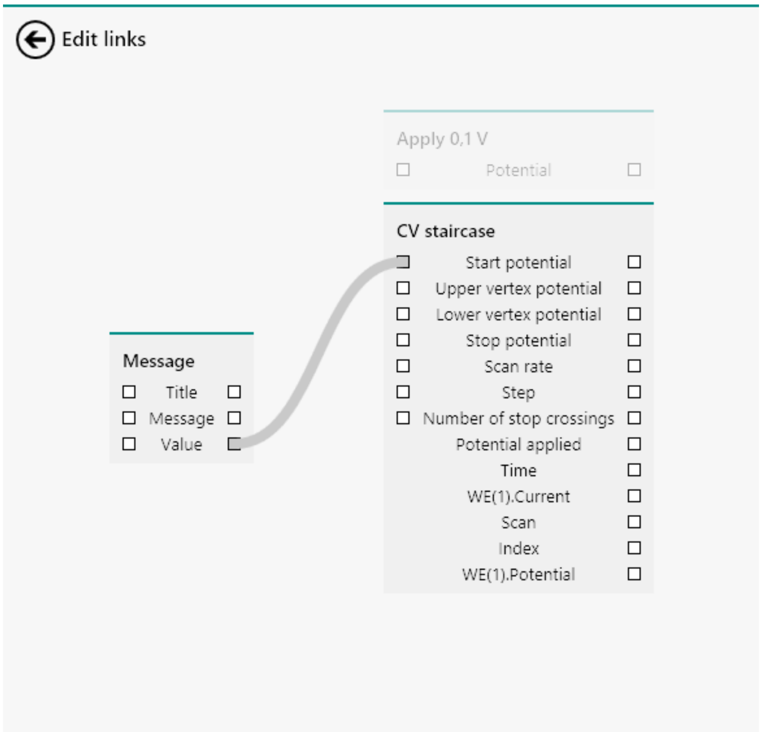


Figure 813 Selecting the Message command first

Both the **Apply** and **CV staircase** commands are shown at the right-hand side of the **Message** command because both commands are located after the **Message** command in the procedure. The **Apply** and **CV staircase** commands are shown above one another, with only one command in focus and the other out of focus (greyed out). The links can be edited between the **Message** command and the command in focus (**CV staircase** in *Figure 813*).

If the **CV staircase** command is selected first, the **Message** and **Apply** commands will be displayed in a different way in the **Edit links** screen *(see Figure 814, page 667)*.

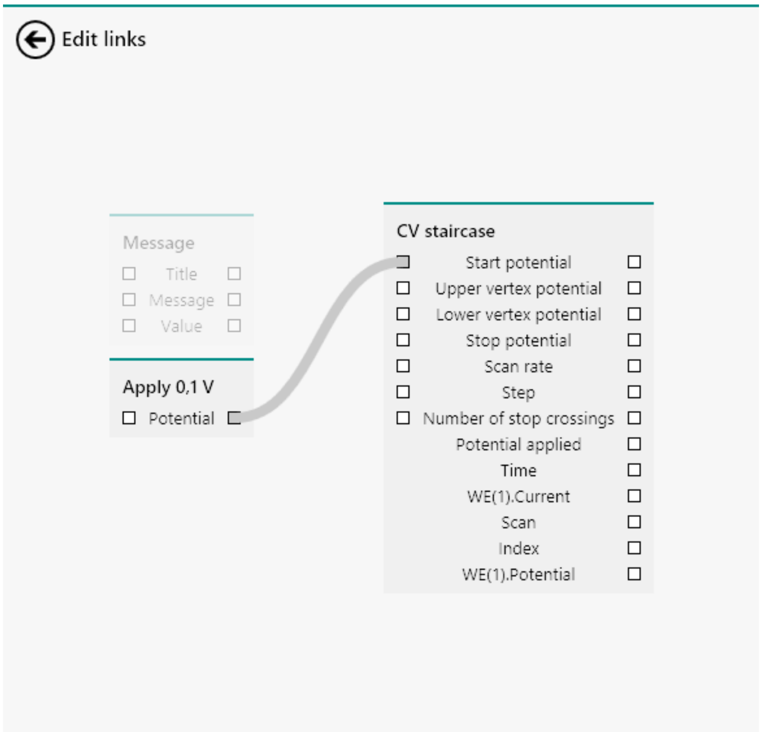


Figure 814 Selecting the CV staircase command first

Both the **Message** and **Apply** commands are shown at the left-hand side of the **CV staircase** command because both commands are located before the **CV staircase** command in the procedure. The **Message** and **Apply** commands are shown above one another, with only one command in focus and the other out of focus (greyed out). The links can be edited between the **CV staircase** command and the command in focus (**Apply** in *Figure 813*).

In both cases, it is possible to click the greyed out command to switch the focus in the **Edit links** screen and view or edit the links of the other command *(see Figure 815, page 668)*.

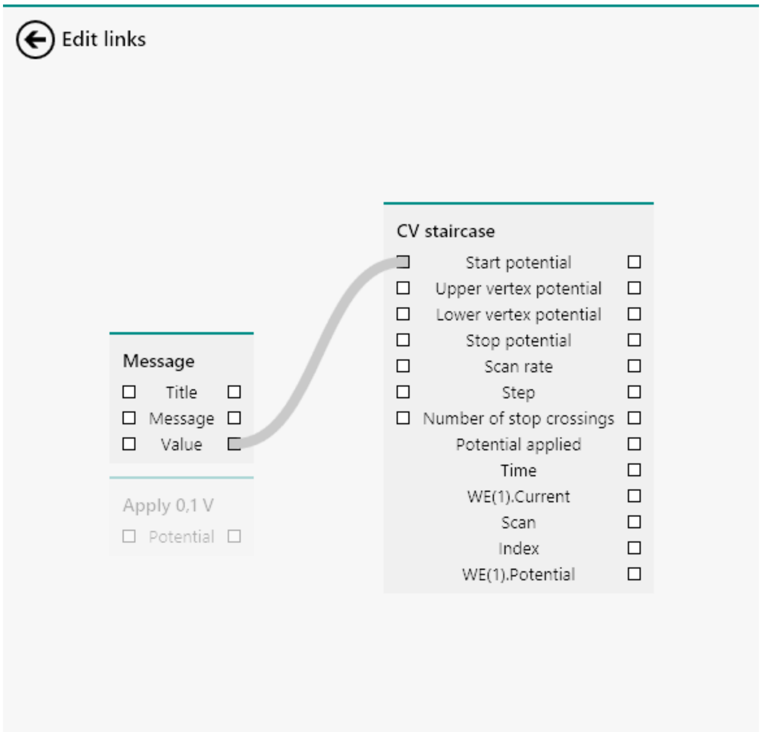


Figure 815 Switching the command focus in the Edit links screen

> **NOTE**
>
> It is also possible to use the mouse wheel to quickly scroll through the out of focus commands in the **Edit links** screen.

10.13.3 Editing links

It is possible to edit or remove links in a procedure at any time. To edit or remove links, it is necessary to open the Edit links screen by selecting one or more linked command and clicking the button or using the **[CTRL]** + **[L]** keyboard shortcut *(see Figure 816, page 668)*.

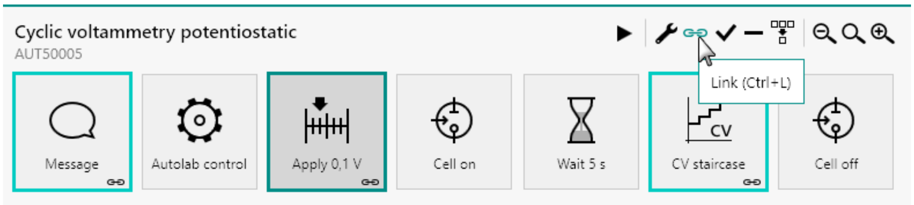


Figure 816 Opening the Edit links screen

This will open the Edit links screen *(see Figure 817, page 669)*.

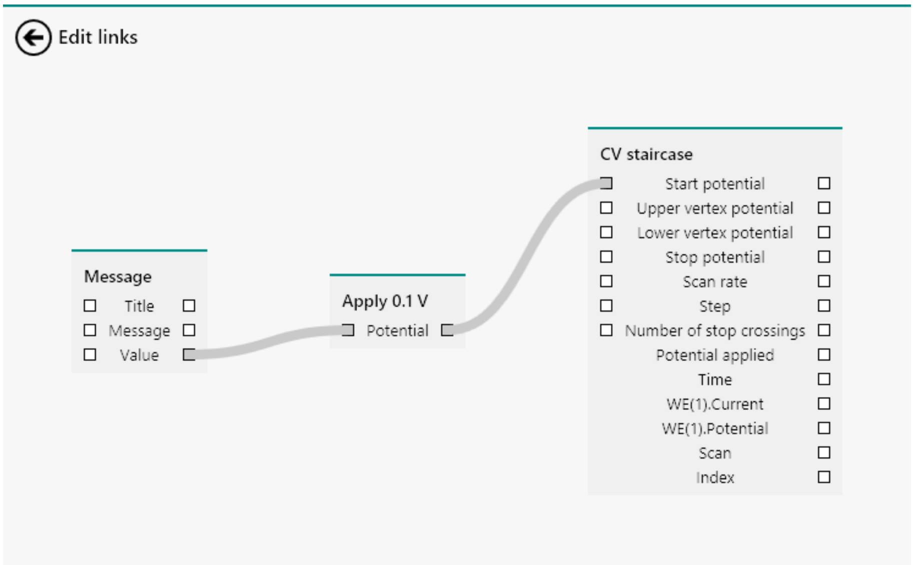


Figure 817 The Edit links screen

In the Edit links screen, it is possible to reroute an existing link by clicking one of the ends of the link and moving this to another anchoring point. In *Figure 818* the link is moved from the Start potential anchoring point to the Stop potential anchoring point.

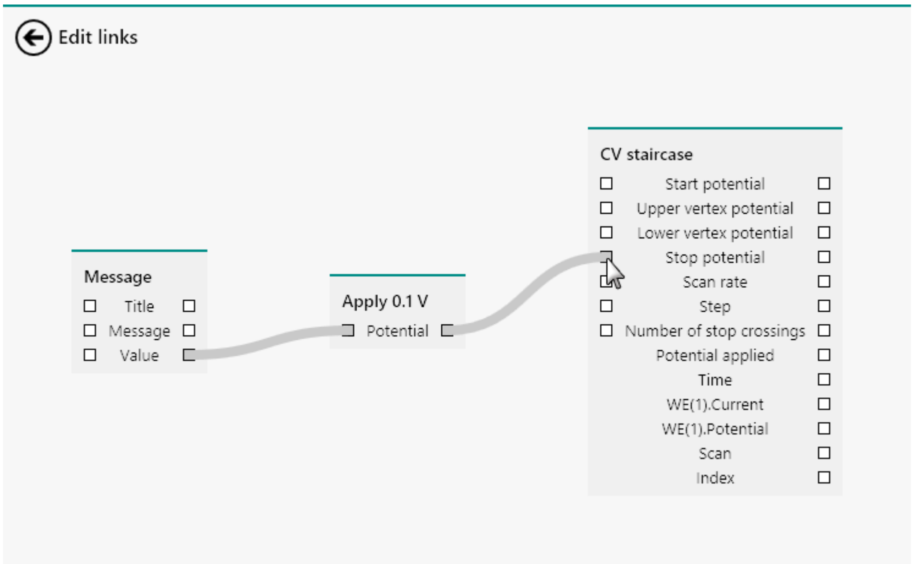


Figure 818 Changing the link to the Start potential property to the Stop potential property

It is also possible to delete a link, by clicking one end of a link and pulling it away from the anchoring point, as shown in *Figure 819*.

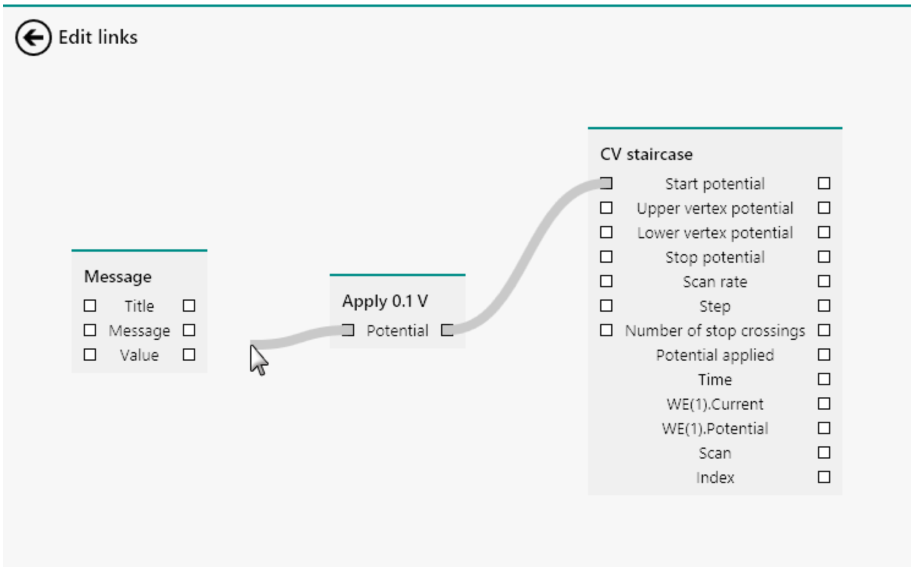


Figure 819 Removing a link

If the mouse button is released, the link will be removed *(see Figure 820, page 670)*.

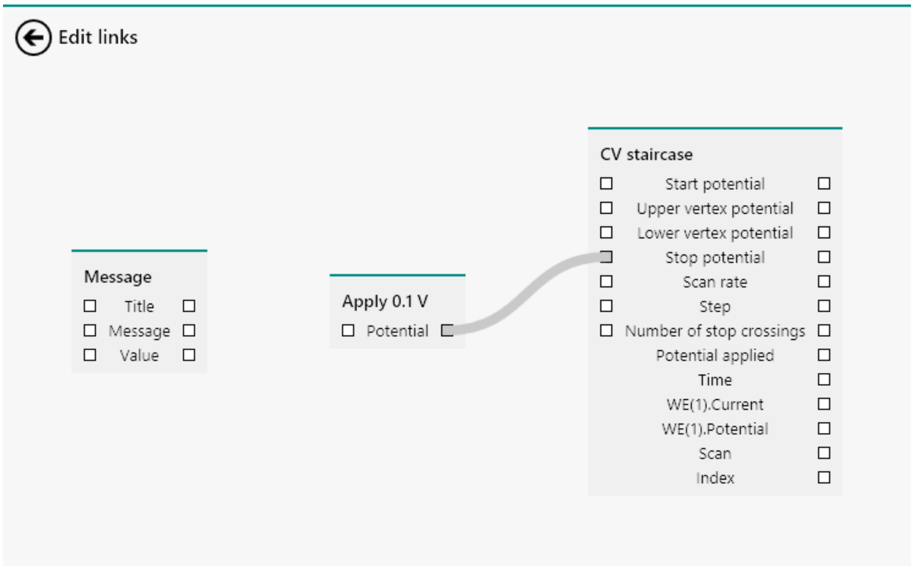


Figure 820 The link is removed

10.14 My commands

NOVA provides the means to save modified commands as new commands in order to facilitate reuse of frequently used commands. My commands are copies of default commands, which are saved in a dedicated location on the computer, alongside all the user-defined properties of the command. By default, My commands are saved in *\My Documents\NOVA 2.X\Commands*.

My commands appear in the dedicated group in the **Commands** panel *(see Figure 821, page 671)*.

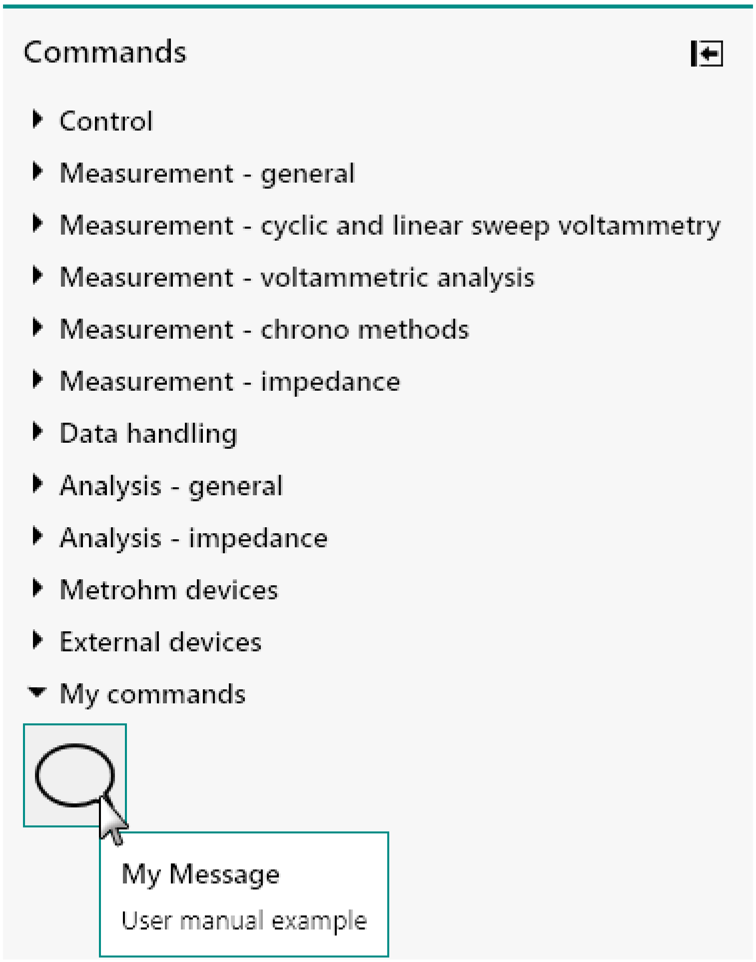


Figure 821 My commands are displayed in the Commands panel

NOTE

The My commands group of commands is only shown if it contains commands.

10.14.1 Saving a My command

To illustrate how to create a My command, the following starting procedure will be used *(see Figure 822, page 672)*. The modified **CV staircase** command will be saved in this example.

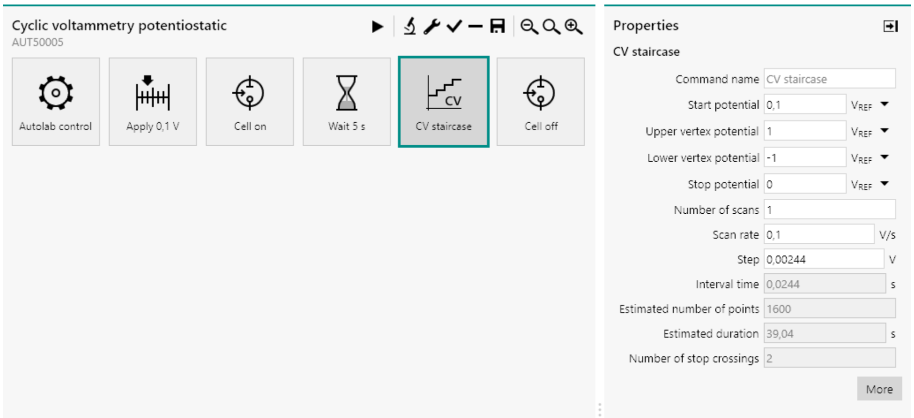


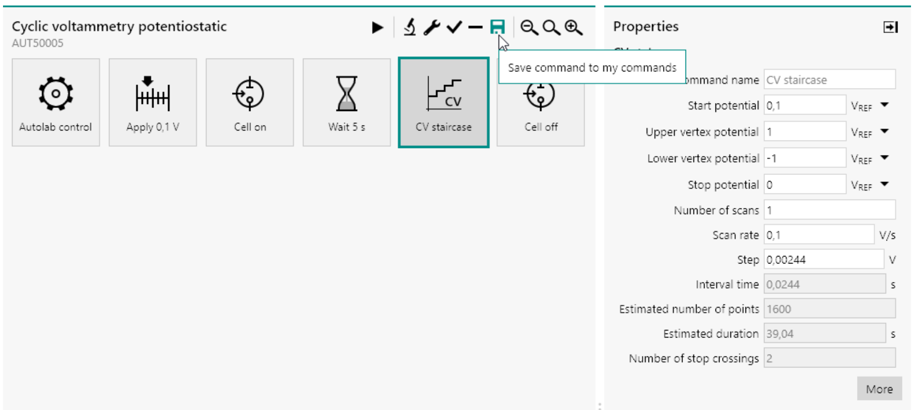
Figure 822 The initial procedure used to create a My command

1 Modify the source command

Modify the source command that will be saved as a My command.

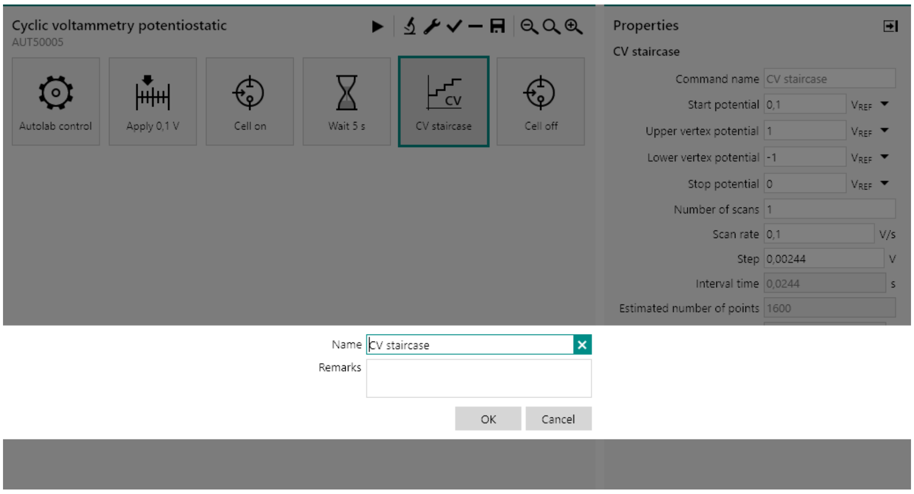
2 Save the command

Select the command to save and click the button in the top-right corner of the **Procedure editor** panel.



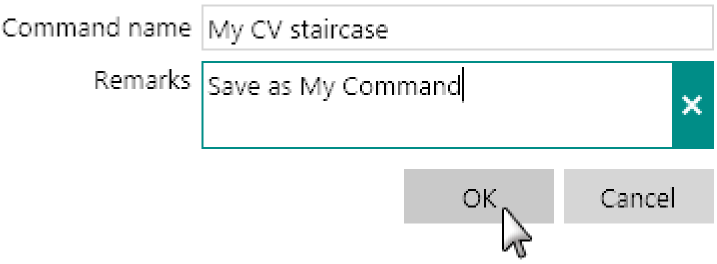
3 Specify name and remarks

An input dialog will be displayed.



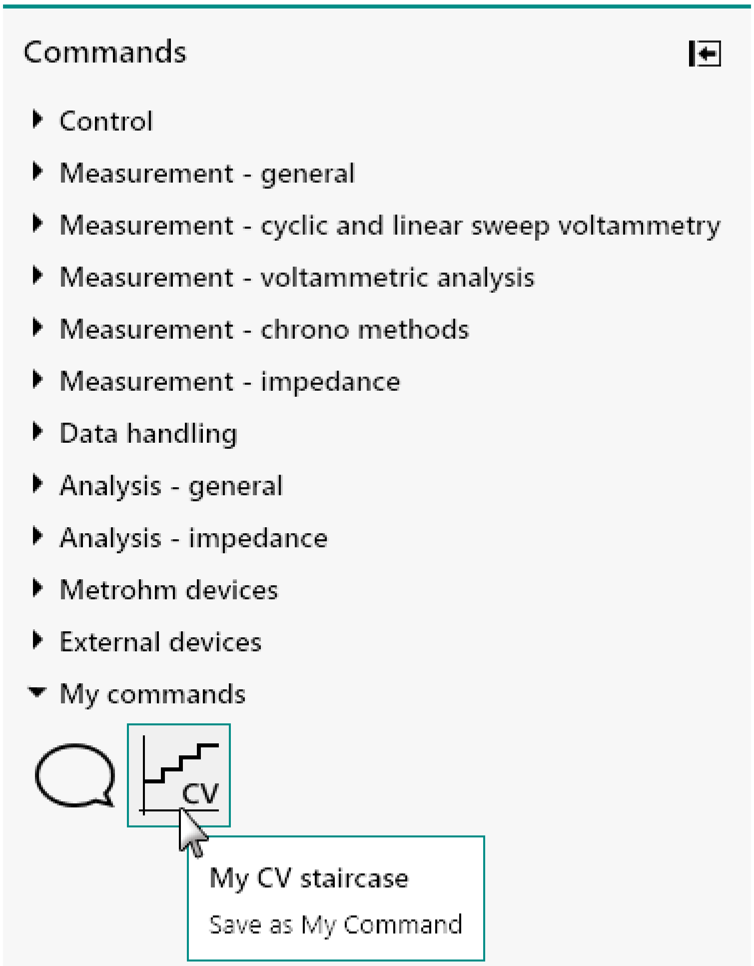
4 Validate the name and remarks

Click the OK button to validate the name and remarks and save the command in My commands.



5 The command is added to the group

The saved command is added to the My commands group.



10.14.2 Editing My commands

Commands that have been saved as My commands can be edited, exported or removed. All editing actions can be accessed by right-clicking the My command in the **Commands** panel and by selecting the required action from the context menu *(see Figure 823, page 675)*.

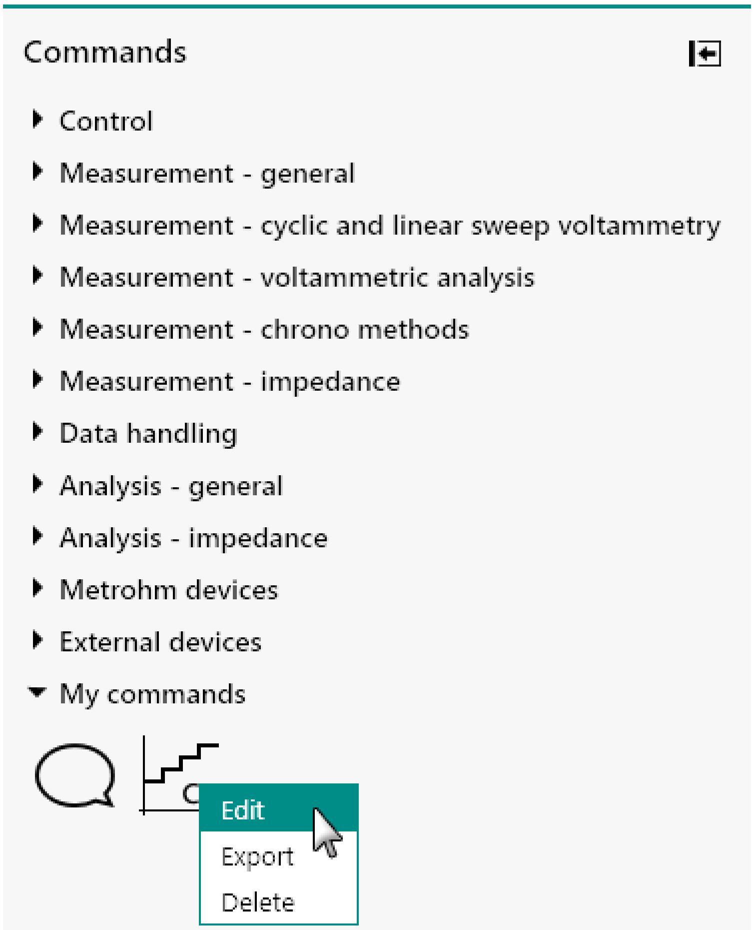


Figure 823 My commands can be edited, exported or deleted

Selecting the **Edit** option displays the name and remarks editor *(see Figure 824, page 675)*.

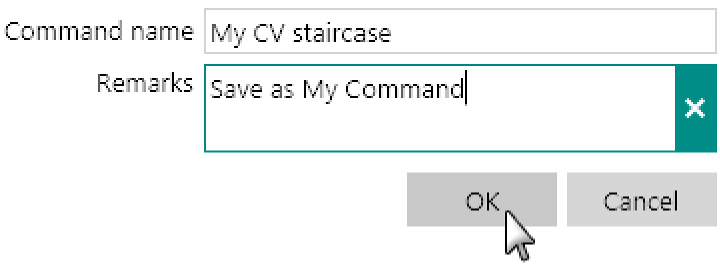


Figure 824 It is possible to adjust command name and remarks

Selecting the **Export** option displays a Windows Explorer dialog which can be used to specify a location and a file name for the My command. This command will be exported to the specified location with the specified name *(see Figure 825, page 676)*.

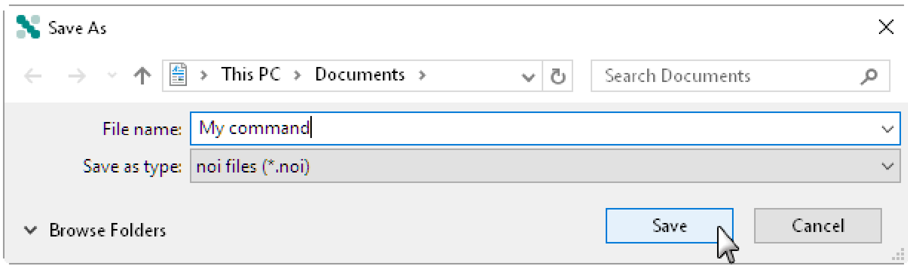


Figure 825 Specifying the name of the command file

NOTE

The extension used by My commands is *.noi*.

Selecting the **Delete** option removes the My command from the computer. A validation message will be displayed *(see Figure 826, page 676)*.

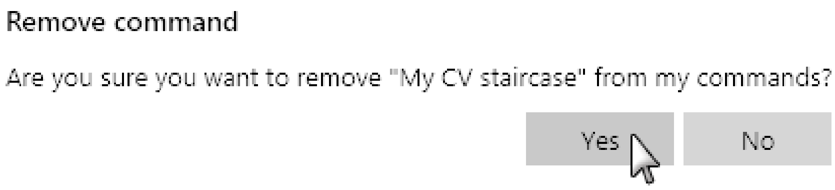


Figure 826 A confirmation message is shown

Click the Yes button to delete the command.

11 Running measurements

11.1 Starting procedure

When an open procedure is ready for an experiment it is possible to run it. To run an open procedure, first select the tab containing the procedure and then do one of the following:

- Click the run button ▶ in the top left corner of the procedure editor *(see Figure 827, page 677)*.

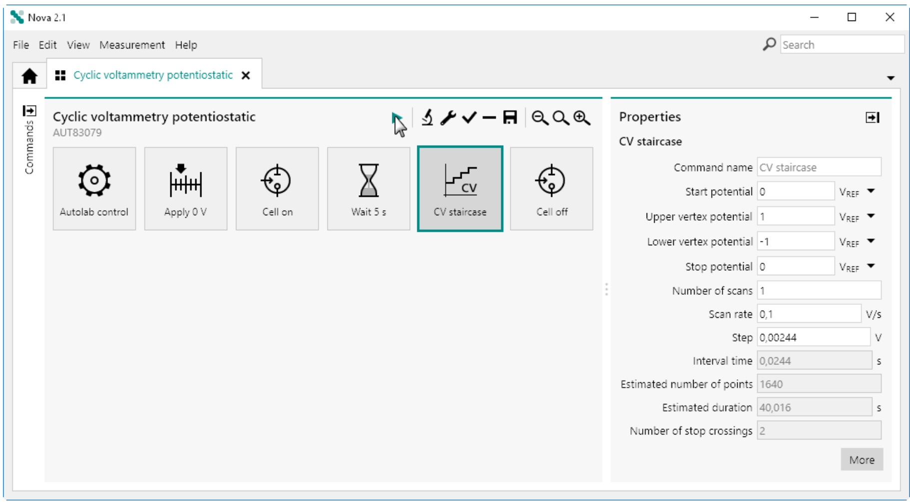


Figure 827 Starting a procedure using the provided button

- Press the **[F5]** shortcut key.
- Select the **Run** option from the **Measurement** menu *(see Figure 828, page 678)*.

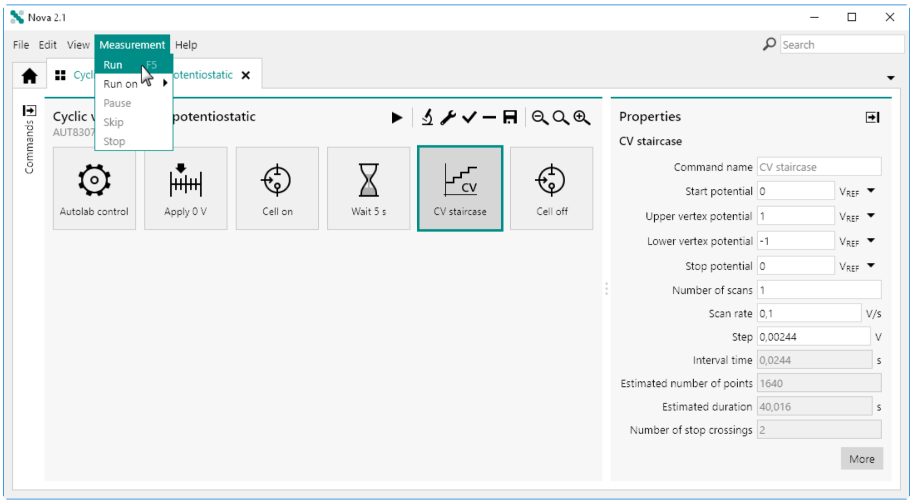


Figure 828 Starting a procedure from the Measurement menu

These three options will start the procedure using the **Default** instrument.

NOTE

The serial number of the **Default** instrument is shown in the top left corner of the procedure editor.

NOTE

It is possible to change the **Default** instrument *(see Chapter 5.1, page 84)*.

It is also possible to start a measurement on any available instrument by specifying on which instrument to run the procedure, using the Measurement menu *(see Figure 829, page 679)*.

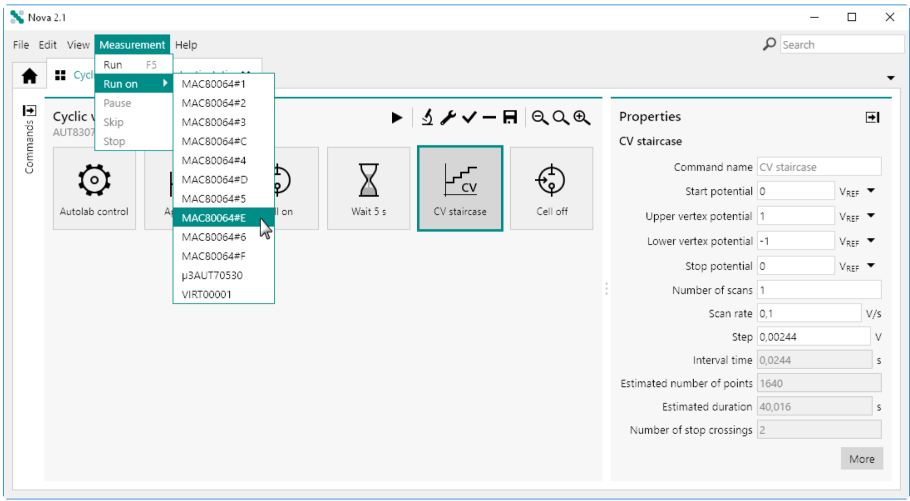


Figure 829 Specifying the instrument on which to run the measurement

When a procedure is started, the following tasks are carried out:

1. The procedure is tested for warnings or errors *(see Chapter 11.2, page 680)*.
2. A new tab opens, with the same name of the source procedure. A clone of the procedure is created in the new tab. The new tab will be used to record and display the measured data while the source procedure remains unchanged *(see Chapter 11.3, page 681)*.
3. A **Plots** frame will appear at the bottom of the screen. This frame will display all the measured data according to the properties defined in the procedure *(see Chapter 11.4, page 683)*.

During a measurement, it is also possible to carry out a number of actions:

1. It is possible to modify some of the measurement properties *(see Chapter 11.5.1, page 687)*.
2. It is possible to hold or stop the procedure and it is possible to skip the command being executed *(see Chapter 11.5.2, page 690)*.
3. It is possible to reserve the scan direction, if applicable *(see Chapter 11.5.3, page 691)*.
4. It is possible to display the instrument **Manual control** panel *(see Chapter 11.5.4, page 692)*.
5. It is possible to enable or disable plots *(see Chapter 11.5.5, page 694)*.

At the end of the measurement, the following tasks are carried out:

1. At the end of the measurement, the information displayed in the new tab will be time stamped for bookkeeping purposes.
2. Post validation is carried out at the very end and information or warning messages are shown, if applicable, indicating possible improvements of the procedure.

11.2 Procedure validation

Whenever a procedure is started, NOVA will verify the properties defined for each command in the procedure and test if these are compatible with the instrument the procedure is started on.

If a **Warning** is detected, the procedure will not start immediately and instead a message will be displayed to the user, providing information about the encountered **Warning** *(see Figure 830, page 680)*.

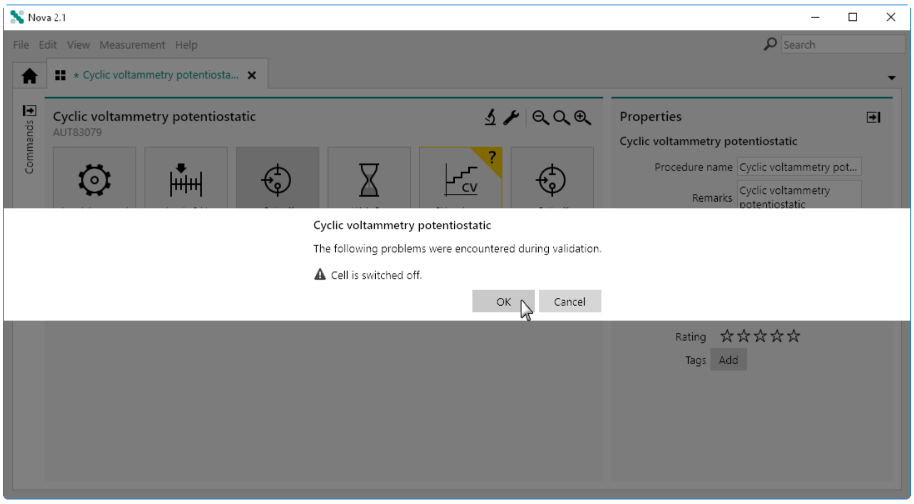


Figure 830 A Warning is detected

It is possible to click the OK button and ignore the **Warning** or click the Cancel button and return to the procedure editor to adjust the procedure.

NOTE

Ignoring a **Warning** is **not** recommended!

If an **Error** is detected, then the procedure will not be allowed to continue and a message will be displayed providing information on the **Error** *(see Figure 831, page 681)*.

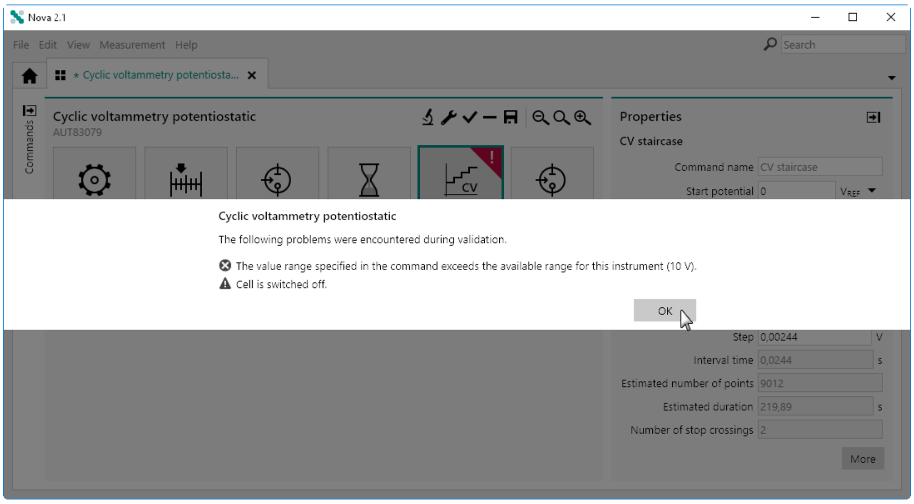


Figure 831 An Error is detected

It is then only possible to click the OK to close the message and return to the procedure editor.

NOTE

If **Errors** and **Warnings** are detected, the **Errors** are listed before the **Warnings** in the validation message, as shown in *Figure 831*.

If no **Warnings** or **Errors** are detected, the procedure is started and the measurement begins.

11.3 Procedure cloning

After validation, the procedure starts. A **clone** of the source procedure is created in a new **tab**. The new tab will have the same name as the tab contained the source procedure. Cloning the source procedure is convenient because it creates a new version of the original procedure that can be modified during the experiment. The source procedure remains unchanged in the original tab.

The procedure then starts in the new tab *(see Figure 832, page 682)*.

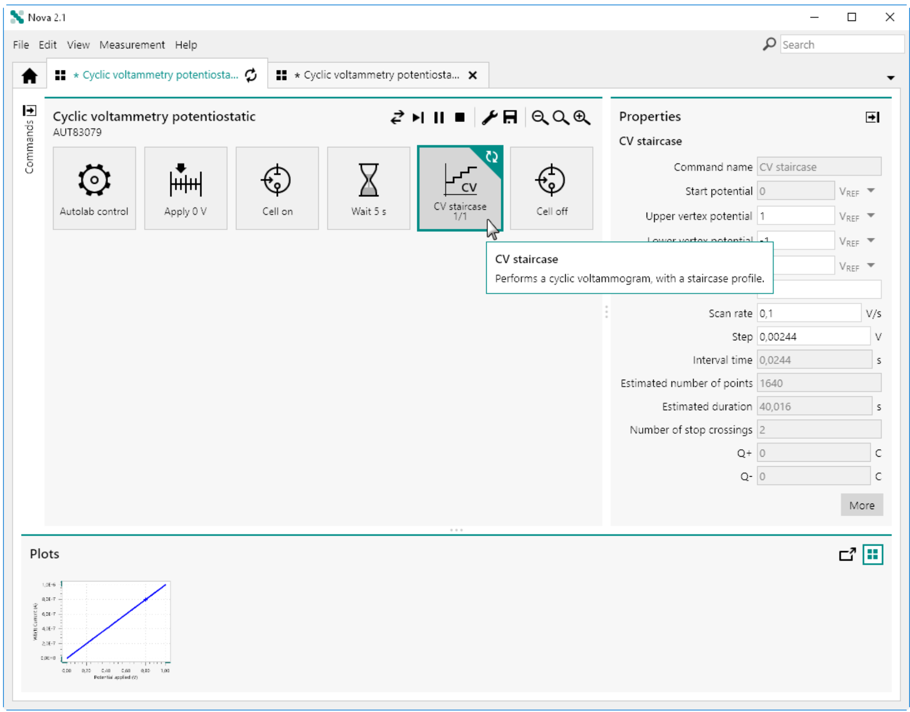


Figure 832 The procedure starts in a new tab

NOTE

The serial number of the instrument on which the procedure is started is reported below the name of the procedure.

The running state is indicated by the spinning wheel symbol, , shown in the tab as well as for the running command in the procedure *(see Figure 832, page 682)*.

The buttons located in the top right corner of the procedure editor of the running procedure can be used to either skip to the next command in the procedure (), pause the running command () or stop the whole procedure (). More information can be found in *Chapter 11.5.2*.

11.4 Plots frame

When a procedure starts, an additional **Plots** frame is opened at the bottom of the screen *(see Figure 833, page 683)*.

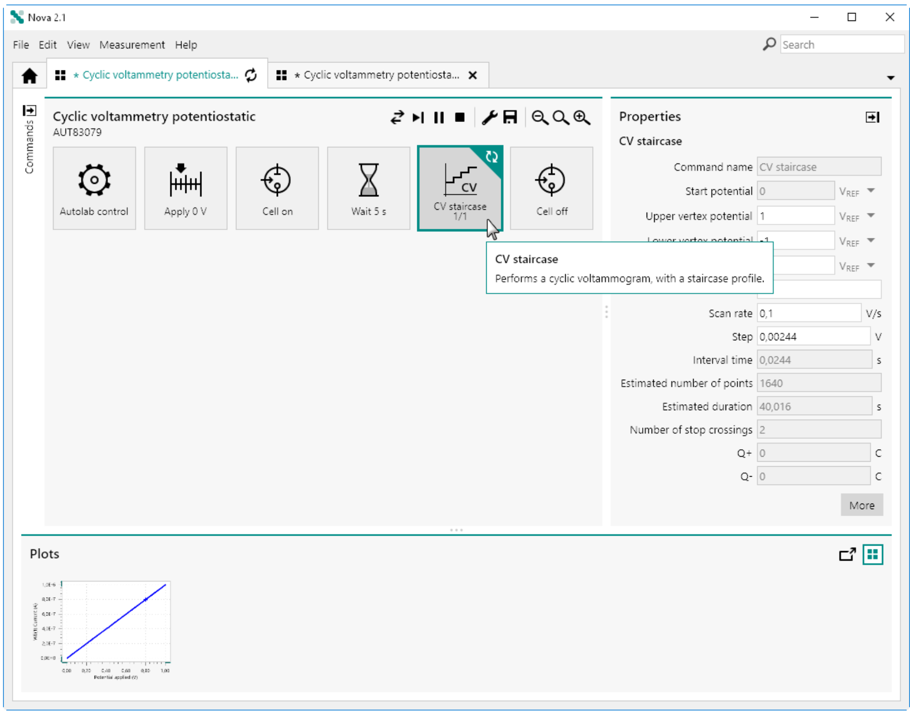


Figure 833 The Plots frame is created at the bottom of the screen

This frame is used for displaying plots during a measurement. All the plots defined in the procedure are created in the **Plots** frame. During the measurement, whenever data becomes available, the plots are populated with measured data points.

NOTE

Plots for which no data is available are shown in the **Plots** frame slightly greyed out. Whenever data becomes available for a plot, the plot will be shown normally.

The **Plots** frame can be resized to increase or decrease the size of the plots shown in the frame. It is also possible to undock the **Plots** frame, by clicking the button in the top right corner of the frame *(see Figure 834, page 684)*.

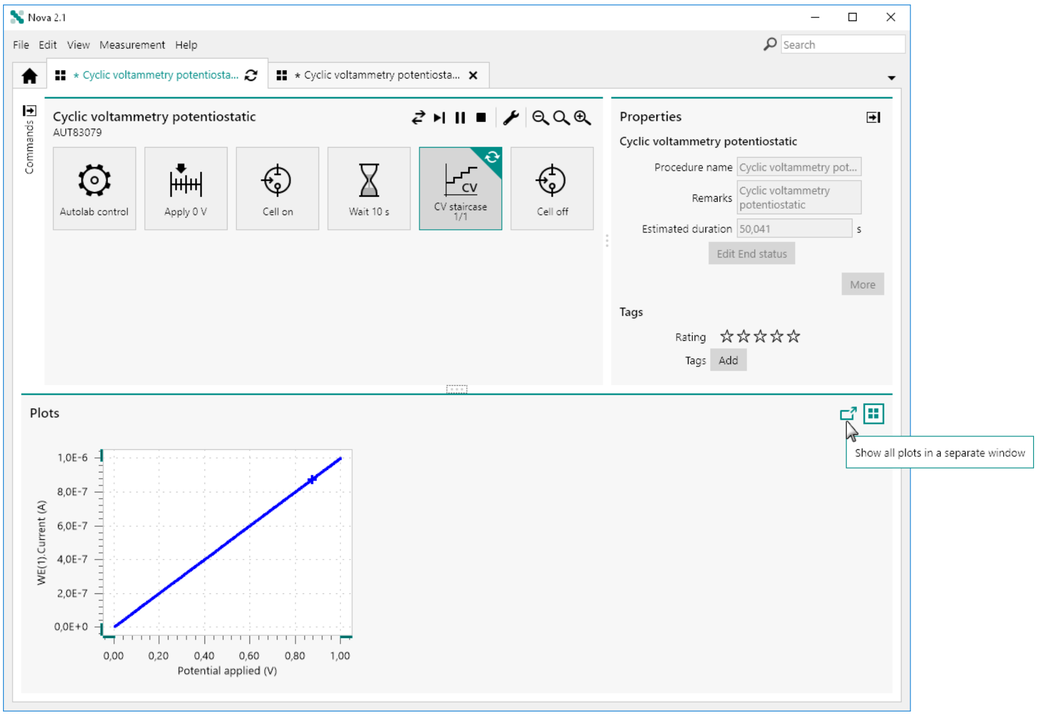


Figure 834 Undocking the Plots frame

A new window will be created, displaying the contents of the **Plots** frame. Zooming in and out buttons are provided in the top right corner to increase or decrease the size of the plots in the window *(see Figure 835, page 685)*.

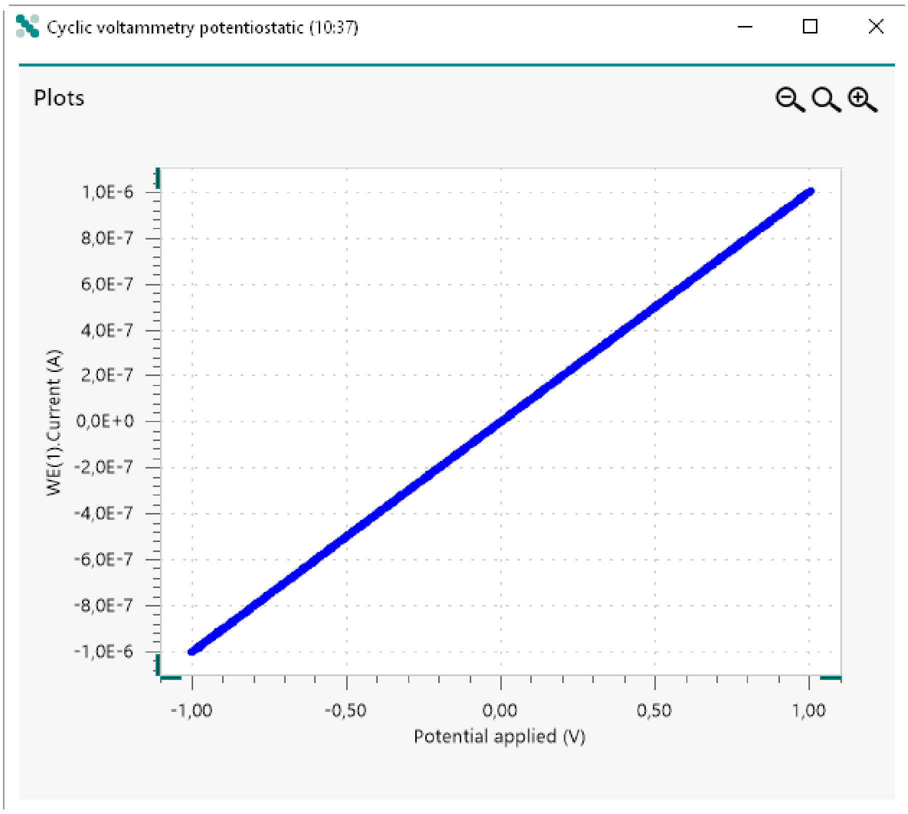


Figure 835 The undocked Plots frame

NOTE

The undocked **Plots** frame can be closed at any time.

11.4.1 Displaying multiple plots

When a procedure generates multiple plots, these plots can be arranged in two different ways:

- **Sequence arrangement:** all the plots defined in the procedure are shown in sequence in the **Plots** frame scaled to the largest available space. If more plots are defined than can be arranged in the Plots frame, a scrollbar will be added to the frame. Using this scrollbar, it is possible to change the plots shown in the frame *(see Figure 836, page 686)*.

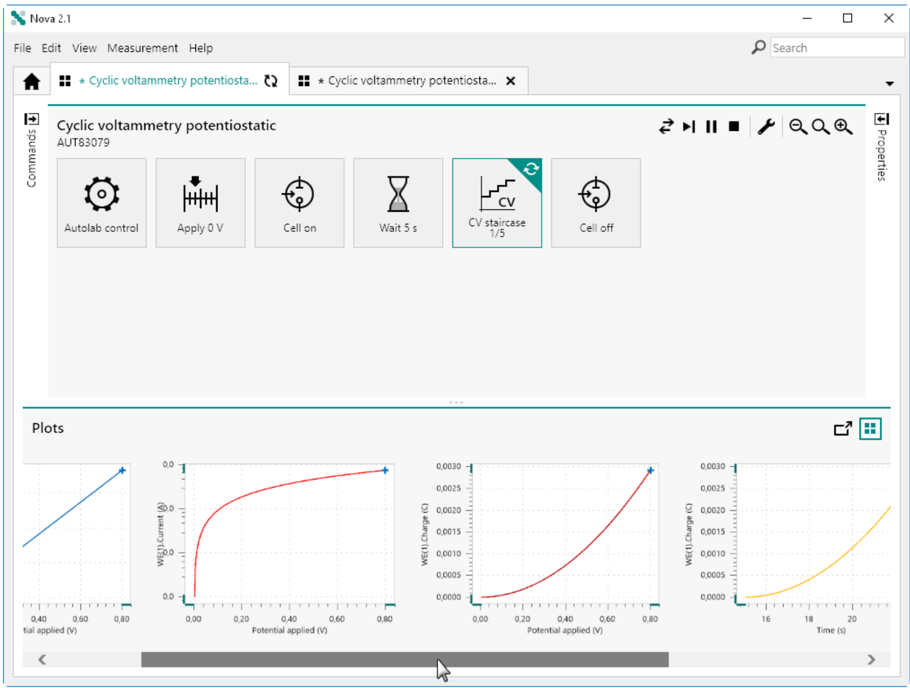


Figure 836 Plots shown in sequence arrangement

- **Tiled arrangement:** all the plots defined in the procedure are shown in the **Plots** frame and are shrinked to size required to show each plot in the frame. No scrollbar is added to the Plots frame in this case *(see Figure 837, page 686)*.

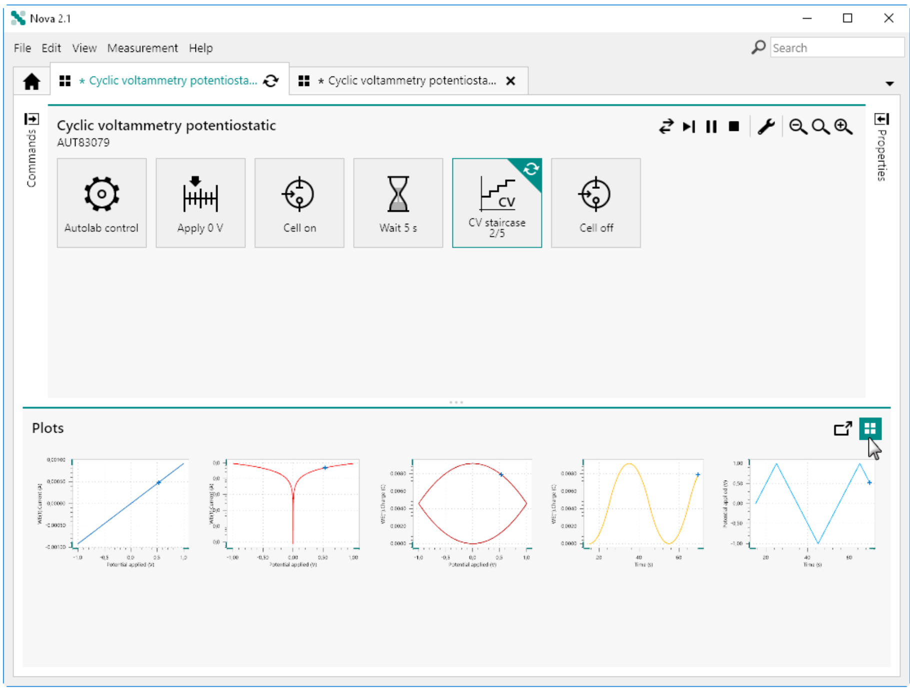


Figure 837 Plots displayed in tiled arrangement

NOTE

It is possible cycle between the sequence arrangement or the tiled arrangement at any time by clicking the ⊞ button in the top right corner of the **Plots** frame.

11.5 Real time modifications

NOVA provides controls that can be used to control a running procedure in real time. The following modification are allowed:

1. Modification of some properties of commands used in the procedure.
2. Direct control of the running measurement by means of dedicated buttons in the procedure editor.
3. For some measurement commands, it is possible to reverse the scan direction.
4. Manual control panels can be displayed and interacted with during a measurement.
5. Plots can be enabled, disabled or modified during a measurement.

Additionally, NOVA may provide feedback during a measurement based on the experimental data recorded by a command in the procedure.

11.5.1 Real-time properties modification

While a measurement is running, it is possible to modify some of the properties of the commands in the procedure. To modify a property of a command during a measurement, click the command in the procedure editor. The properties that can be modified will the displayed in the **Properties** panel *(see Figure 838, page 688)*.

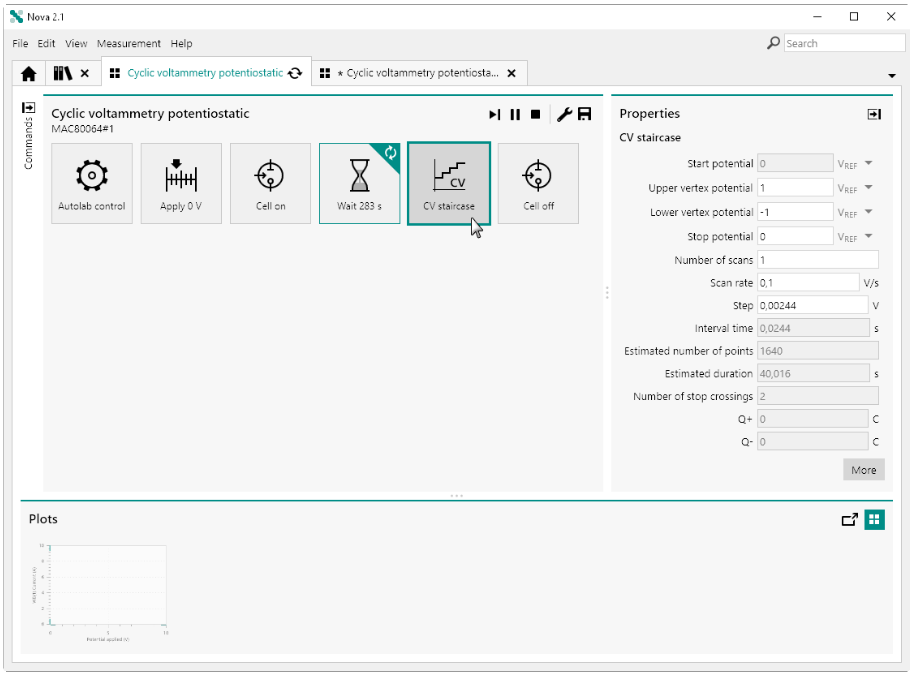


Figure 838 Selecting a command during a measurement shows the available properties

NOTE

The greyed out properties cannot be modified in real-time.

NOTE

Changing the properties of a command that has already been executed is possible but will not have any effect on the running procedure. This modification may however become active when the measured data set is converted to a new procedure, as explained in *(see Chapter 11.10, page 724)*.

It is possible to specify a new value for one or more of the available properties *(see Figure 839, page 689)*.

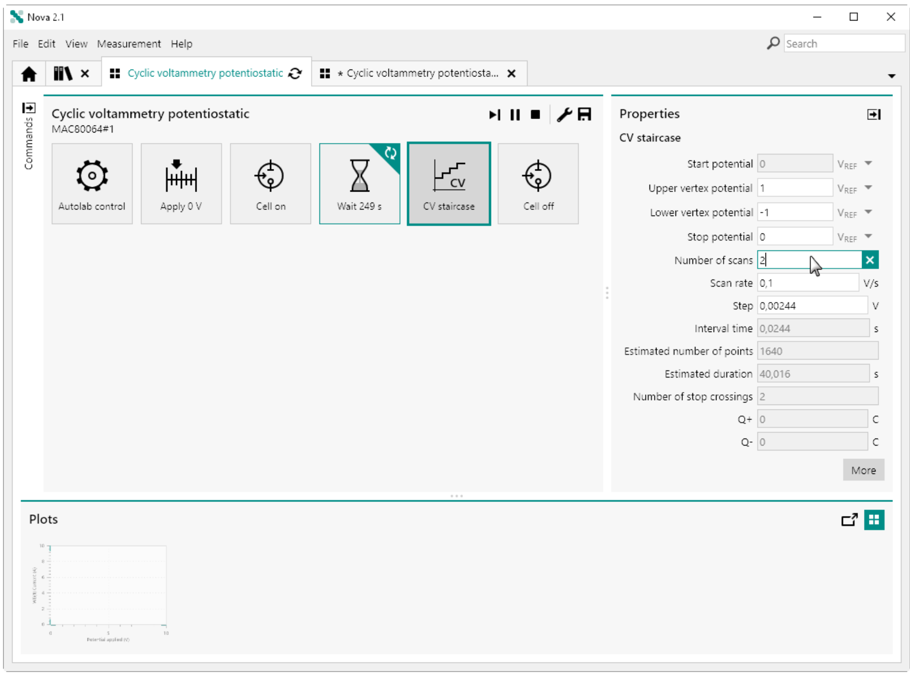


Figure 839 Modifying the number of scans

A new value will be validated by pressing the **[Enter]** key or by clicking away from property value being edited. The new value will be validated before becoming active. If the new value is not acceptable for the edited property, it will be displayed with a red frame around it, indicating that it is invalid *(see Figure 840, page 689)*.

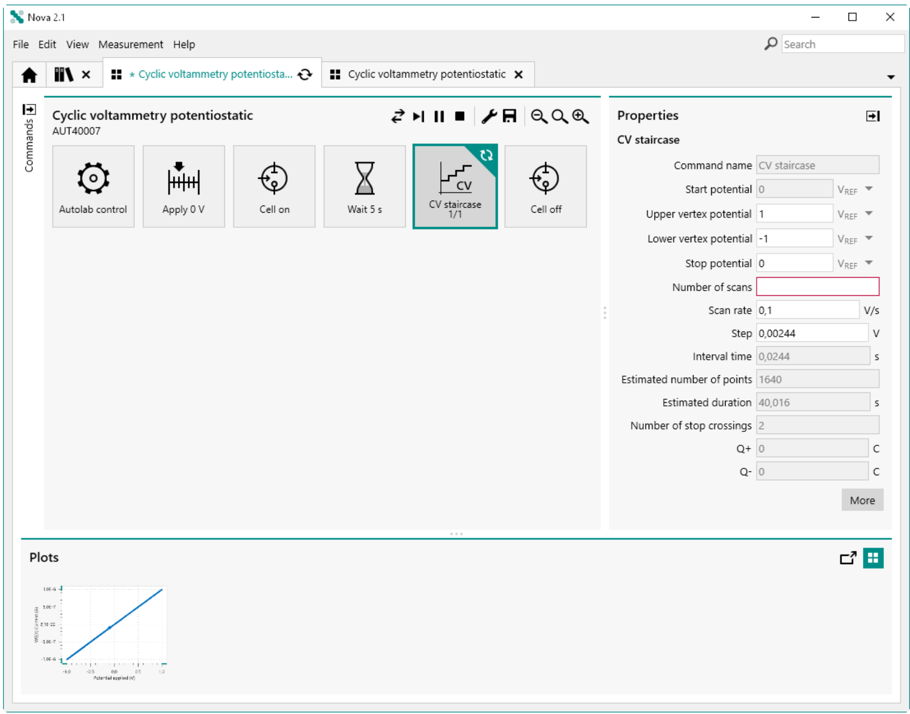


Figure 840 New properties are validated before becoming active

If the new value is valid, it will be updated in the running procedure and used in the applicable command instead of the original value *(see Figure 841, page 690)*.

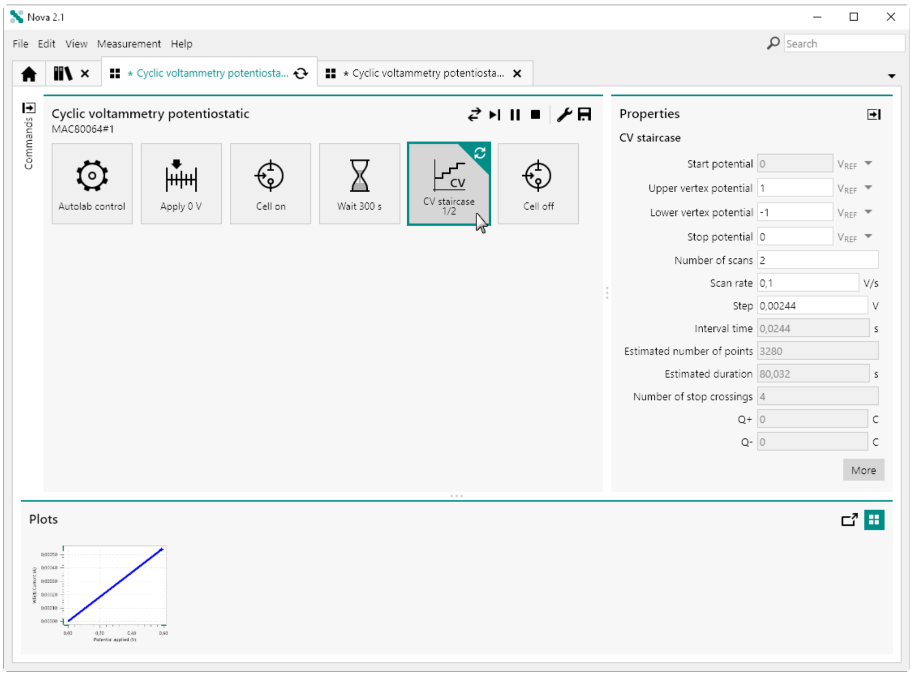


Figure 841 The new property is used during the measurement

NOTE

Modifying procedure properties in real-time does not affect the source procedure from which the procedure was started.

NOTE

All real time modifications of measurement properties are logged into the data grid and stored in the data file.

11.5.2 Procedure control

The buttons located in the top right corner of the procedure editor of the running procedure can be used to either skip to the next command in the procedure (▶▌), pause the running command (▌▌) or stop the whole procedure (■). The procedure editor will update the status of a command affected by these controls, if applicable *(see Figure 842, page 691)*.

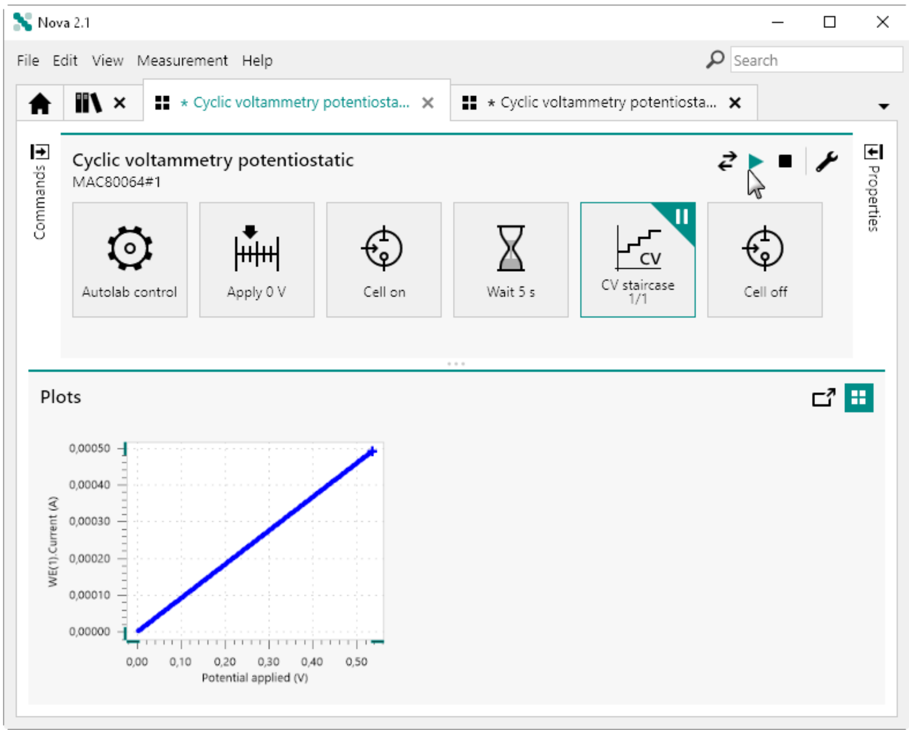


Figure 842 Holding the CV staircase command

NOTE

All interactions with the procedure controls buttons are logged into the data grid and stored in the data file.

11.5.3 Reverse scan direction

NOTE

This option is only available for the **CV staircase** command and the **LSV staircase** command.

During a measurement, it may be possible to modify the scan direction by clicking the ⇄ button in the top right corner of the procedure editor. This button is only shown while the command that supports this option is running *(see Figure 843, page 692)*.

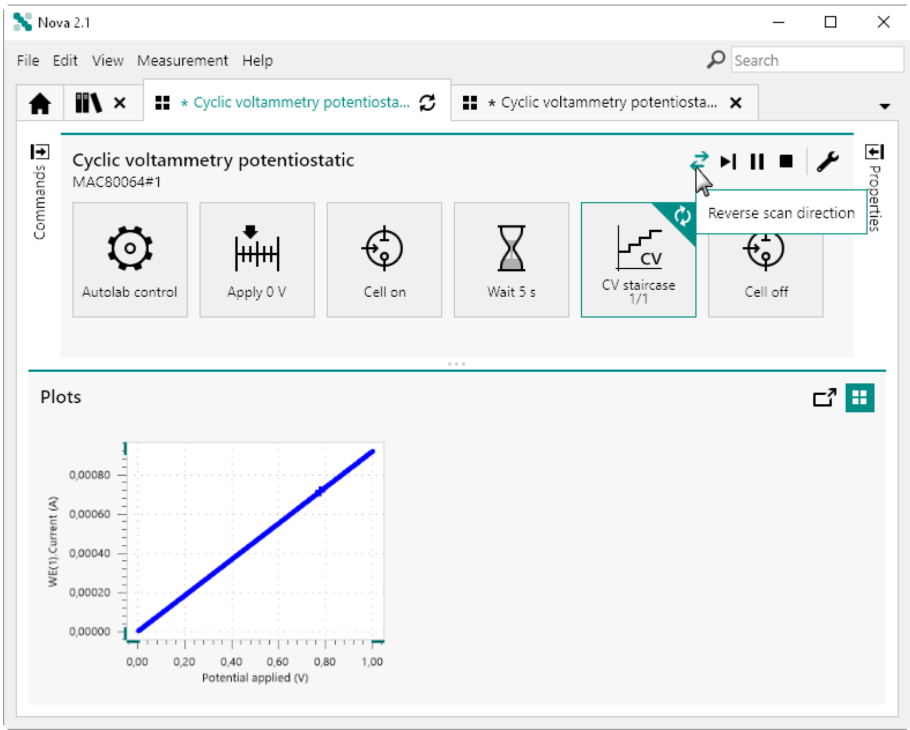


Figure 843 *Reversing the scan direction*

If the scan direction is reversed, the command will continue running until the requirements for that command specified by the user are fulfilled.

NOTE

All interactions with the reverse scan button are logged into the data grid and stored in the data file.

11.5.4 Display the Manual control panel

At any time during a measurement, it is possible to display the **Manual control** panel of the instrument involved in the measurement. This can be done by selecting the *Manual control* option from the **View** menu or by pressing the **[F10]** shortcut key *(see Figure 844, page 693)*.

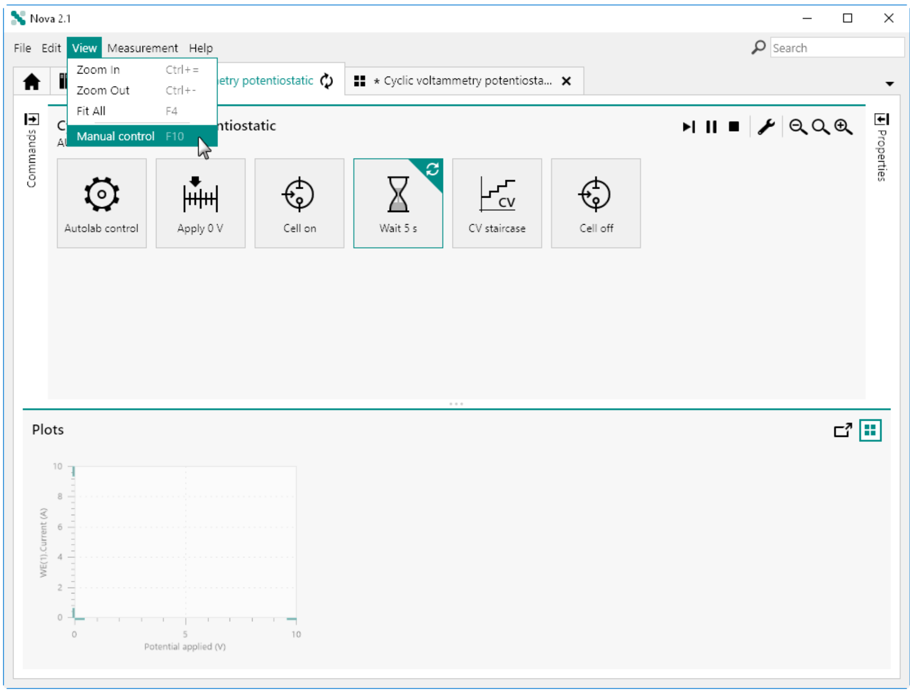


Figure 844 Displaying the instrument manual control

The **Manual control** panel can be used to modify some of the hardware controls during a measurement *(see Figure 845, page 693)*.

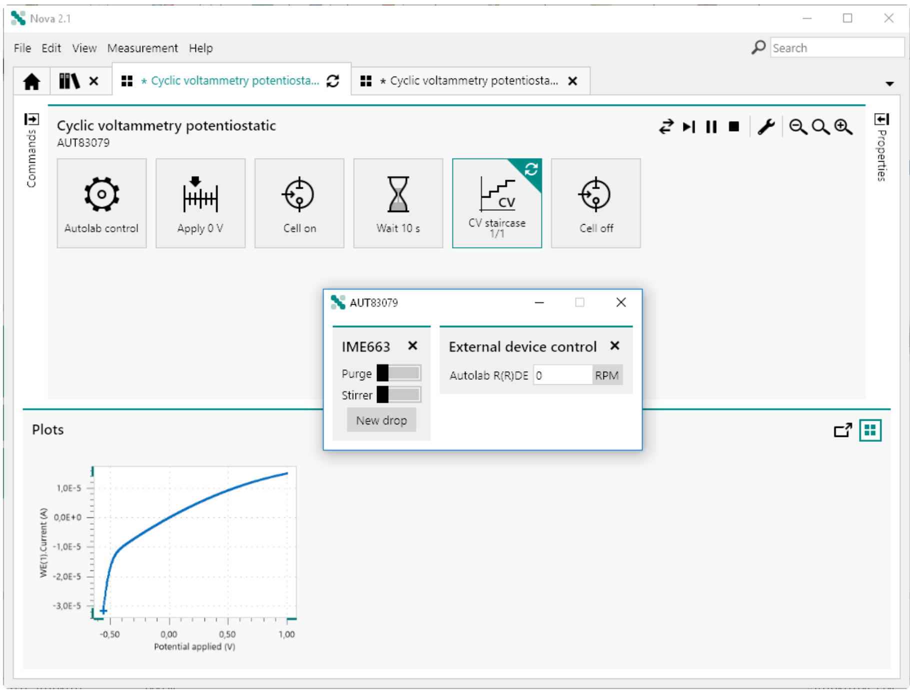


Figure 845 The Manual control panel can be used to modify instrument settings during a measurement

11.5.5 Enable and disable plots

While a procedure is running, it is possible to click the More button in the command **Properties** panel or to double click a measurement command in the procedure to adjust the plot settings *(see Figure 846, page 694)*.

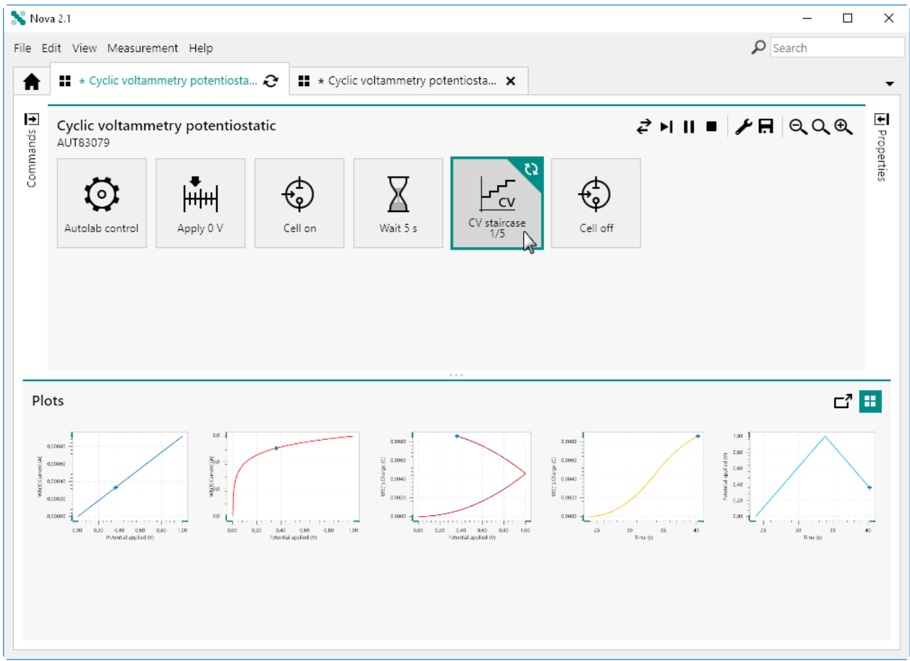


Figure 846 Double click a measurement command to adjust the plots shown in the Plots frame

A new screen will be shown, presenting controls that can be used to adjust the plots visibility *(see Figure 847, page 695)*.

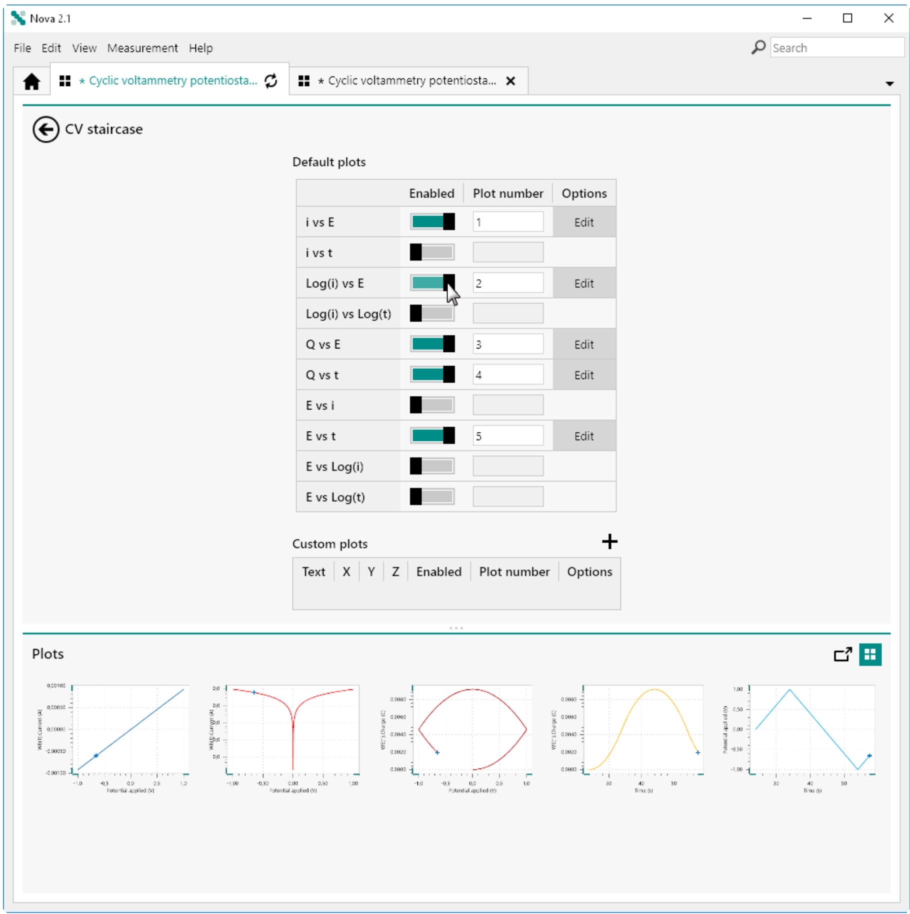


Figure 847 *Plots can be enabled or disabled at any time during a measurement*

NOTE

The screen shown in *Figure 847* is the same as the one shown in *Figure 752*, without the **Sampler** and the **Options**, which cannot be modified in real-time.

In this screen, it is possible to disable pre-defined plots or to enable new plots, if needed, using the provided toggles. It is possible to disable a pre-defined plot in the **Plots** frame directly by right-clicking a plot to disable and selecting the corresponding option from the context menu *(see Figure 848, page 696)*.

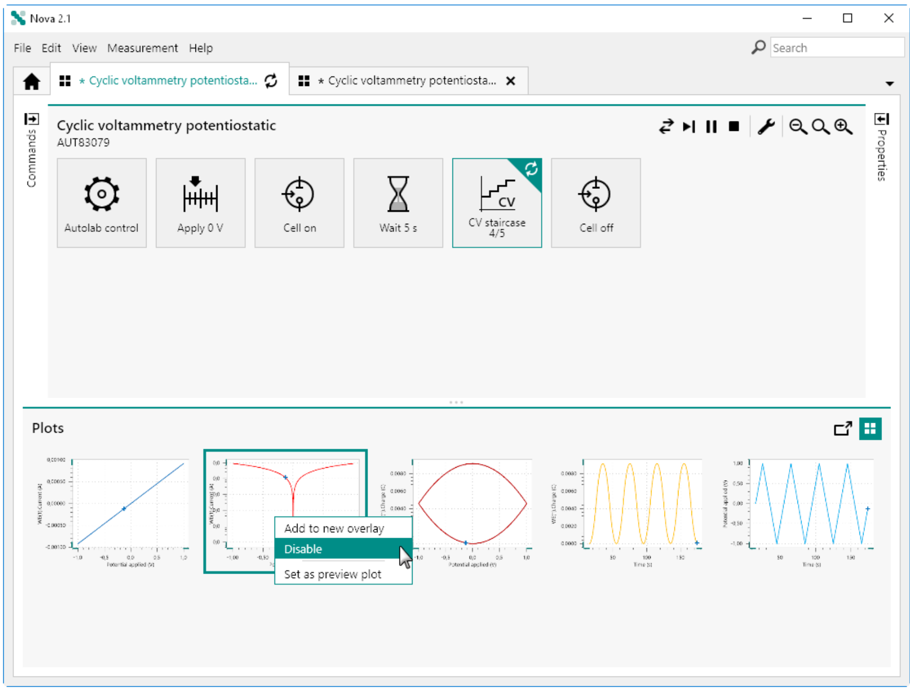


Figure 848 Quickly disabling a plot in the Plots frame

The plot will be removed from the **Plots** frame *(see Figure 849, page 696)*.

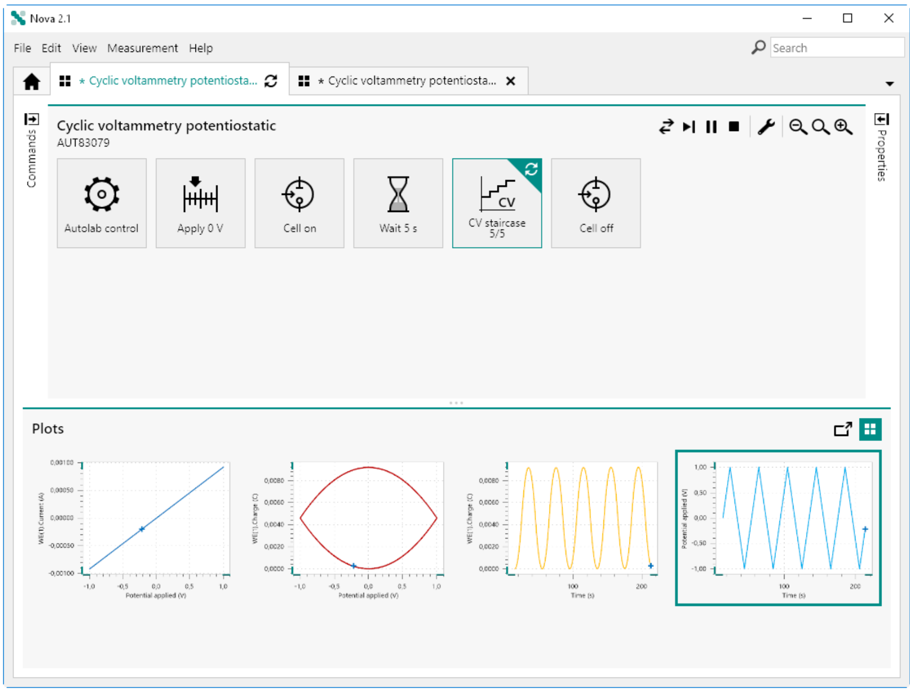


Figure 849 The selected plot is disabled

NOTE

Disabled plots can be enabled again using the method described at the beginning of this Section *(see Figure 847, page 695)*.

11.5.6 Q+ and Q- determination

NOTE

This option is only available for the **CV staircase** command.

During the execution of the **CV staircase** command, after each scan is completed, the anodic and cathodic charge (Q+ and Q-) is automatically determined from each cyclic voltammogram and reported in the **Properties** panel *(see Figure 850, page 697)*.

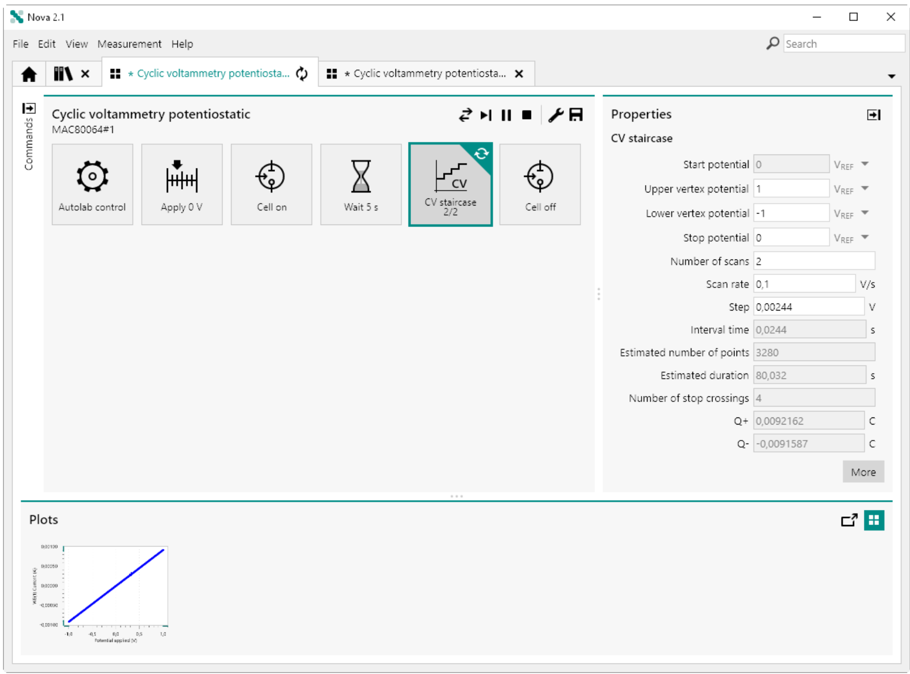


Figure 850 The values of Q+ and Q- are automatically added to the Properties panel

The Q+ and Q- values are determined at the end of each scan. These values are reported in C.

NOTE

The values of Q+ and Q- are also saved alongside the other electrochemical signals sampled during the measurement.

11.6 End of measurement

When a measurement finishes, the measured data becomes available for evaluation and analysis. Depending on the settings defined in the NOVA **Options**, the data may or may not be saved automatically *(see Chapter 1.9, page 13)*.

NOVA will also carry out the following activities at the end of each measurement:

1. **Time stamping:** the measured data is time stamped using the time and date of the beginning of the measurement *(see Chapter 11.6.1, page 698)*.
2. **Post validation:** the measured data is evaluated and information or warnings are provided, if applicable *(see Chapter 11.6.2, page 699)*.

11.6.1 Procedure time stamp

At the end of measurement, the procedure is issued a **time stamp**. The time stamp corresponds to the starting time of the measurement *(see Figure 851, page 699)*.

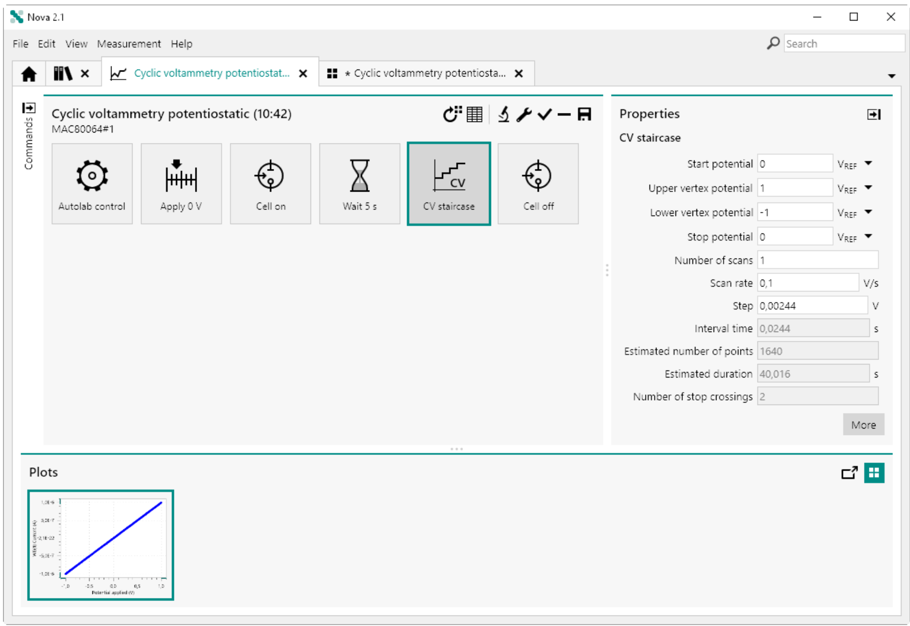


Figure 851 The procedure is time stamped at the end of each measurement

The time stamp is formatted as "Day/Month/Year Hour:Minute".

NOTE

The Day/Month/Year part of the time stamp is only shown if the current day is different from the day of the procedure time stamp.

11.6.2 Post validation

At the end of measurement, the procedure is tested for possible information or warnings. If the experimental conditions used by one of the commands in the procedure can be improved, that command will be highlighted in blue and more information will be provided in the tooltip *(see Figure 852, page 700)*.

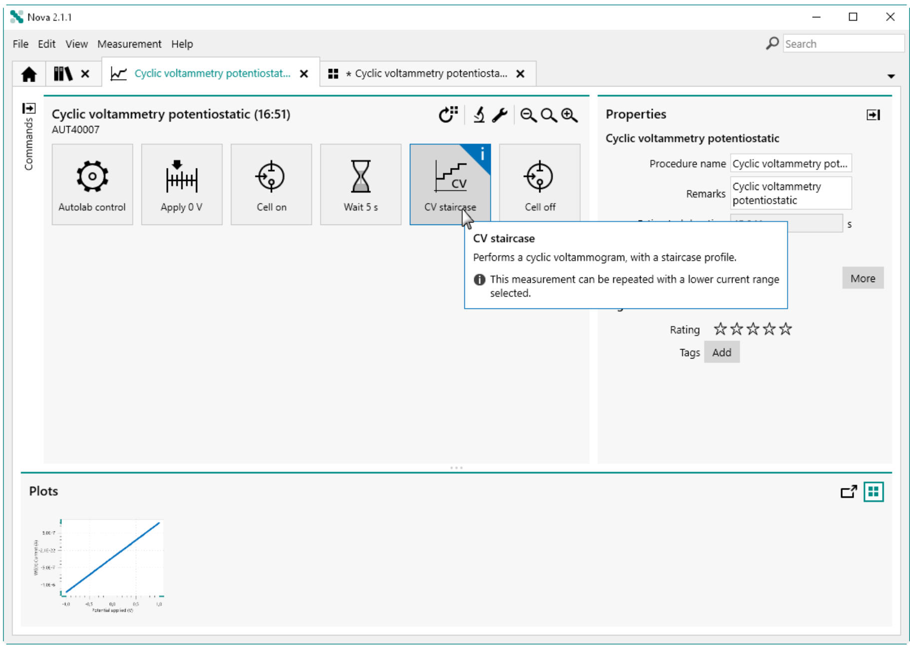


Figure 852 A post validation information message

If a warning is detected after the measurement is finished, the command for which the warning was detected will be highlighted yellow and the tooltip will provide the details of the warning *(see Figure 853, page 700)*.

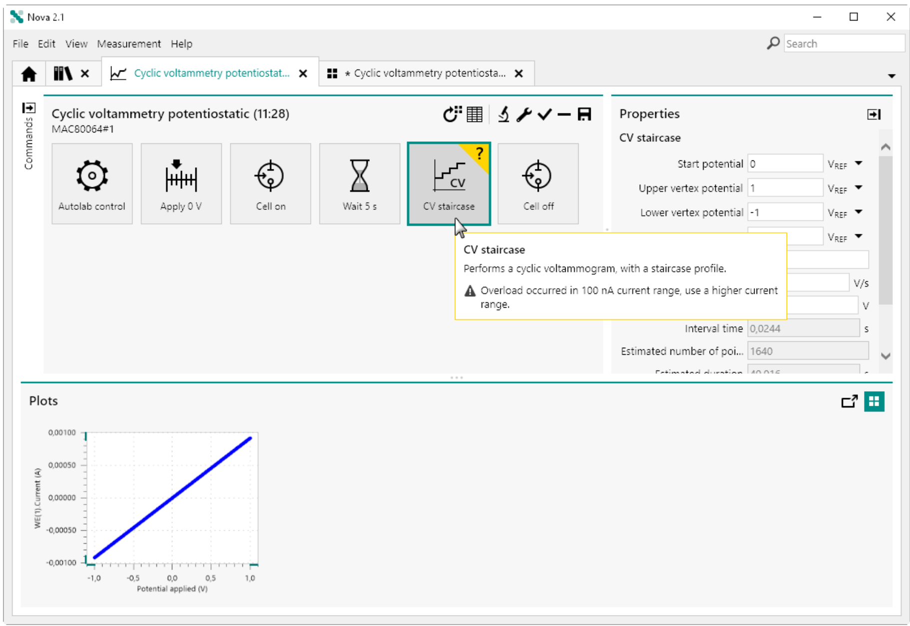


Figure 853 A post validation warning message

Post validation messages generally provide indications which can be used for finetune the measurement conditions.

11.7 Specify plot preview

Whenever a data set is saved in the **Library**, a plot preview is created. This plot preview can be displayed in a tooltip in the **Library** to provide a preview of the data as shown in *Chapter 6.9*.

By default, the first plot in the **Plot** frame is used as a plot preview, however it is possible to specify another plot as the preview plot at any time. To change the plot preview, right-click the plot to use and select the *Set as preview plot* from the context menu *(see Figure 854, page 701)*.

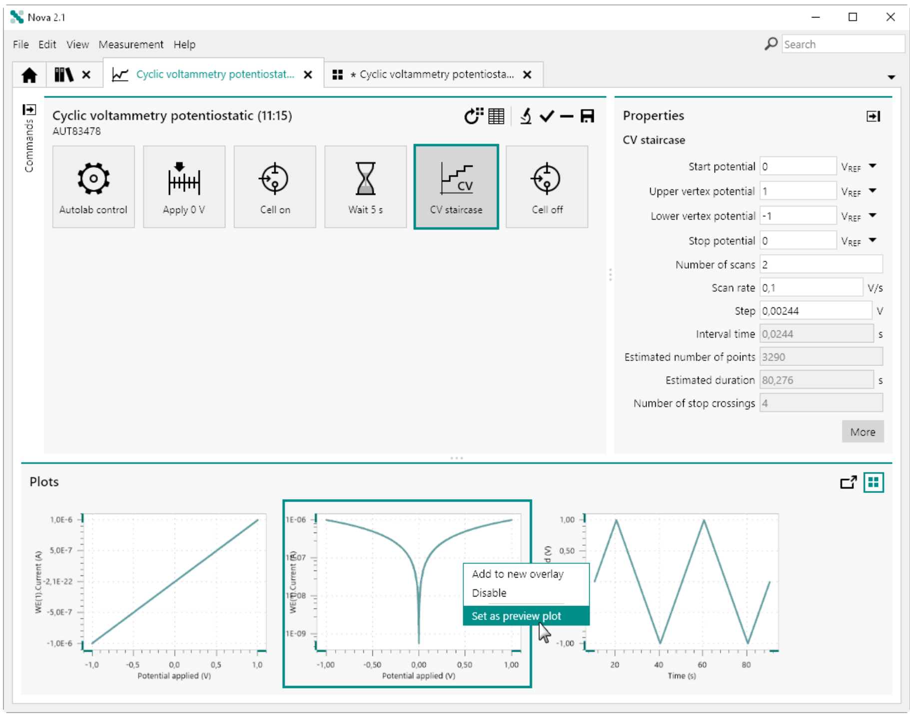


Figure 854 Specifying the plot preview

> **NOTE**
>
> The new plot preview will be updated when the modifications to the data file are saved.

11.8 Detailed plot view

It is possible to double click a plot shown in the **Plots** frame to obtain a larger view of the plot, change some of the plot properties or toggle to a 3D view of the plot, if available. The detailed view of the plot replaces the procedure editor view *(see Figure 855, page 702)*.

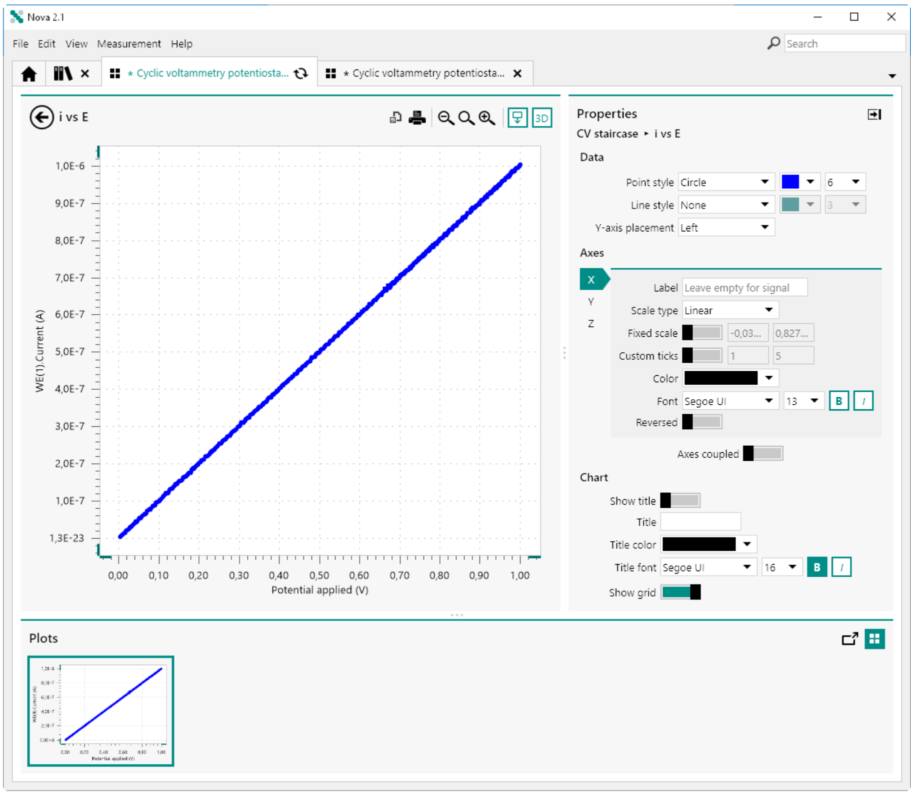


Figure 855 Detailed view of a plot

The detailed plot view provides the following controls:

- **Plot panel:** a large panel showing the selected plot. A number of button are located in the top right corner of this frame to add a data analysis command, view the data marker or toggle the 3D view on or off.
- **Properties panel:** a panel that can be used to change the plot properties during the measurement. This panel can be collapsed if necessary, by clicking the button.

Clicking the button closes the detailed plot view and returns to the procedure editor.

11.8.1 Plot properties

The **Properties** panel, shown in the right hand side of the screen, can be used to modify the plot properties of the active plot at any time *(see Figure 856, page 703)*.

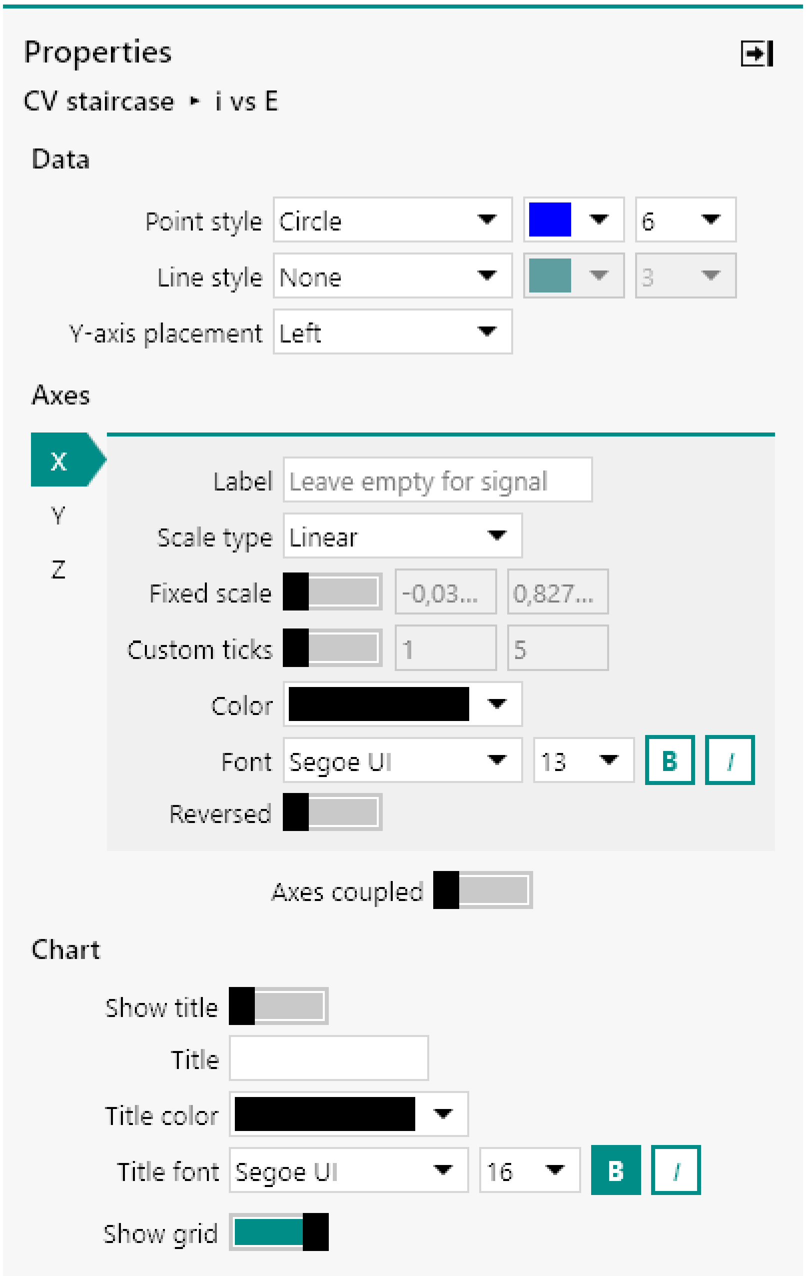


Figure 856 The Properties panel can be used to adjust the active plot

NOTE

The properties shown in the **Properties** panel are the same as the properties available for default and custom plots *(see Chapter 9.5.3, page 615)*.

11.8.2 Toggle the 3D view

Clicking the 3D button in the top right corner of the **Plot** panel toggles the 3D view on or off *(see Figure 857, page 704)*.

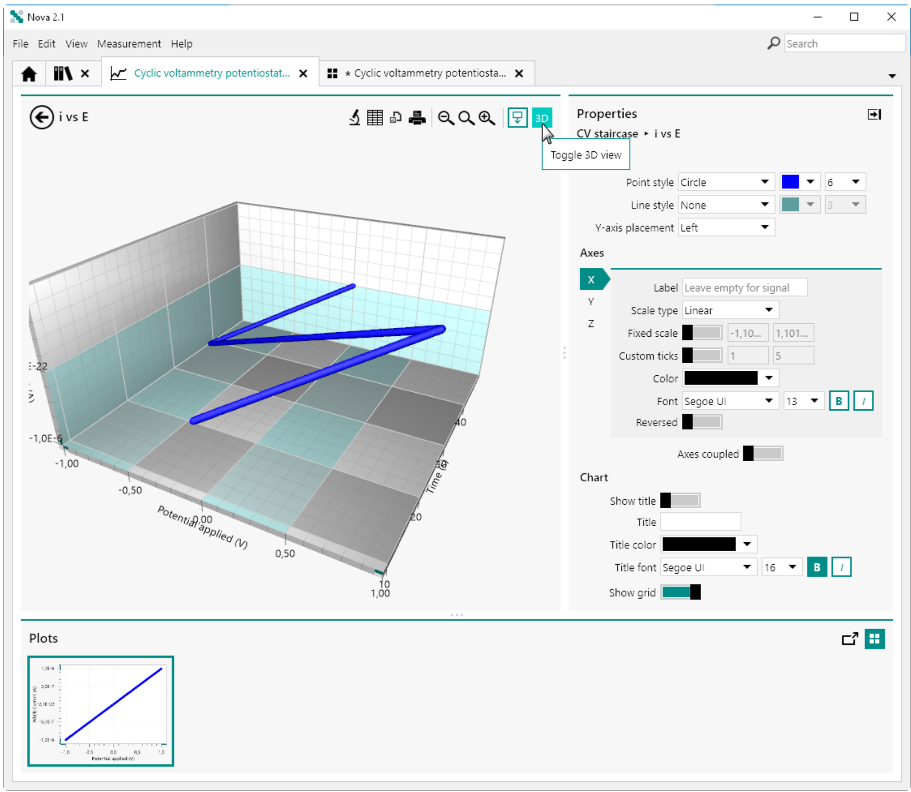


Figure 857 Toggling the 3D view on or off

The 3D view shows the same data using one additional Z axis. The plot can be rotated using by clicking and dragging the mouse.

NOTE

It is only possible to display the data in 3D when a signal that can be plotted in real-time has been assigned to the Z axis of the plot.

11.8.3 Toggle the step through data mode

Clicking the button in the top right corner of the **Plot** panel toggles the *Step through data* mode on or off *(see Figure 858, page 705)*.

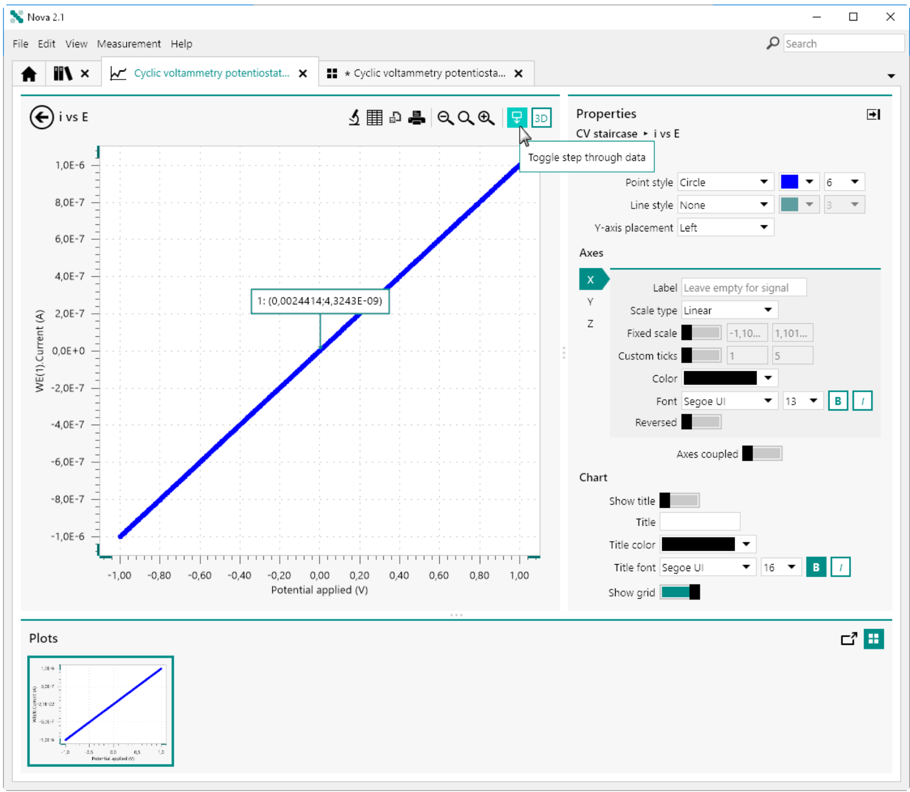


Figure 858 Toggling the step through data mode on or off

When the *Step through data* mode is on, an additional indicator is added to the plot, showing the X and Y coordinates of the point indicated by the arrow, in the case of a 2D plot, and the X, Y and Z coordinates of the point indicated by the arrow, in the case of a 3D plot.

NOTE

The indicator is always shown for the first data point of the plot.

Using the mouse, it is possible to perform the following action (2D plot):

- Click anywhere in the plot area: the indicator is relocated to the closest data point of the plot.

Using the keyboard it is possible to perform the following actions (2D and 3D plot):

- **[←]**/**[→]**: the indicator can be moved by 1 point at a time.

- **[←]/[→]** and **[CTRL]**: the indicator can be moved by 10 points at a time.
- **[←]/[→]** and **[CTRL]** and **[SHIFT]**: the indicator can be moved by 100 points at a time.

11.8.4 Add an analysis command

Clicking the button in the top right corner of the **Plot** panel displays a popout menu from which an analysis command can be selected *(see Figure 859, page 706)*.

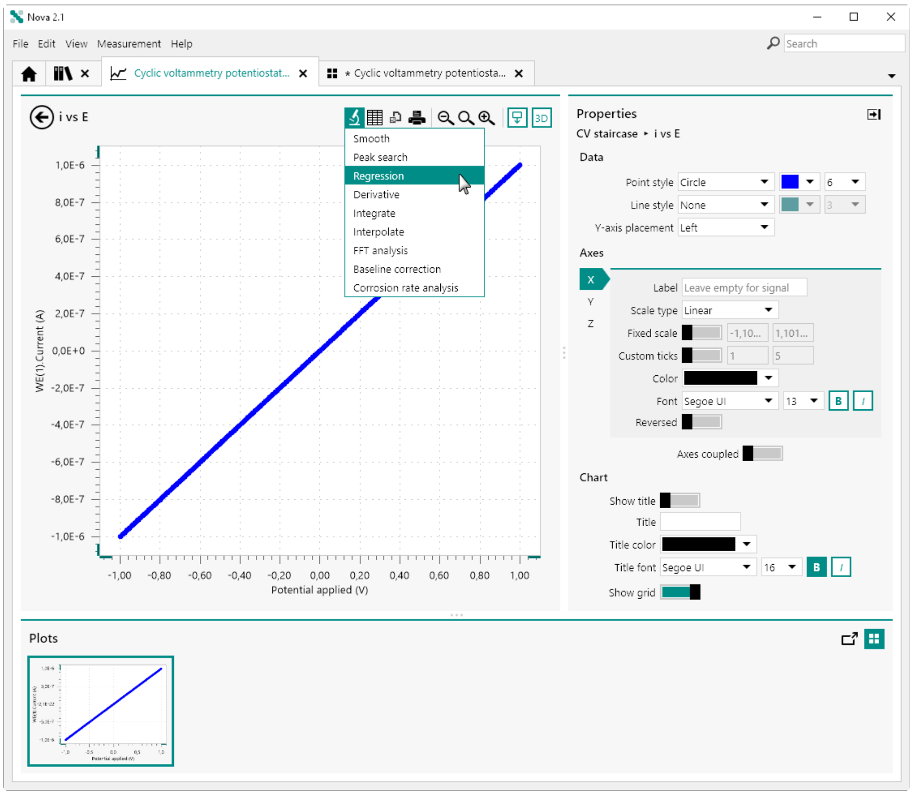


Figure 859 Adding an analysis command

The selected analysis command will be added to the procedure and will be applied on the active plot.

> **NOTE**
>
> The analysis commands displayed in the popout menu depend on the type of data shown in the active plot.

NOTE

More information on data analysis is provided in *Chapter 12*.

11.8.5 Zooming options

The controls located above the plot frame provide the means to zoom in and out on the plot and provide the means to rescale the plot for optimal display *(see Figure 860, page 707)*.

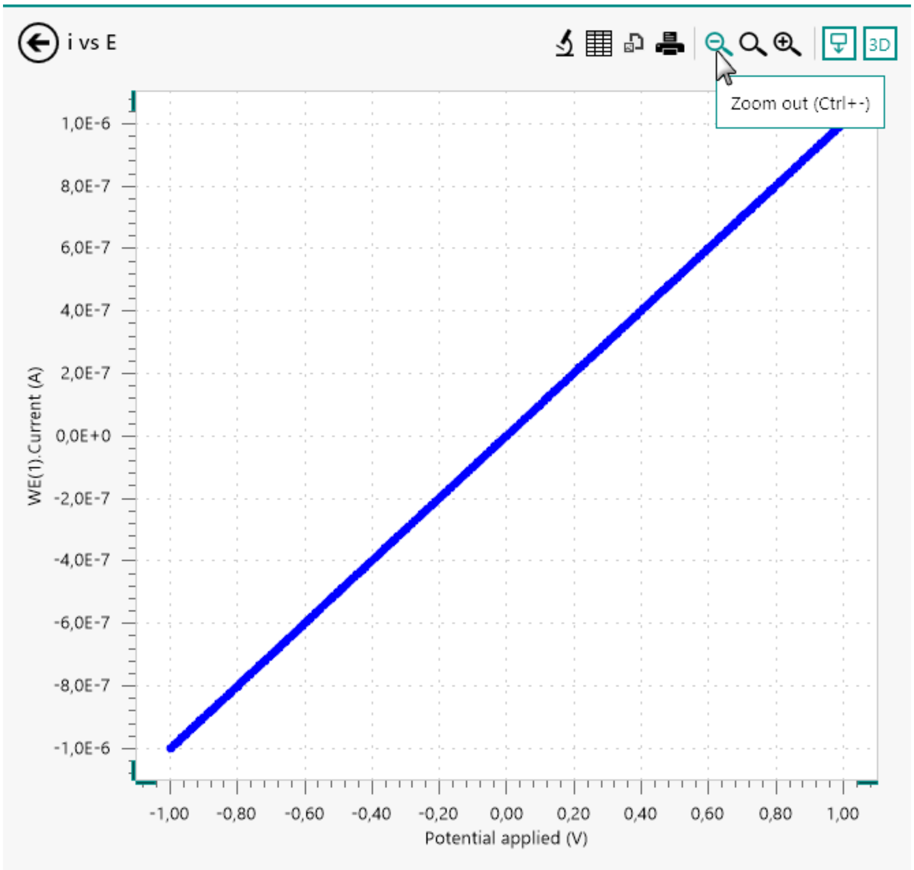


Figure 860 Zooming options are located above the plot

The following zooming options are available:

- **Zoom out:** increases the scaling of the X and Y axis on 2D plots and X, Y and Z axis on 3D plots. The button or **[CTRL]** + **[-]** keyboard shortcut can be used to do this.
- **Fit view:** adjusts the scaling of the X and Y axis on 2D plots and X, Y and Z axis on 3D plots. The button or **[F4]** keyboard shortcut can be used to do this.

- **Zoom in:** decreases the scaling of the X and Y axis on 2D plots and X, Y and Z axis on 3D plots. The button or **[CTRL]** + **[=]** keyboard shortcut can be used to do this.

NOTE

It is also possible possible to manipulate the scaling of the plot by using the **View** menu and by using the mouse directly on the plot.

11.8.6 Print plot

NOVA support the printing of plots to a printer connected to the computer. It is possible to print the visible plot, by clicking the button, located above the plot *(see Figure 861, page 708)*.

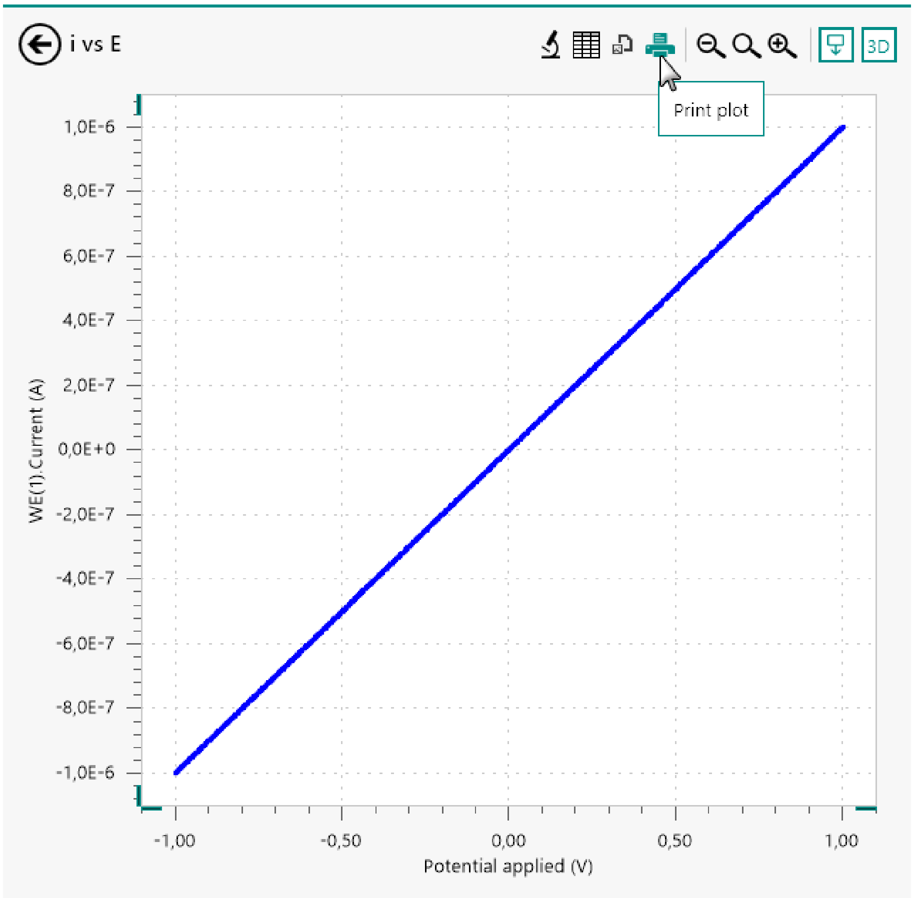


Figure 861 Printing the visible plot

A Print Settings/Preview window will be displayed *(see Figure 862, page 709)*.

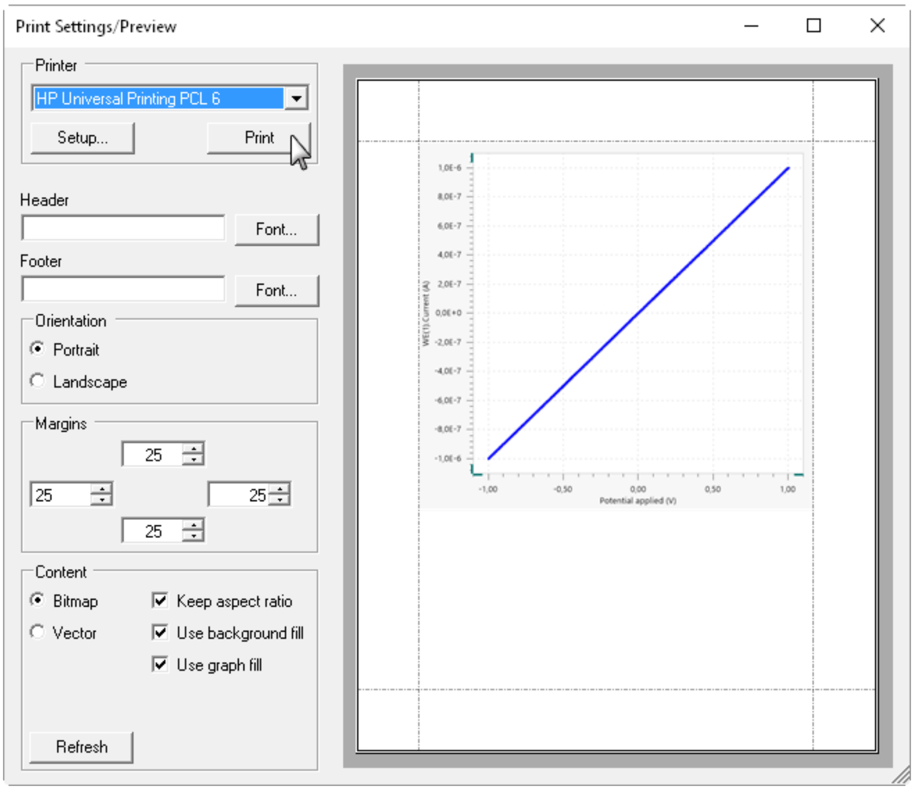


Figure 862 The Print Settings/Preview window

The following settings can be edited:

- **Printer:** specifies the printer used to print the plot. The printer can be selected using the provided drop-down list and the settings of the printer can be adjusted using the dedicated Setup... button. The Print button can be used to print the plot on the selected printer using the specified settings.
- **Header:** specifies an optional header. The font can be specified using the dedicated Font... button.
- **Footer:** specifies an optional footer. The font can be specified using the dedicated Font... button.
- **Orientation:** specifies the orientation of the plot. Radio buttons provide the choice between Portrait and Landscape.
- **Margins:** specifies the margin settings (top, bottom, left and right).

- **Content:** specifies additional options for the printing output. The following additional controls are available:
  – **Bitmap/Vector:** specifies the rendering of the plot in the preview. Radio buttons provide the choice between Bitmap (pixel) output or Vector output.
  – **Keep aspect ratio:** a checkbox that can be used to specify if the aspect ration of the plot should be maintained or not.
  – **Use background fill:** a checkbox that can be used to specify if the background of the plot should be visible or not.
  – **Use graph fill:** a checkbox that can be used to specify if the plot background should be visible or not.
  – **Refresh:** a Refresh button that can be used to refresh the preview.

NOTE

The Use graph fill checkbox has no effect in the current version of NOVA.

11.8.7 Export plot to image file

NOVA support the exporting of plots to an image file, which can be used in third party applications. Two types of image types can be used when exporting plots:

- **Pixel based output:** the data is exported to a pixel based file format, with or without compression (*.bmp, *.png, *.jpg, *.tiff, *.gif).
- **Vector based output:** the data is exported to a vector based file format (*.emf, *.svg, *.wmf).

It is possible to export the visible plot to an image file, by clicking the button, located above the plot *(see Figure 863, page 711)*.

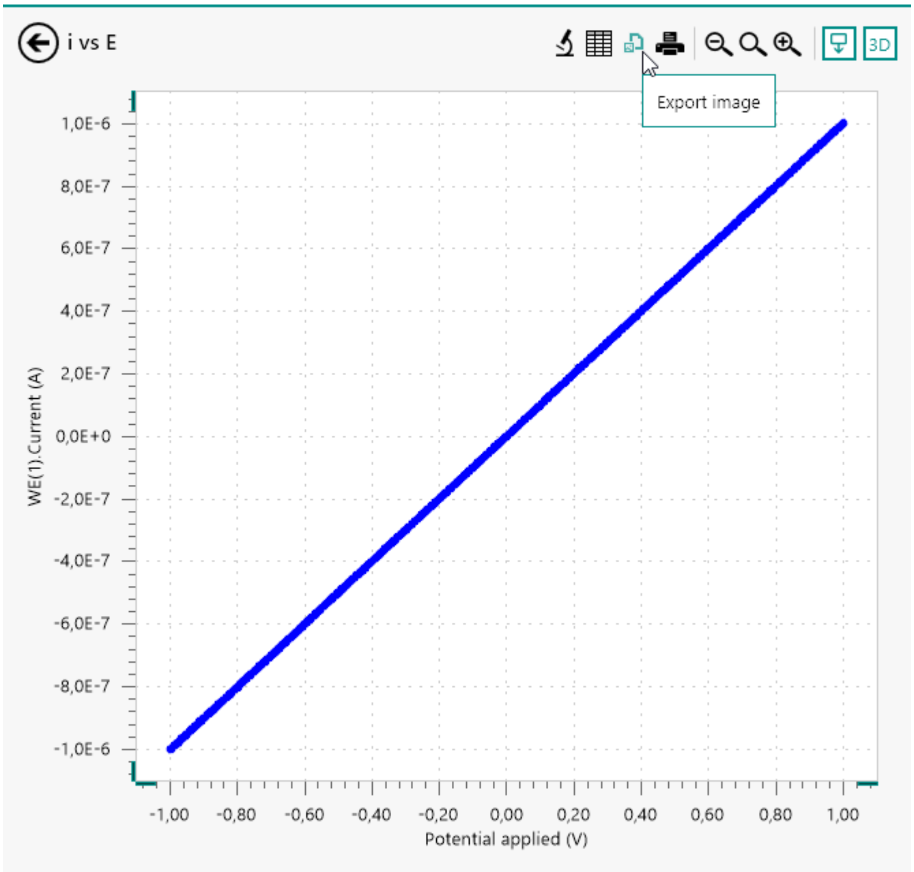


Figure 863 Exporting the plot to an image file

A popout menu will be displayed, as shown in *Figure 864*, providing the means of specifying the size of the image to export in pixels (in the case of a pixel based output file) or in arbitrary units (in the case of a vector based output file).

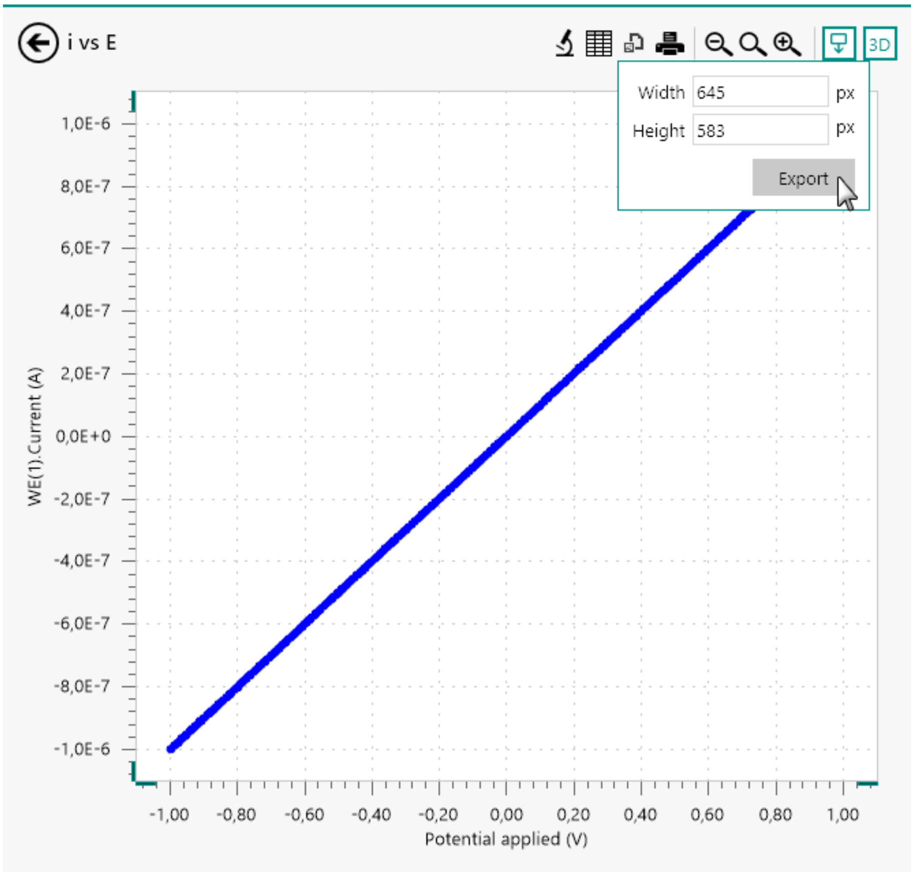


Figure 864 Specifying the size of the exported image

Clicking the Export button displays a Windows explorer dialog which can be used to specify the path, name and file type used to create the output image file *(see Figure 865, page 712)*.

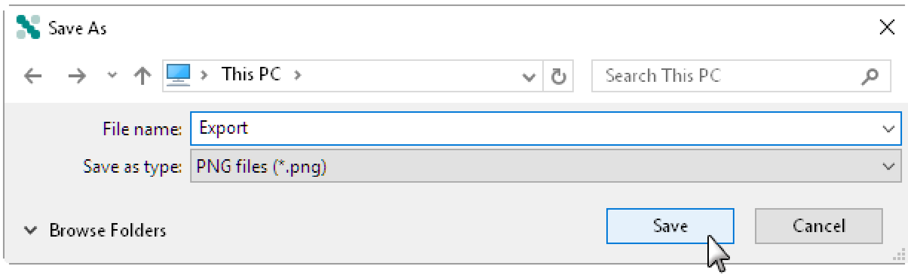


Figure 865 Specifying the name, location and type of output file

11.8.8 Relocate plots

It is possible, when the measurement is finished, to change the location of the plots using the drag and drop method directly in the **Plots** frame *(see Figure 866, page 713)*.

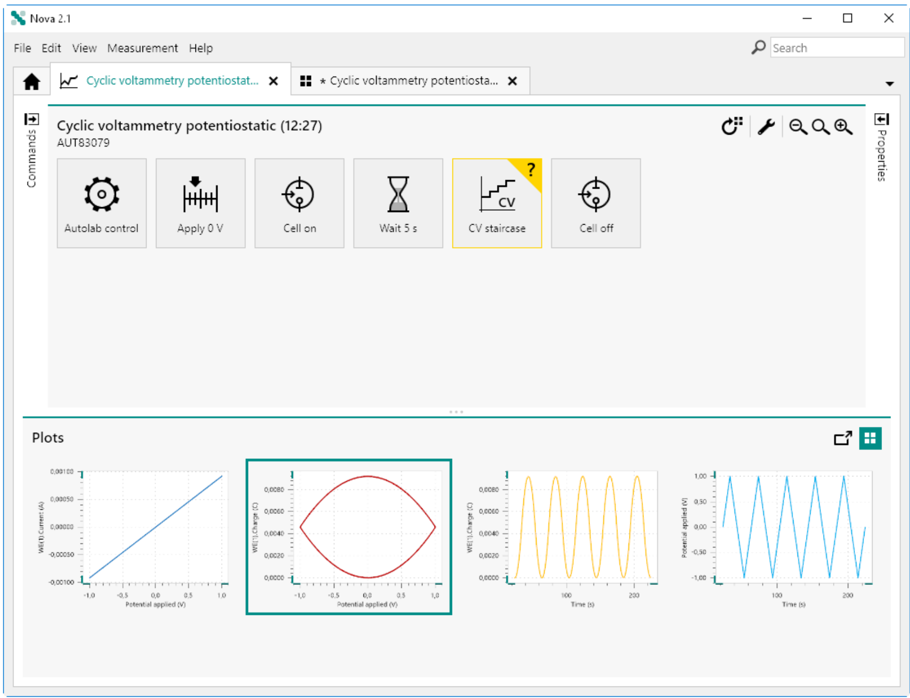


Figure 866 Plot positions can be adjusted after a measurement is finished

Click and drag a plot in the **Plots** frame to adjust its position. A grey line will be shown, indicating the new position of the dragged plot *(see Figure 867, page 713)*.

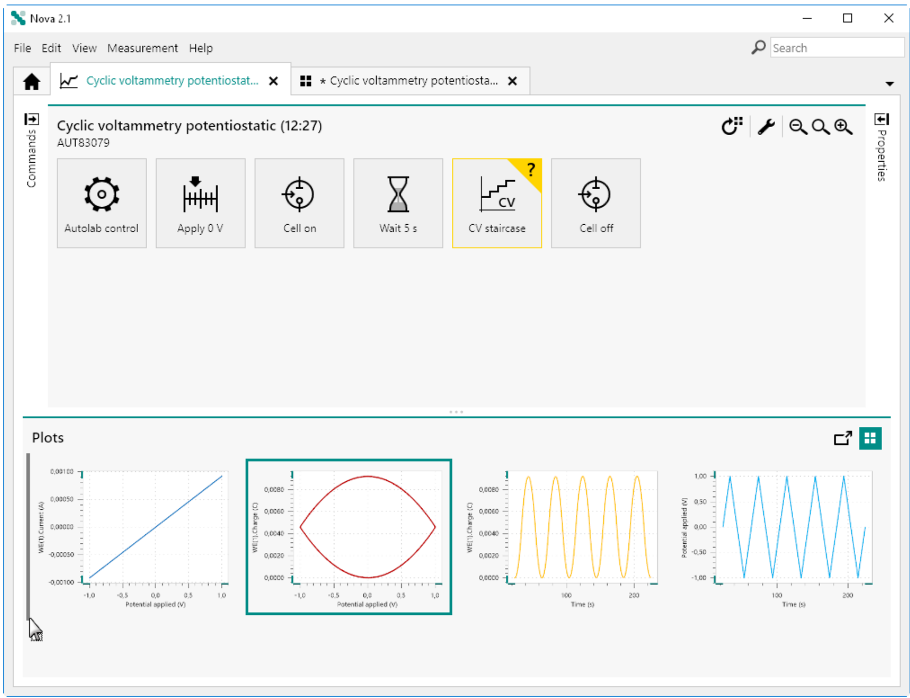


Figure 867 A grey line shows the new position of the plot

Releasing the mouse button confirms the new position of the plot in the **Plots** frame *(see Figure 868, page 714)*.

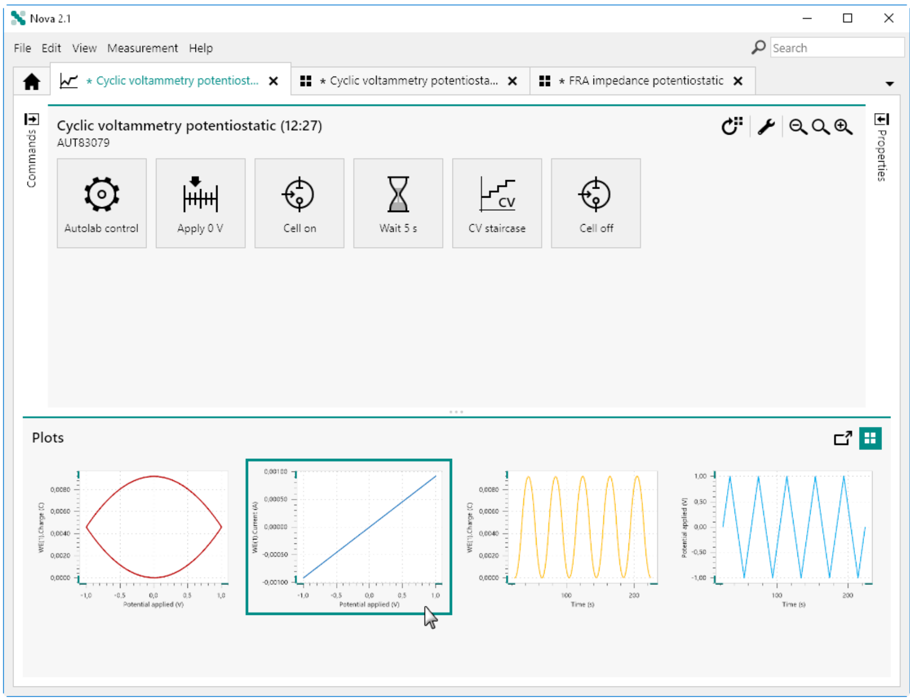


Figure 868 The plots are rearranged when the mouse button is released

If the selected plot is dragged over another plot in the frame and the mouse button is released, the selected plot will be added to the existing plot as an overlay *(see Figure 869, page 715)*.

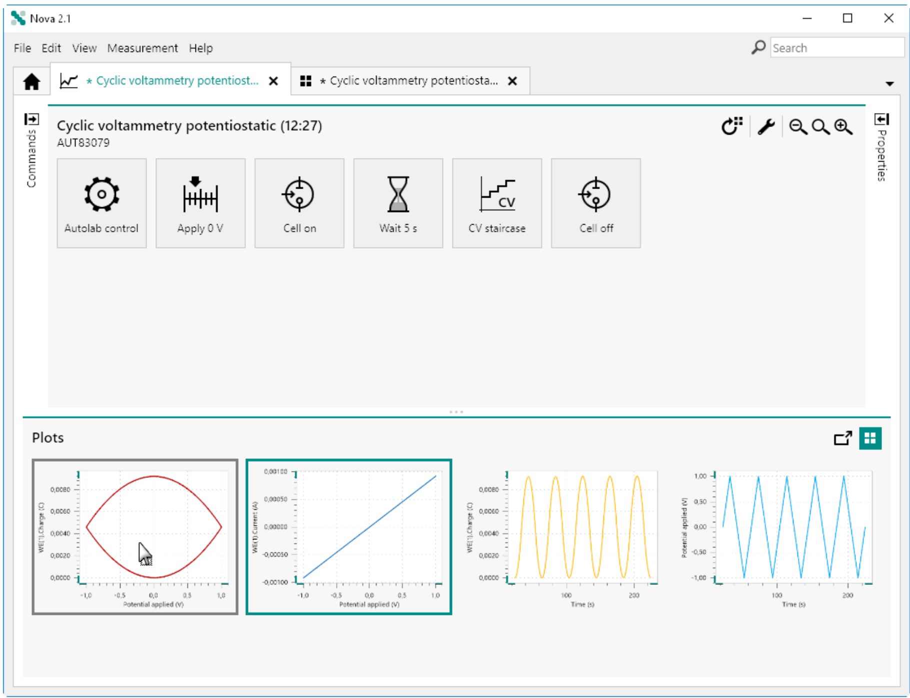


Figure 869 Dragging a plot onto another plot

The two plots will now be displayed in the same location *(see Figure 870, page 715)*.

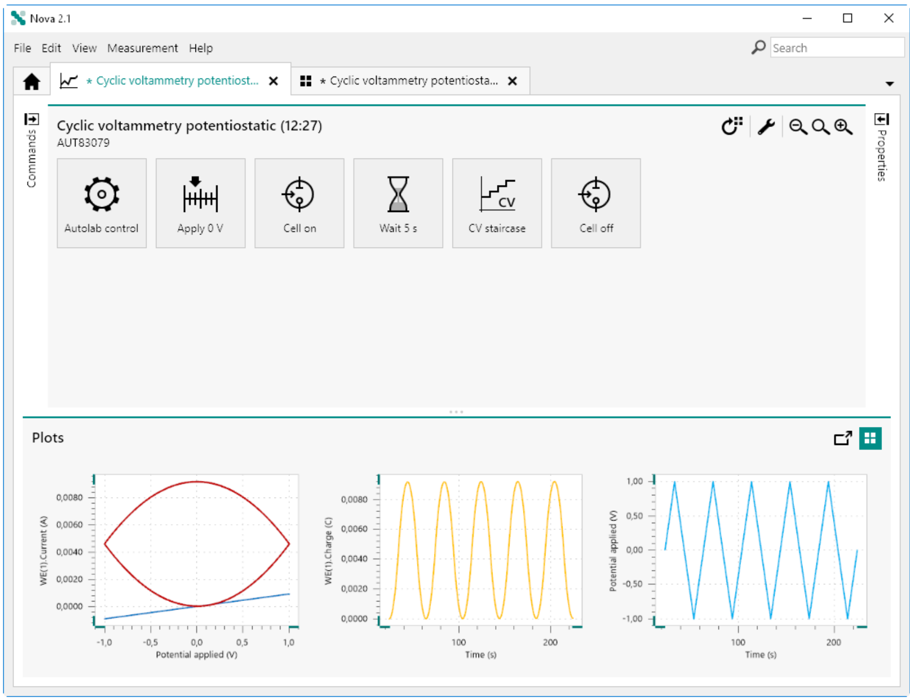


Figure 870 The plots are now assigned the same location

11.9 Viewing the data grid

When a measurement is finished, it is possible to inspect the details of all the data and events recorded by each measurement command in the procedure in the **data grid**. The data grid can be accessed by selecting a command and clicking the ▦ button, located in the top right corner of the procedure editor *(see Figure 871, page 716)*.

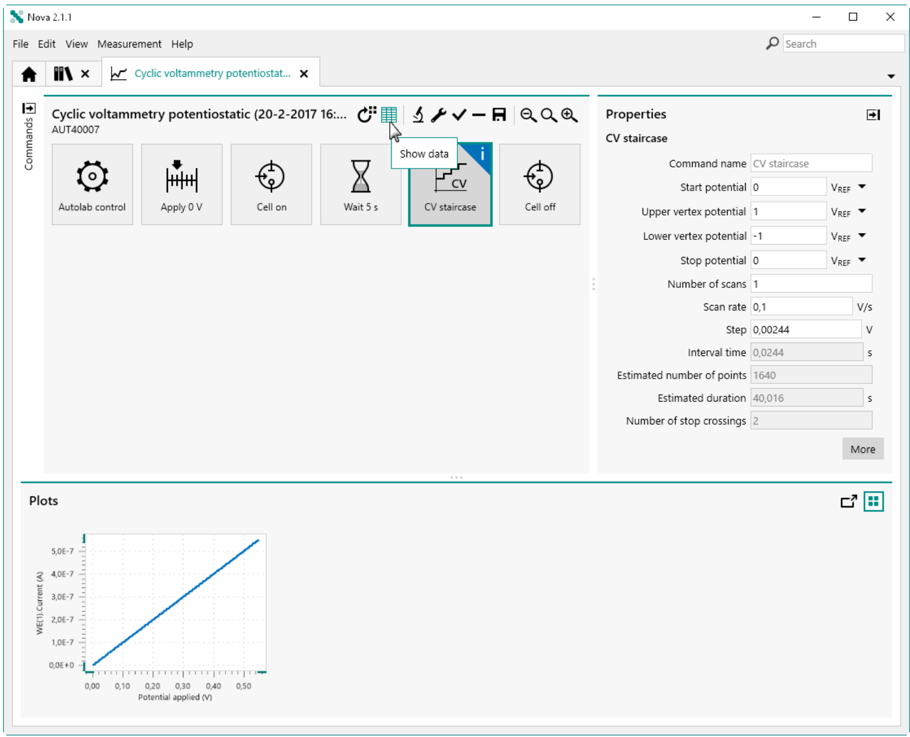


Figure 871 Opening the data grid from the procedure editor

It is also possible to display the data grid directly from the detailed view of a plot *(see Figure 872, page 717)*.

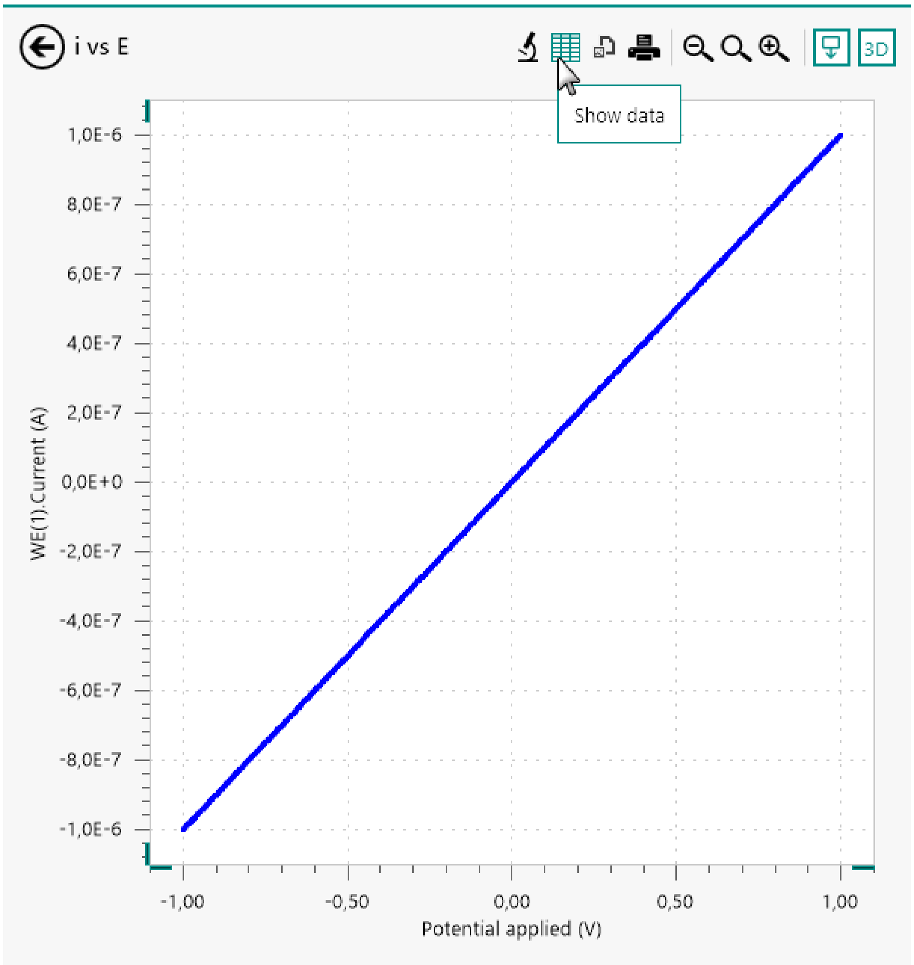
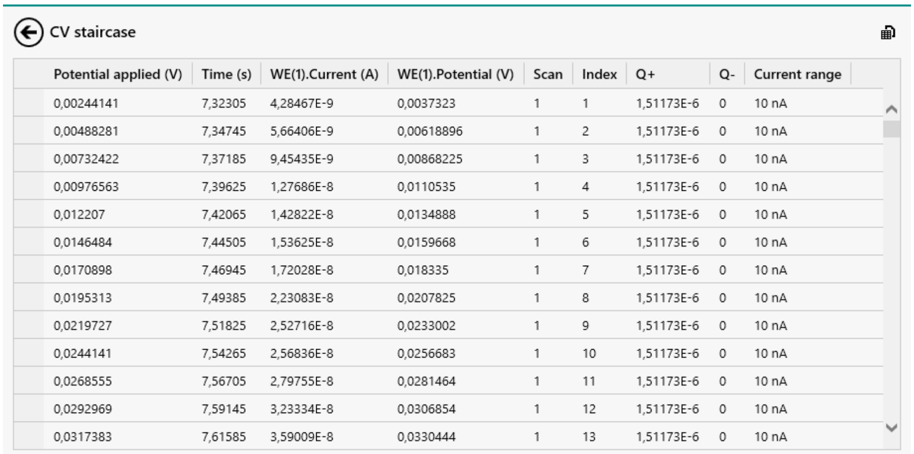


Figure 872 Opening the data grid from the detailed plot view

The data grid will be displayed. The data grid contains all the data and events recorded by the selected measurement command *(see Figure 873, page 717)*.



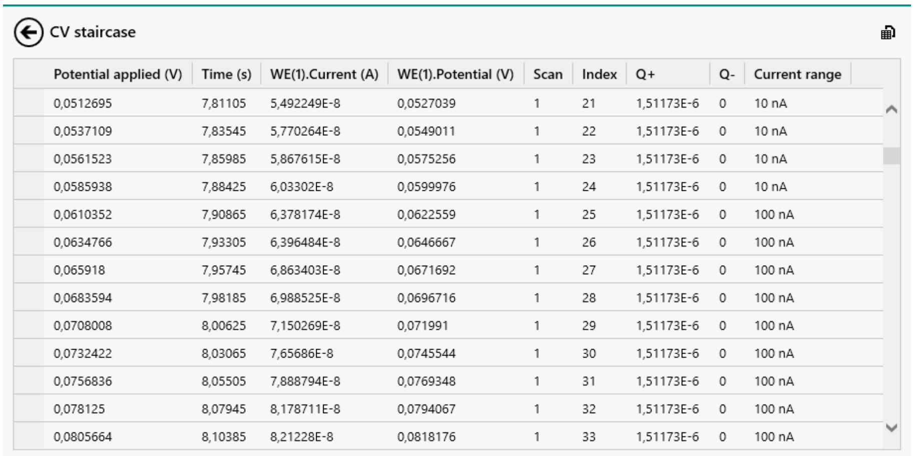
CV staircase

Potential applied (V)	Time (s)	WE(1).Current (A)	WE(1).Potential (V)	Scan	Index	Q+	Q-	Current range
0,00244141	7,32305	4,28467E-9	0,0037323	1	1	1,51173E-6	0	10 nA
0,00488281	7,34745	5,66406E-9	0,00618896	1	2	1,51173E-6	0	10 nA
0,00732422	7,37185	9,45435E-9	0,00868225	1	3	1,51173E-6	0	10 nA
0,00976563	7,39625	1,27686E-8	0,0110535	1	4	1,51173E-6	0	10 nA
0,012207	7,42065	1,42822E-8	0,0134888	1	5	1,51173E-6	0	10 nA
0,0146484	7,44505	1,53625E-8	0,0159668	1	6	1,51173E-6	0	10 nA
0,0170898	7,46945	1,72028E-8	0,018335	1	7	1,51173E-6	0	10 nA
0,0195313	7,49385	2,23083E-8	0,0207825	1	8	1,51173E-6	0	10 nA
0,0219727	7,51825	2,52716E-8	0,0233002	1	9	1,51173E-6	0	10 nA
0,0244141	7,54265	2,56836E-8	0,0256683	1	10	1,51173E-6	0	10 nA
0,0268555	7,56705	2,79755E-8	0,0281464	1	11	1,51173E-6	0	10 nA
0,0292969	7,59145	3,23334E-8	0,0306854	1	12	1,51173E-6	0	10 nA
0,0317383	7,61585	3,59009E-8	0,0330444	1	13	1,51173E-6	0	10 nA

Figure 873 The data grid shows all the measured data for the measurement command

11.9.1 Current range logged in the data grid

The current range used for recording the WE(1).Current in any electrochemical measurement is always reported in the data grid *(see Figure 874, page 718)*.



CV staircase

Potential applied (V)	Time (s)	WE(1).Current (A)	WE(1).Potential (V)	Scan	Index	Q+	Q-	Current range
0,0512695	7,81105	5,492249E-8	0,0527039	1	21	1,51173E-6	0	10 nA
0,0537109	7,83545	5,770264E-8	0,0549011	1	22	1,51173E-6	0	10 nA
0,0561523	7,85985	5,867615E-8	0,0575256	1	23	1,51173E-6	0	10 nA
0,0585938	7,88425	6,03302E-8	0,0599976	1	24	1,51173E-6	0	10 nA
0,0610352	7,90865	6,378174E-8	0,0622559	1	25	1,51173E-6	0	100 nA
0,0634766	7,93305	6,396484E-8	0,0646667	1	26	1,51173E-6	0	100 nA
0,065918	7,95745	6,863403E-8	0,0671692	1	27	1,51173E-6	0	100 nA
0,0683594	7,98185	6,988525E-8	0,0696716	1	28	1,51173E-6	0	100 nA
0,0708008	8,00625	7,150269E-8	0,071991	1	29	1,51173E-6	0	100 nA
0,0732422	8,03065	7,65686E-8	0,0745544	1	30	1,51173E-6	0	100 nA
0,0756836	8,05505	7,888794E-8	0,0769348	1	31	1,51173E-6	0	100 nA
0,078125	8,07945	8,178711E-8	0,0794067	1	32	1,51173E-6	0	100 nA
0,0805664	8,10385	8,21228E-8	0,0818176	1	33	1,51173E-6	0	100 nA

Figure 874 The current range by the instrument is logged in the data grid

If the current range was modified by the procedure during a measurement, this will be visible in the Current range column of the data grid.

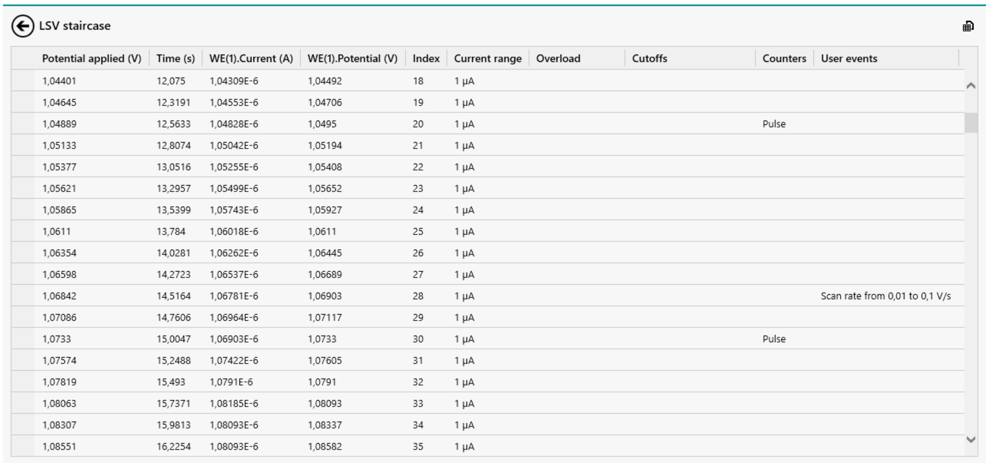
> **NOTE**
>
> Only the current range of the Autolab PGSTAT instrument is logged during a measurement.

11.9.2 Events logged in the data grid

Events taking place during a measurement are logged in the data grid. The following columns may become visible in the data grid if an applicable event was detected during a measurement:

- **Overloads:** these events correspond to situations where a current, voltage or temperature overload was detected during a measurement.
- **Cutoffs:** these events correspond to situations where a cutoff condition is met.
- **Counters:** these events correspond to situations where a counter is activated.
- **User events:** these events correspond to situations where the user changed a measurement property during a measurement or used a flow control option (stop, pause, reverse scan direction) provided by NOVA.

Figure 875 shows an example of events logged in the data grid.



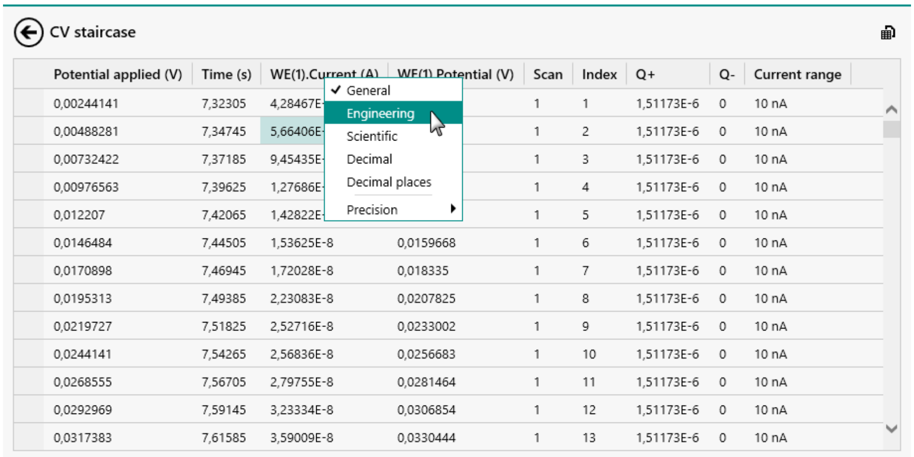
LSV staircase

Potential applied (V)	Time (s)	WE(1).Current (A)	WE(1).Potential (V)	Index	Current range	Overload	Cutoffs	Counters	User events
1,04401	12,075	1,04309E-6	1,04492	18	1 µA				
1,04645	12,3191	1,04553E-6	1,04706	19	1 µA				
1,04889	12,5633	1,04828E-6	1,0495	20	1 µA			Pulse	
1,05133	12,8074	1,05042E-6	1,05194	21	1 µA				
1,05377	13,0516	1,05255E-6	1,05408	22	1 µA				
1,05621	13,2957	1,05499E-6	1,05652	23	1 µA				
1,05865	13,5399	1,05743E-6	1,05927	24	1 µA				
1,0611	13,784	1,06018E-6	1,0611	25	1 µA				
1,06354	14,0281	1,06262E-6	1,06445	26	1 µA				
1,06598	14,2723	1,06537E-6	1,06689	27	1 µA				
1,06842	14,5164	1,06781E-6	1,06903	28	1 µA				Scan rate from 0,01 to 0,1 V/s
1,07086	14,7606	1,06964E-6	1,07117	29	1 µA				
1,0733	15,0047	1,06903E-6	1,0733	30	1 µA			Pulse	
1,07574	15,2488	1,07422E-6	1,07605	31	1 µA				
1,07819	15,493	1,0791E-6	1,0791	32	1 µA				
1,08063	15,7371	1,08185E-6	1,08093	33	1 µA				
1,08307	15,9813	1,08093E-6	1,08337	34	1 µA				
1,08551	16,2254	1,08093E-6	1,08582	35	1 µA				

Figure 875 Events are logged in the data grid

11.9.3 Formatting the data grid

The formatting of the columns can be modified by right-clicking one of the column headers and selecting the required number formatting from the context menu *(see Figure 876, page 719)*.



CV staircase

Potential applied (V)	Time (s)	WE(1).Current (A)	WE(1).Potential (V)	Scan	Index	Q+	Q-	Current range
0,00244141	7,32305	4,28467E		1	1	1,51173E-6	0	10 nA
0,00488281	7,34745	5,66406E		1	2	1,51173E-6	0	10 nA
0,00732422	7,37185	9,45435E		1	3	1,51173E-6	0	10 nA
0,00976563	7,39625	1,27686E		1	4	1,51173E-6	0	10 nA
0,012207	7,42065	1,42822E		1	5	1,51173E-6	0	10 nA
0,0146484	7,44505	1,53625E-8	0,0159668	1	6	1,51173E-6	0	10 nA
0,0170898	7,46945	1,72028E-8	0,018335	1	7	1,51173E-6	0	10 nA
0,0195313	7,49385	2,23083E-8	0,0207825	1	8	1,51173E-6	0	10 nA
0,0219727	7,51825	2,52716E-8	0,0233002	1	9	1,51173E-6	0	10 nA
0,0244141	7,54265	2,56836E-8	0,0256683	1	10	1,51173E-6	0	10 nA
0,0268555	7,56705	2,79755E-8	0,0281464	1	11	1,51173E-6	0	10 nA
0,0292969	7,59145	3,23334E-8	0,0306854	1	12	1,51173E-6	0	10 nA
0,0317383	7,61585	3,59009E-8	0,0330444	1	13	1,51173E-6	0	10 nA

Figure 876 The formatting used in the data grid can be specified

The number of significant digits or decimals can also be specified for each signal, by extending the context menu and specifying the required precision *(see Figure 877, page 720)*.

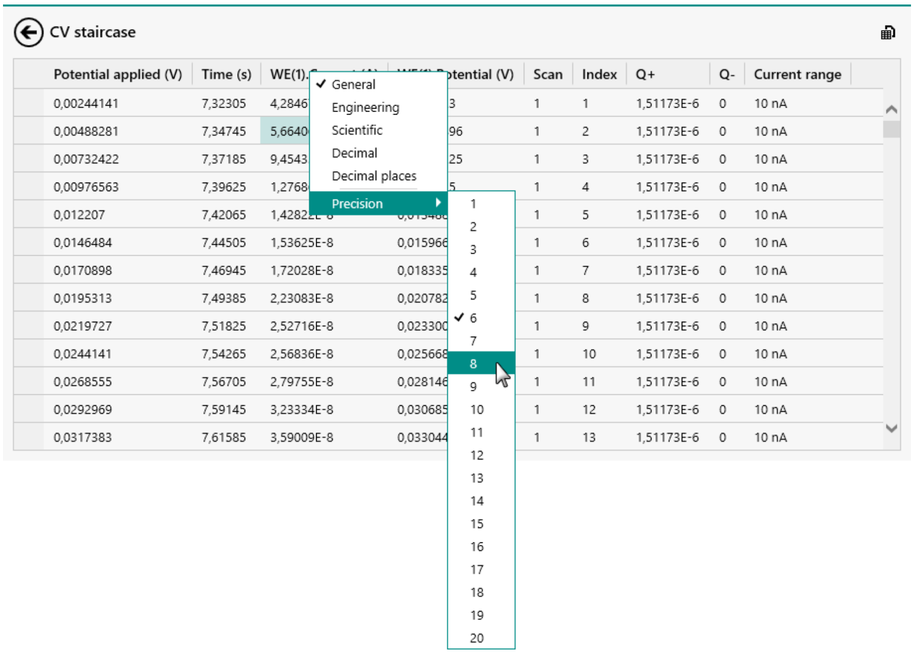


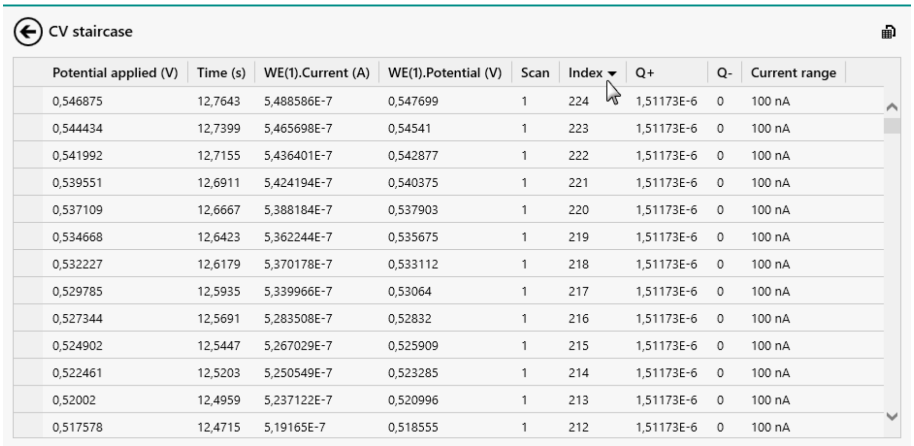
Figure 877 The number of significant digits or decimals can be specified

NOTE

The formatting of the columns in the data grid is saved when the file is saved.

11.9.4 Sorting the data grid

It is possible to sort the contents of the data grid by clicking one of the column header. This will sort the content of the column ascending or descending and the other columns of the data grid will be sorted based on the new order of the sorted column. Clicking the column header cycles from ascending sorting to descending sorting *(see Figure 878, page 721)*.


CV staircase

Potential applied (V)	Time (s)	WE(1).Current (A)	WE(1).Potential (V)	Scan	Index ▼	Q+	Q-	Current range
0,546875	12,7643	5,488586E-7	0,547699	1	224	1,51173E-6	0	100 nA
0,544434	12,7399	5,465698E-7	0,54541	1	223	1,51173E-6	0	100 nA
0,541992	12,7155	5,436401E-7	0,542877	1	222	1,51173E-6	0	100 nA
0,539551	12,6911	5,424194E-7	0,540375	1	221	1,51173E-6	0	100 nA
0,537109	12,6667	5,388184E-7	0,537903	1	220	1,51173E-6	0	100 nA
0,534668	12,6423	5,362244E-7	0,535675	1	219	1,51173E-6	0	100 nA
0,532227	12,6179	5,370178E-7	0,533112	1	218	1,51173E-6	0	100 nA
0,529785	12,5935	5,339966E-7	0,53064	1	217	1,51173E-6	0	100 nA
0,527344	12,5691	5,283508E-7	0,52832	1	216	1,51173E-6	0	100 nA
0,524902	12,5447	5,267029E-7	0,525909	1	215	1,51173E-6	0	100 nA
0,522461	12,5203	5,250549E-7	0,523285	1	214	1,51173E-6	0	100 nA
0,52002	12,4959	5,237122E-7	0,520996	1	213	1,51173E-6	0	100 nA
0,517578	12,4715	5,19165E-7	0,518555	1	212	1,51173E-6	0	100 nA

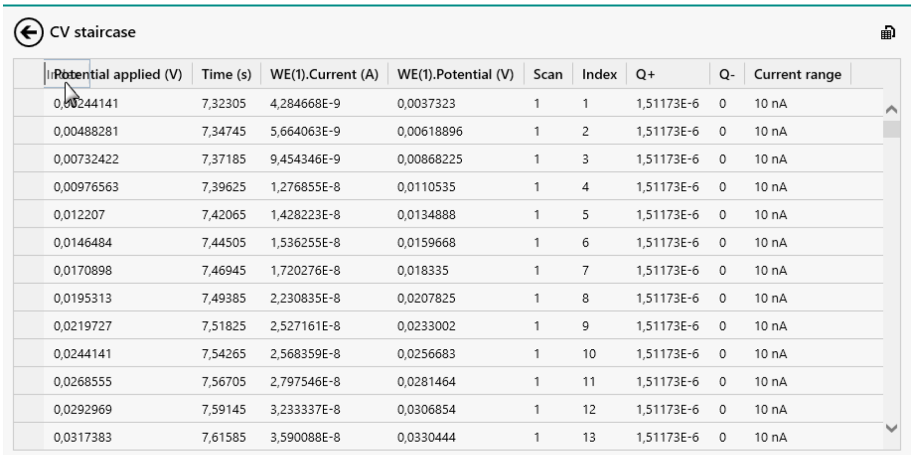
Figure 878 Sorting the contents of the data grid

NOTE

A column sorted in ascending mode is indicated by the ▼ symbol. A column sorted in descending mode is indicated by the ▲ symbol.

11.9.5 Changing the order of the columns in the data grid

It is possible to change the order of the columns. To move a column in the data grid, click one of the column headers and drag the mouse left or right in the grid, while holding the mouse button *(see Figure 879, page 721)*.


CV staircase

Potential applied (V)	Time (s)	WE(1).Current (A)	WE(1).Potential (V)	Scan	Index	Q+	Q-	Current range
0,[illegible]244141	7,32305	4,284668E-9	0,0037323	1	1	1,51173E-6	0	10 nA
0,00488281	7,34745	5,664063E-9	0,00618896	1	2	1,51173E-6	0	10 nA
0,00732422	7,37185	9,454346E-9	0,00868225	1	3	1,51173E-6	0	10 nA
0,00976563	7,39625	1,276855E-8	0,0110535	1	4	1,51173E-6	0	10 nA
0,012207	7,42065	1,428223E-8	0,0134888	1	5	1,51173E-6	0	10 nA
0,0146484	7,44505	1,536255E-8	0,0159668	1	6	1,51173E-6	0	10 nA
0,0170898	7,46945	1,720276E-8	0,018335	1	7	1,51173E-6	0	10 nA
0,0195313	7,49385	2,230835E-8	0,0207825	1	8	1,51173E-6	0	10 nA
0,0219727	7,51825	2,527161E-8	0,0233002	1	9	1,51173E-6	0	10 nA
0,0244141	7,54265	2,568359E-8	0,0256683	1	10	1,51173E-6	0	10 nA
0,0268555	7,56705	2,797546E-8	0,0281464	1	11	1,51173E-6	0	10 nA
0,0292969	7,59145	3,233337E-8	0,0306854	1	12	1,51173E-6	0	10 nA
0,0317383	7,61585	3,590088E-8	0,0330444	1	13	1,51173E-6	0	10 nA

Figure 879 The order of the columns can be modified

Release the mouse button validate the new location of the column *(see Figure 880, page 722)*.

CV staircase

Index	Potential applied (V)	Time (s)	WE(1).Current (A)	WE(1).Potential (V)	Scan	Q+	Q-	Current range
1	0,00244141	7,32305	4,284668E-9	0,0037323	1	1,51173E-6	0	10 nA
2	0,00488281	7,34745	5,664063E-9	0,00618896	1	1,51173E-6	0	10 nA
3	0,00732422	7,37185	9,454346E-9	0,00868225	1	1,51173E-6	0	10 nA
4	0,00976563	7,39625	1,276855E-8	0,0110535	1	1,51173E-6	0	10 nA
5	0,012207	7,42065	1,428223E-8	0,0134888	1	1,51173E-6	0	10 nA
6	0,0146484	7,44505	1,536255E-8	0,0159668	1	1,51173E-6	0	10 nA
7	0,0170898	7,46945	1,720276E-8	0,018335	1	1,51173E-6	0	10 nA
8	0,0195313	7,49385	2,230835E-8	0,0207825	1	1,51173E-6	0	10 nA
9	0,0219727	7,51825	2,527161E-8	0,0233002	1	1,51173E-6	0	10 nA
10	0,0244141	7,54265	2,568359E-8	0,0256683	1	1,51173E-6	0	10 nA
11	0,0268555	7,56705	2,797546E-8	0,0281464	1	1,51173E-6	0	10 nA
12	0,0292969	7,59145	3,233337E-8	0,0306854	1	1,51173E-6	0	10 nA
13	0,0317383	7,61585	3,590088E-8	0,0330444	1	1,51173E-6	0	10 nA

Figure 880 The new order of columns in the data grid

NOTE

The order of the columns in the data grid is saved when the file is saved.

11.9.6 Exporting the data from the data grid

Finally, the data grid can also be used to export the data in the grid to an ASCII file or an Excel file. To export the data, click the button in the top right corner *(see Figure 881, page 722)*.

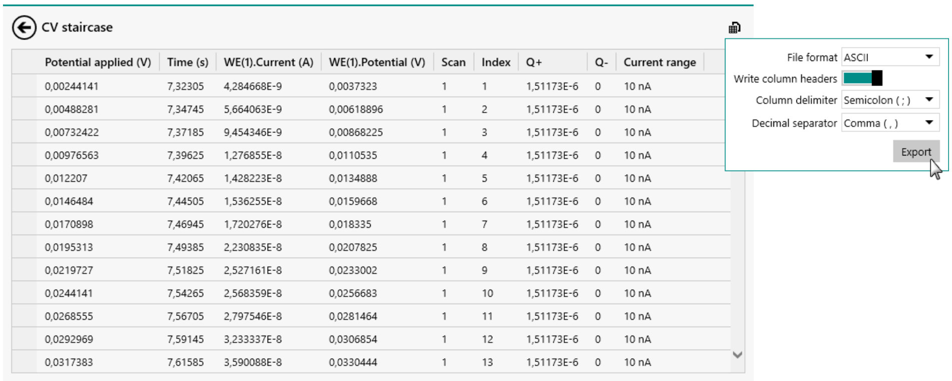
CV staircase

Export data

Potential applied (V)	Time (s)	WE(1).Current (A)	WE(1).Potential (V)	Scan	Index	Q+	Q-	Current range
0,00244141	7,32305	4,284668E-9	0,0037323	1	1	1,51173E-6	0	10 nA
0,00488281	7,34745	5,664063E-9	0,00618896	1	2	1,51173E-6	0	10 nA
0,00732422	7,37185	9,454346E-9	0,00868225	1	3	1,51173E-6	0	10 nA
0,00976563	7,39625	1,276855E-8	0,0110535	1	4	1,51173E-6	0	10 nA
0,012207	7,42065	1,428223E-8	0,0134888	1	5	1,51173E-6	0	10 nA
0,0146484	7,44505	1,536255E-8	0,0159668	1	6	1,51173E-6	0	10 nA
0,0170898	7,46945	1,720276E-8	0,018335	1	7	1,51173E-6	0	10 nA
0,0195313	7,49385	2,230835E-8	0,0207825	1	8	1,51173E-6	0	10 nA
0,0219727	7,51825	2,527161E-8	0,0233002	1	9	1,51173E-6	0	10 nA
0,0244141	7,54265	2,568359E-8	0,0256683	1	10	1,51173E-6	0	10 nA
0,0268555	7,56705	2,797546E-8	0,0281464	1	11	1,51173E-6	0	10 nA
0,0292969	7,59145	3,233337E-8	0,0306854	1	12	1,51173E-6	0	10 nA
0,0317383	7,61585	3,590088E-8	0,0330444	1	13	1,51173E-6	0	10 nA

Figure 881 The data points can be exported using the provided button

A menu will pop-out, as shown in *Figure 882*.



Potential applied (V)	Time (s)	WE(1).Current (A)	WE(1).Potential (V)	Scan	Index	Q+	Q-	Current range
0,00244141	7,32305	4,284668E-9	0,0037323	1	1	1,51173E-6	0	10 nA
0,00488281	7,34745	5,664063E-9	0,00618896	1	2	1,51173E-6	0	10 nA
0,00732422	7,37185	9,454346E-9	0,00868225	1	3	1,51173E-6	0	10 nA
0,00976563	7,39625	1,276855E-8	0,0110535	1	4	1,51173E-6	0	10 nA
0,012207	7,42065	1,428223E-8	0,0134888	1	5	1,51173E-6	0	10 nA
0,0146484	7,44505	1,536255E-8	0,0159668	1	6	1,51173E-6	0	10 nA
0,0170898	7,46945	1,720276E-8	0,018335	1	7	1,51173E-6	0	10 nA
0,0195313	7,49385	2,230835E-8	0,0207825	1	8	1,51173E-6	0	10 nA
0,0219727	7,51825	2,527161E-8	0,0233002	1	9	1,51173E-6	0	10 nA
0,0244141	7,54265	2,568359E-8	0,0256683	1	10	1,51173E-6	0	10 nA
0,0268555	7,56705	2,797546E-8	0,0281464	1	11	1,51173E-6	0	10 nA
0,0292969	7,59145	3,233337E-8	0,0306854	1	12	1,51173E-6	0	10 nA
0,0317383	7,61585	3,590088E-8	0,0330444	1	13	1,51173E-6	0	10 nA

Figure 882 The export settings are specified in the a dedicated pop-out menu

The menu can be used to specify the file format, using the provided drop-down list *(see Figure 883, page 723)*.

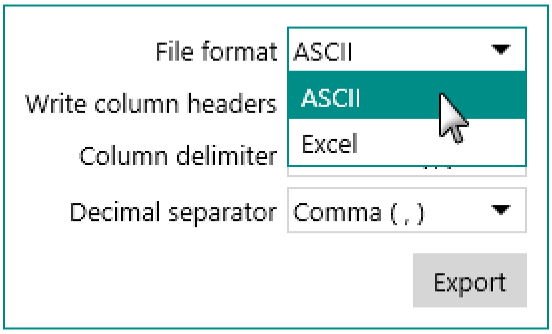


Figure 883 The data can be exported as ASCII or Excel

When the data is exported as ASCII (Comma Separated Values), additional settings can be specified *(see Figure 884, page 723)*. These settings depend on the required output format of the data.

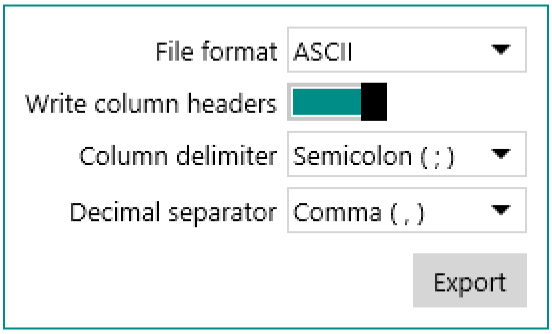


Figure 884 The settings used to export data to a ASCII file

When the data is exported as Excel, the file is automatically formatted *(see Figure 885, page 724)*.

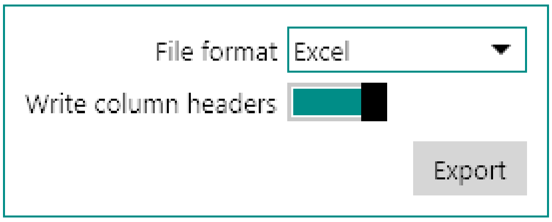


Figure 885 The settings used to export data to an Excel file

The following settings can be specified:

- **File format:** specifies the format of the output file (ASCII or Excel), using the provided drop-down list.
- **Write column headers:** a toggle that can be used to indicate if the names of the signals need to be added to the output file.
- **Column delimiter:** specifies the symbol used as a column separator, using the provided drop-down list. This property is only available for ASCII output.
- **Decimal separator:** specifies the decimal separator symbol used in the output file, using the provided drop-down list. This property is only available for ASCII output.

Clicking the Export button displays a save dialog window which can be used to specify the filename and location *(see Figure 886, page 724)*.

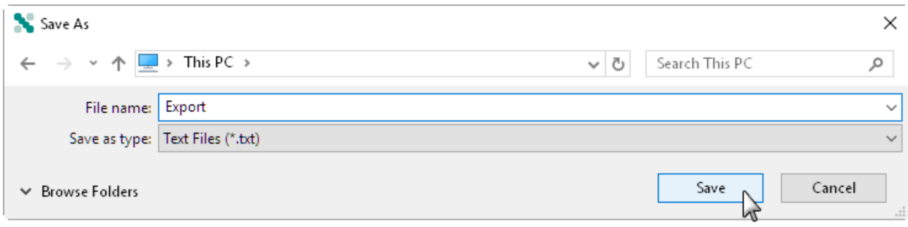


Figure 886 Specifying the filename and location

11.10 Convert data to procedure

At the end of measurement, the measured data can be converted to a new procedure. In order to do this, click the button in the top right corner of the procedure editor *(see Figure 887, page 725)*.

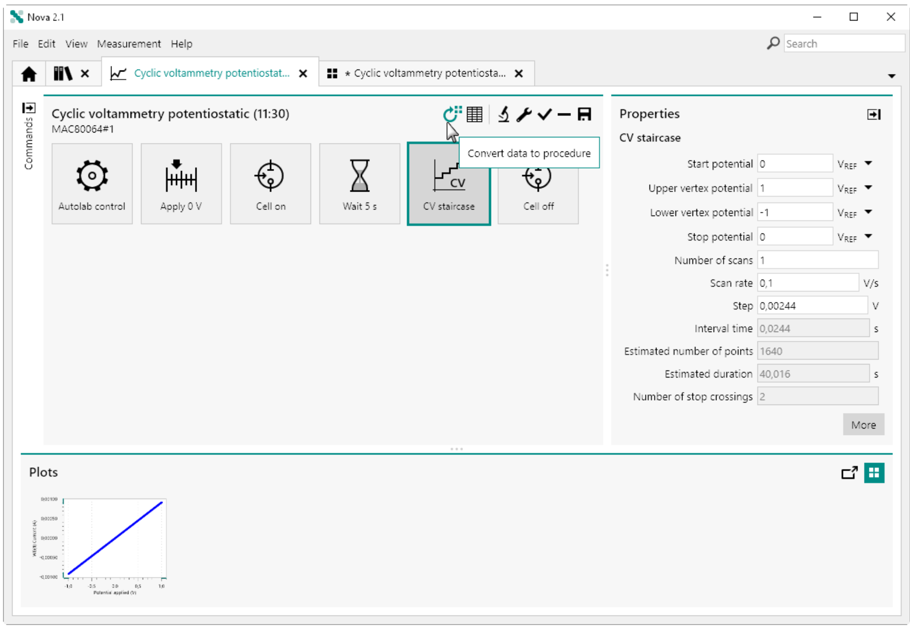


Figure 887 Converting data to procedure

If the procedure was modified during the measurement or if data analysis tools were added to the data, a message will be displayed when the button is clicked, providing the means to define how the data should be converted to a new procedure *(see Figure 888, page 725)*.

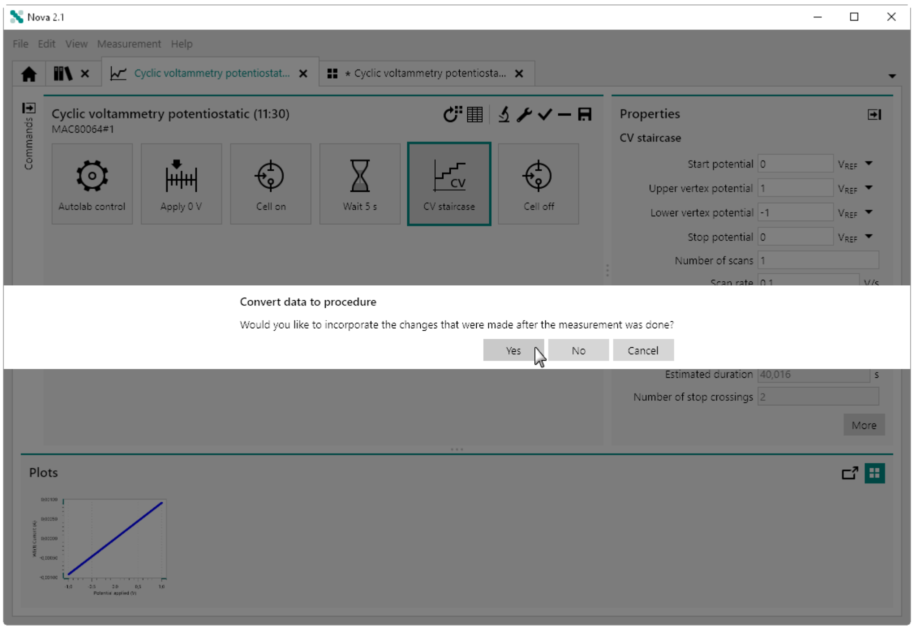


Figure 888 The changes can be kept or discarded

- Clicking the **Yes** button will convert the modified data to a new procedure. All modifications and changes that were carried out during and after the measurement will be added to the source procedure used to generate this data.

- Clicking the No button will convert the data to a new procedure but will discard all the changes that were carried out during and after the measurement.
- Clicking the Cancel button will cancel this action.

12 Data analysis

When data has been measured, it is possible to use the data analysis commands provided in NOVA to analyze the data. To analyze acquired data in NOVA, it is necessary to add the required command to the measured procedure and apply the function of these commands on the measure data.

NOTE

Data analysis commands can be added to the initial procedure or the procedure after the measurement is finished.

To add a data analysis command to a measured procedure, two methods can be used:

- Drag and drop the analysis command in the procedure
- Use the contextual shortcut button, located in the top right corner of the procedure editor

The functionality of the data analysis commands commands is explained in the previous chapters and will not be detailed again in this chapter. This chapter focuses on the use of these commands on **measured** data. Only the commands that provide controls that are used in a specific way on analyzed data are detailed in this chapter.

The following commands are detailed:

- Smooth
- Peak search
- Regression
- Integrate
- Interpolate
- Baseline correction
- Corrosion rate
- Hydrodynamic analysis
- Electrochemical circle fit
- Fit and simulation

12.1 Smooth analysis

The **Smooth** command provides additional controls that can be used when the command is used to analyze data. To use the **Smooth** command, this command can be added to the procedure as a command, using the drag and drop method, or by using the button. In the latter case, a pop-out menu is displayed, providing a list of commands and possible plots on which these command can be applied *(see Figure 889, page 728)*.

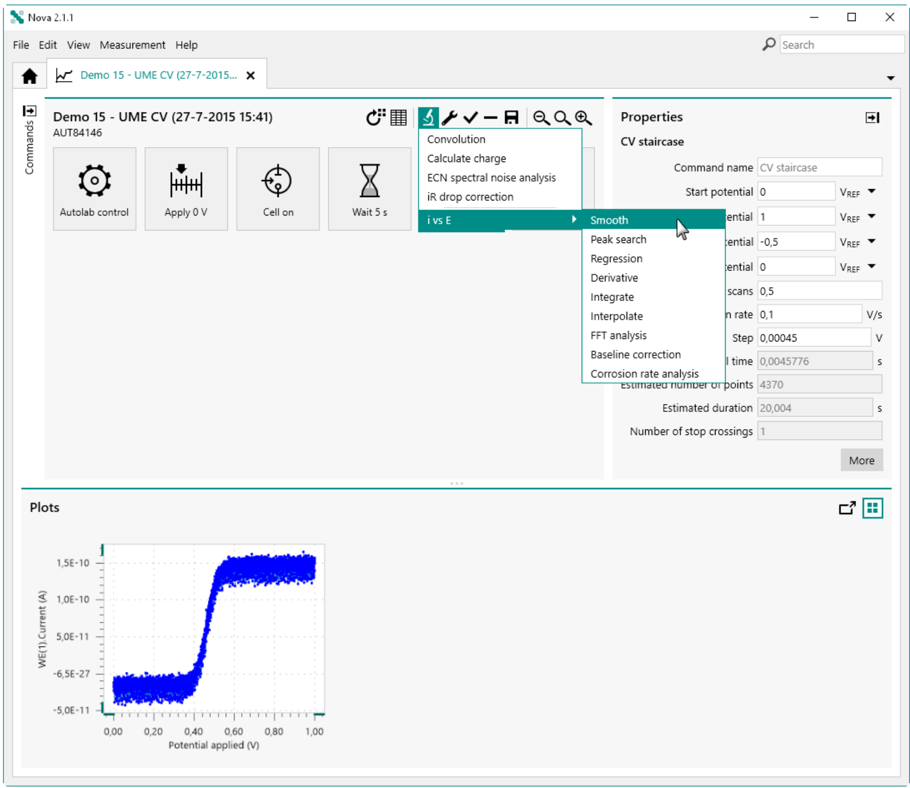


Figure 889 Adding a Smooth command to the i vs E plot

The **Smooth** command is added to the procedure editor. Clicking the command shows the properties in the dedicated panel on the right hand side *(see Figure 890, page 729)*.

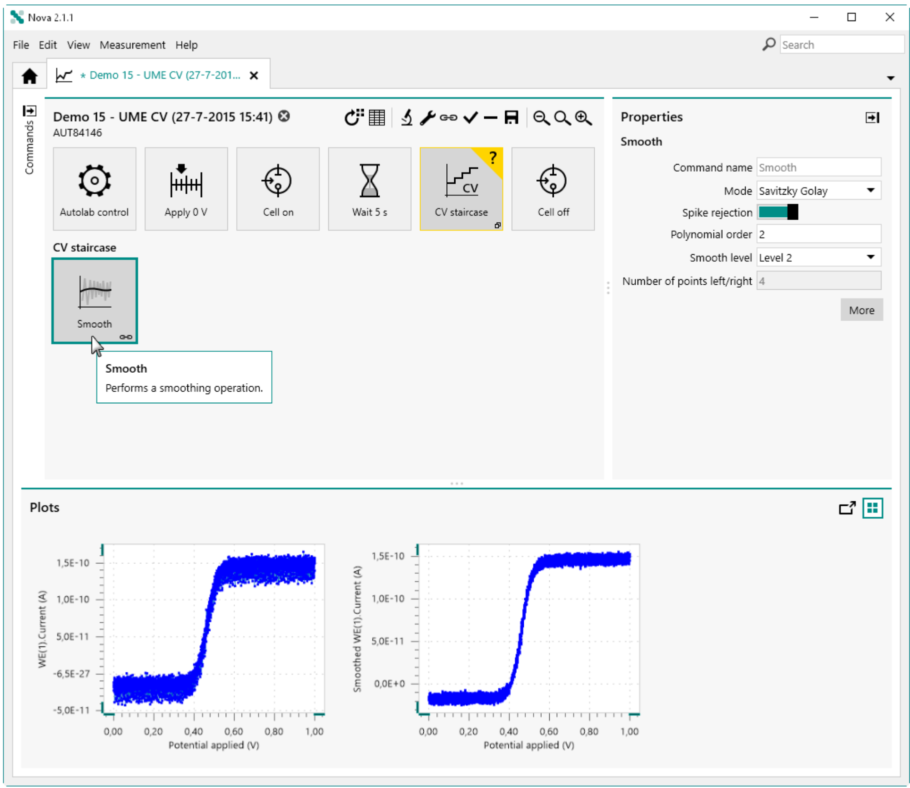


Figure 890 The Smooth command is added to the procedure

NOTE

For more information on the properties of the **Smooth** command, please refer to *Chapter 7.8.1*.

Clicking the More button opens a new screen in which the additional controls of the **Smooth** command are shown for the scope of data analysis. The plot on the left hand side shows the source data and the curve drawn by the **Smooth** command. The properties of the **Smooth** command are all set to their default values *(see Figure 891, page 730)*.

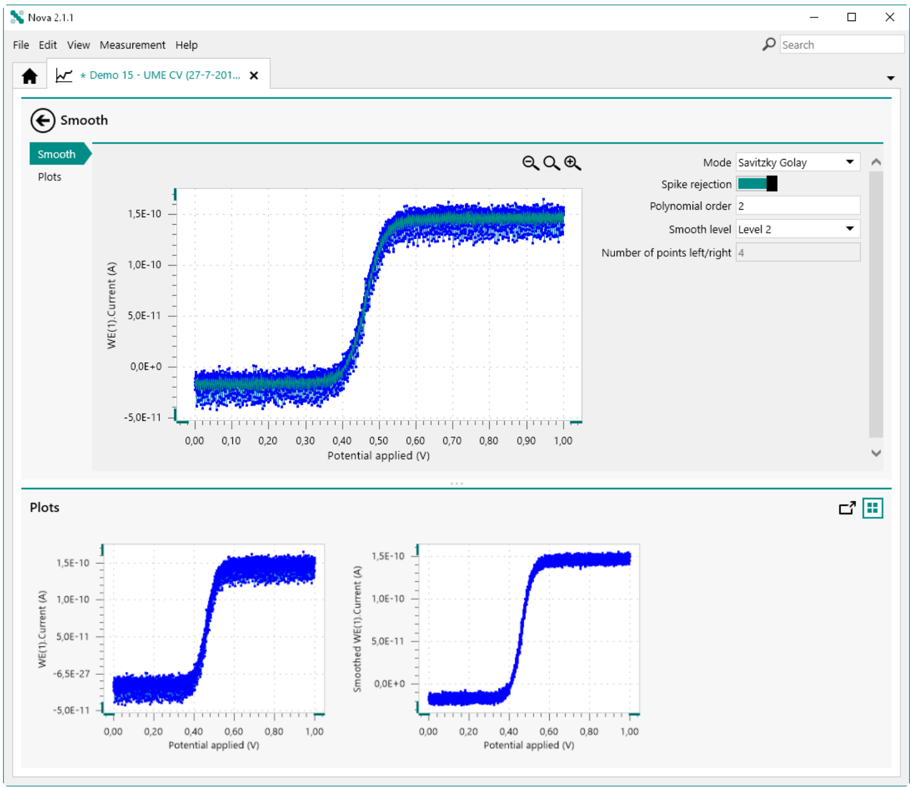


Figure 891 The additional controls of the Smooth command

Since the **Smooth** command has two different modes, each mode provides dedicated controls. The *Mode* drop-down list can be used to change the mode of the command.

12.1.1 SG mode

In *Savitzky-Golay* mode (SG), the **Smooth** command applies a smoothing algorithm based on the Savitzky-Golay algorithm. The properties can be adjusted in the panel on the right hand side *(see Figure 892, page 731)*.

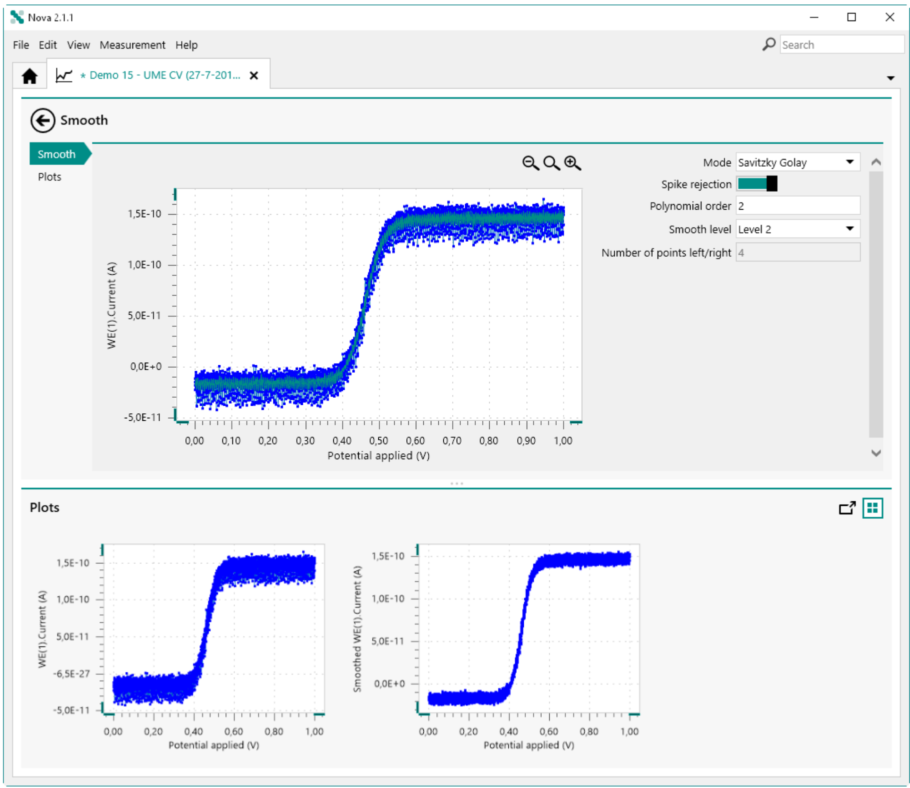


Figure 892 The Smooth command with the default properties of the Savitzky-Golay mode

A preview of the smoothed curved, obtained by using the properties defined on the right hand side, is shown in green, overlayed on the source data. Using this preview, it is possible to fine tune the properties and see the effect on the expected result of the smoothing *(see Figure 893, page 731)*.

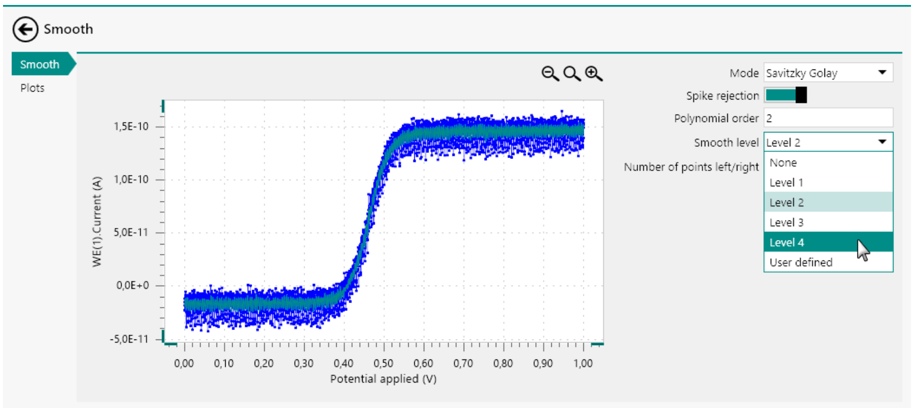


Figure 893 The preview curve is automatically adjusted when the properties are changed in the panel on the right hand side

NOTE

The smoothed plot is automatically adjusted each time one of the properties is adjusted.

12.1.2 FFT mode

In *FFT* mode, the **Smooth** command applies transform the source data to the frequency domain using the Fast Fourier Transform (FFT) algorithm. The frequency domain data, calculated from the source data, is displayed in a dedicated plot above the source data *(see Figure 894, page 732)*.

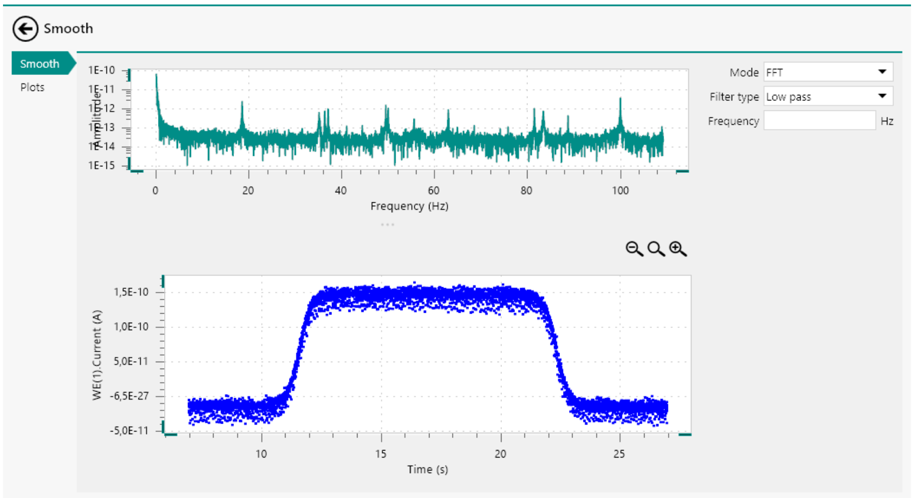


Figure 894 The FFT mode of the Smooth command

For the *Low pass* and *High pass* filter types, it is possible to specify the frequency, in Hz, directly in the panel on the right hand side or it is possible to click the FFT plot and manually position and move the vertical line indicating the frequency used by the filter. The resulting curve is shown in green, overlayed on the source data *(see Figure 895, page 733)*.

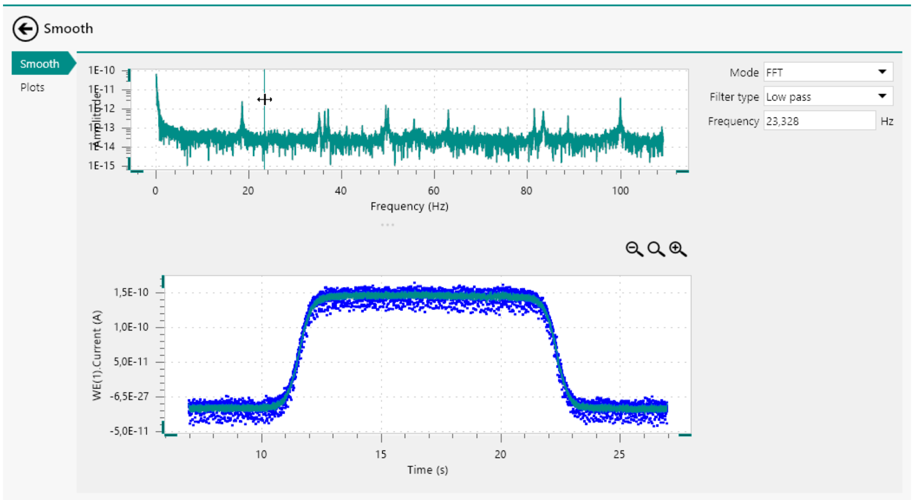


Figure 895 The frequency used by the command can be adjusted directly in the plot on the properties panel

For the *Band pass* and *Band stop* filter types, it is possible to specify the frequencies, in Hz, directly in the panel on the right hand side or it is possible to click the FFT plot and manually drag the frequency boundaries used by filter. It is possible to adjust these boundaries after placing them, manually or graphically. The smoothed plot is shown as an overlay, in green, over the source data in the plot *(see Figure 896, page 733)*.

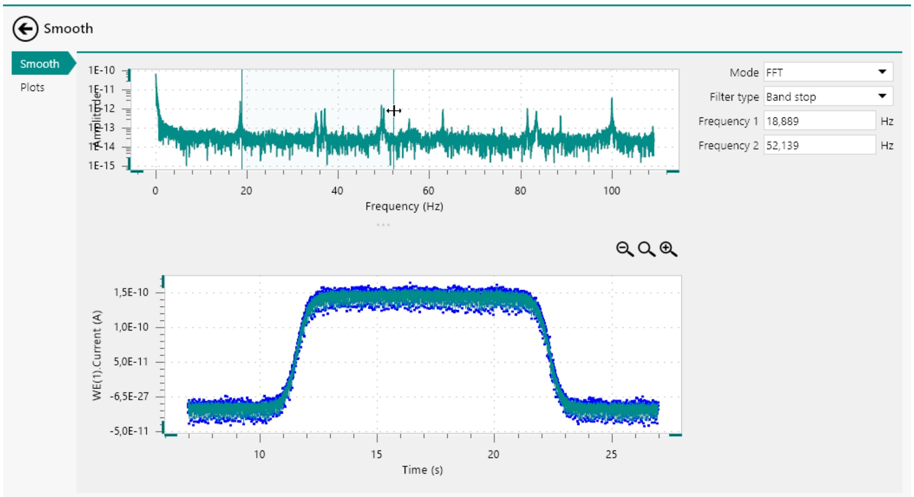


Figure 896 The frequency boundaries used by the command can be adjusted directly in the plot on the properties panel

NOTE

The smoothed plot is automatically adjusted each time one of the properties is adjusted, manually or graphically.

12.2 Peak search

The **Peak search** command provides additional controls that can be used when the command is used to analyze data. To use the **Peak search** command, this command can be added to the procedure as a command, using the drag and drop method, or by using the button. In the latter case, a pop-out menu is displayed, providing a list of commands and possible plots on which these command can be applied *(see Figure 897, page 734)*.

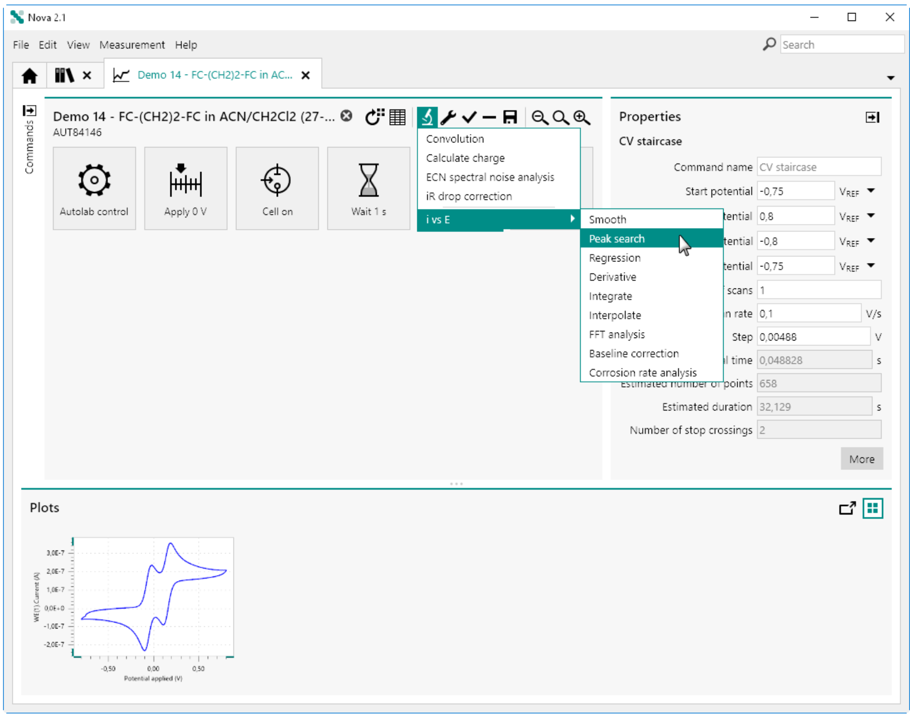


Figure 897 Adding a Peak search command to the i vs E plot

The **Peak search** command is added to the procedure editor. Clicking the command shows the properties in the dedicated panel on the right hand side *(see Figure 898, page 735)*.

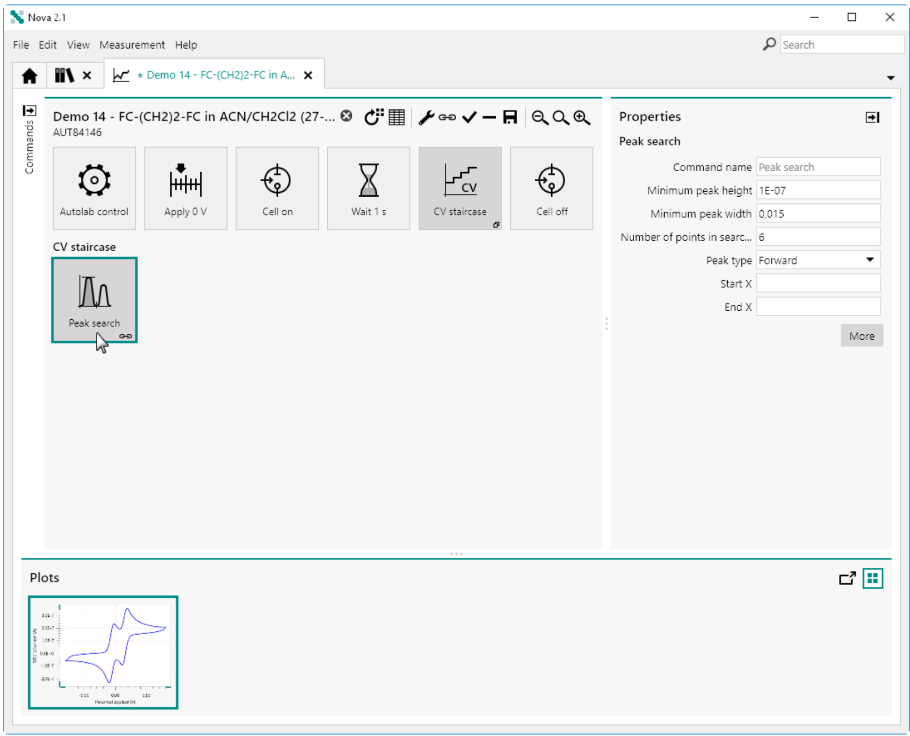


Figure 898 The Peak search command is added to the procedure

NOTE

For more information on the properties of the **Peak search** command, please refer to *Chapter 7.8.2*.

Clicking the More button opens a new screen in which the additional controls of the **Peak search** command are shown for the scope of data analysis. The plot on the left hand side shows the source data and the peaks identified by the **Peak search** command. The properties of the **Peak search** command are all set to their default values *(see Figure 899, page 736)*.

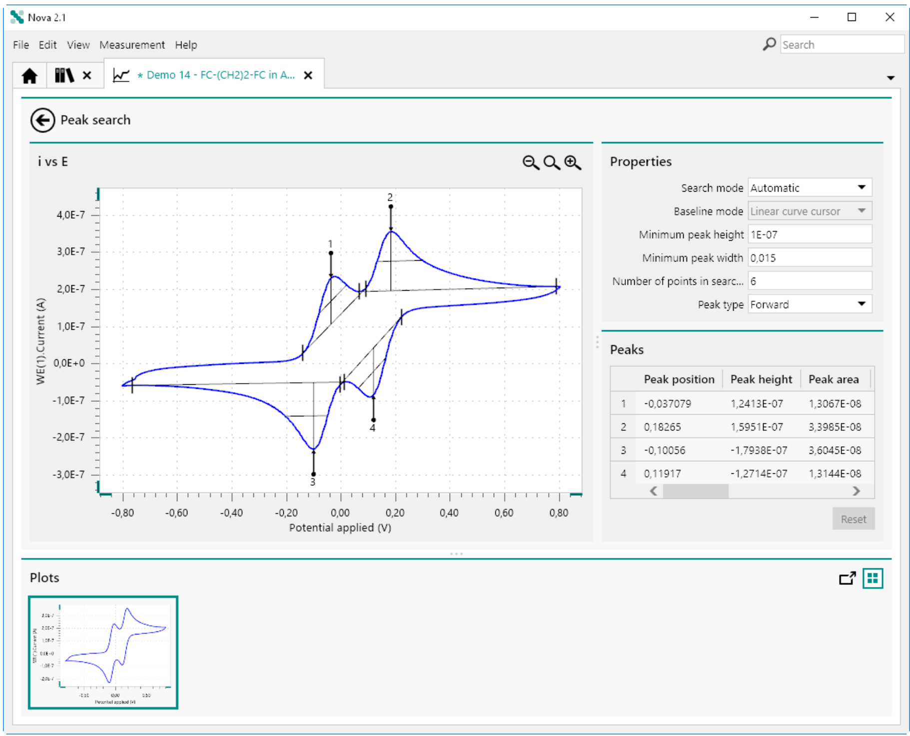


Figure 899 The additional controls of the Peak search command

Since the **Peak search** command has two different search modes *(see Figure 900, page 736)*:

- **Automatic peak:** the peaks are automatically found based on the properties defined in the panel on the right hand side.
- **Manual:** peaks are identified by specifying a baseline manually.

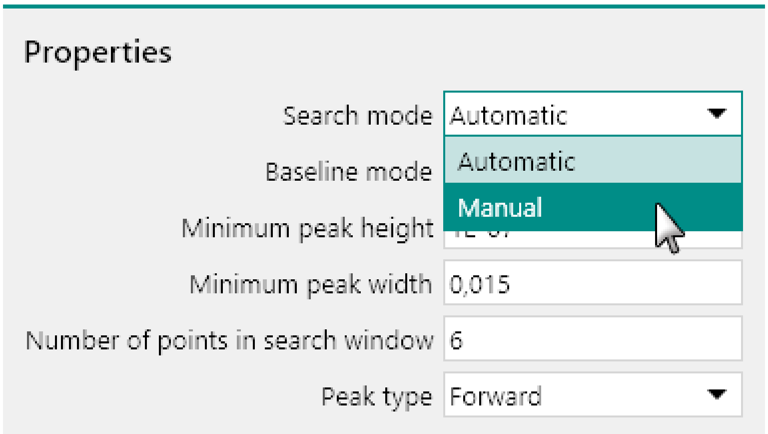


Figure 900 The search mode can be set to Automatic or Manual

CAUTION

Switching search mode will clear all previously found peaks.

12.2.1 Automatic search mode

When the *Search mode* is set to Automatic, the peaks are automatically identified by the **Peak search** command. If the search properties are modified, the command is automatically refreshed.

NOTE

The Automatic Search mode uses a Linear tangent type of baseline. This baseline is only available for the Automatic Search mode.

12.2.2 Manual peak search

When the *Search mode* is set to Manual, the type of baseline used to find the peaks can be specified using the provided *Baseline mode* dropdown list *(see Figure 901, page 737)*.

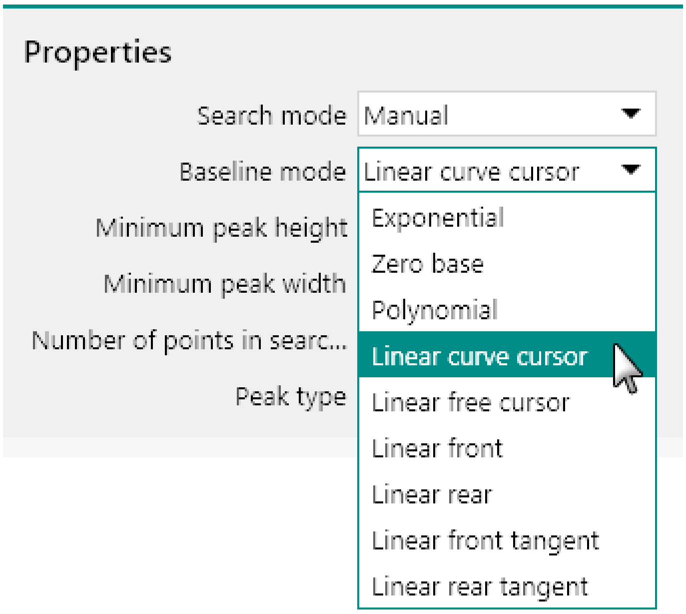


Figure 901 The base line mode can be specified using the provided dropdown list

The following baseline modes are available:

- Exponential
- Zero base
- Polynomial
- Linear curve cursor
- Linear free cursor
- Linear front
- Linear rear
- Linear front tangent

- Linear rear tangent

To carry out a manual peak search, it is necessary to draw a baseline using the mouse pointer. To do this, click the plot near the beginning of a peak and drag the mouse pointer across the plot area to draw the region in which the baseline should be located *(see Figure 902, page 738)*.

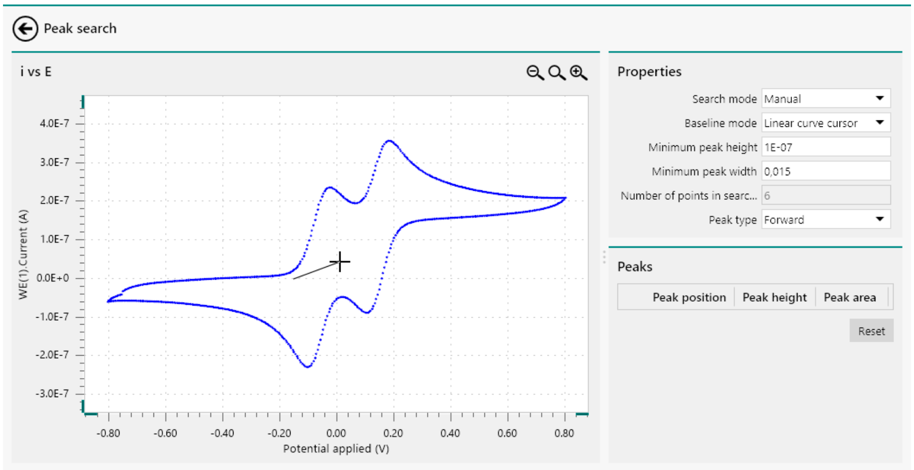


Figure 902 Defining a manual baseline

A line will be drawn as the mouse is dragged across the plot area. Click the mouse button is again to define the end of the baseline. When the baseline is defined, the peaks will be identified based on the properties defined in the **Properties** panel *(see Figure 903, page 738)*.

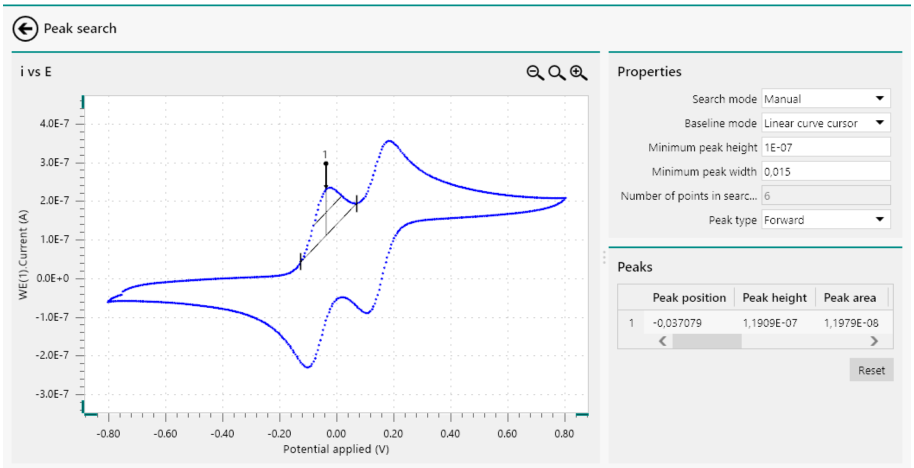


Figure 903 The baseline is drawn on the plot

NOTE

The points used to define the baseline are identified by small vertical lines on the plot.

At any point it is possible to click the Reset button to remove all the peaks found *(see Figure 904, page 739)*.

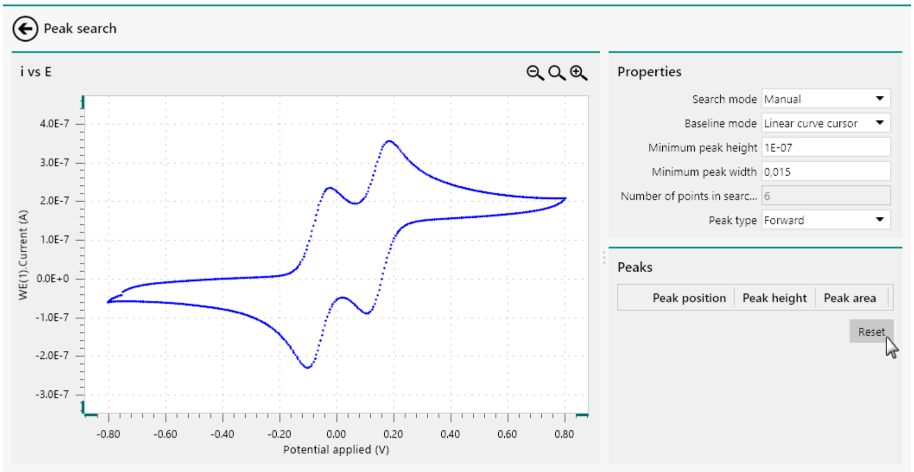


Figure 904 Resetting the peak search results

12.2.2.1 Exponential

This option uses an exponential baseline in the determination of the peaks.

To define the baseline, click on the plot area to define the start point of the baseline and drag the mouse across the plot area to define the baseline. Click the mouse button again to define the end point of the baseline. When the start point and end point of the baseline have been defined, the X coordinates of these points will be used to find the closest points on the curve and the exponential baseline will be drawn between these points *(see Figure 905, page 740)*.

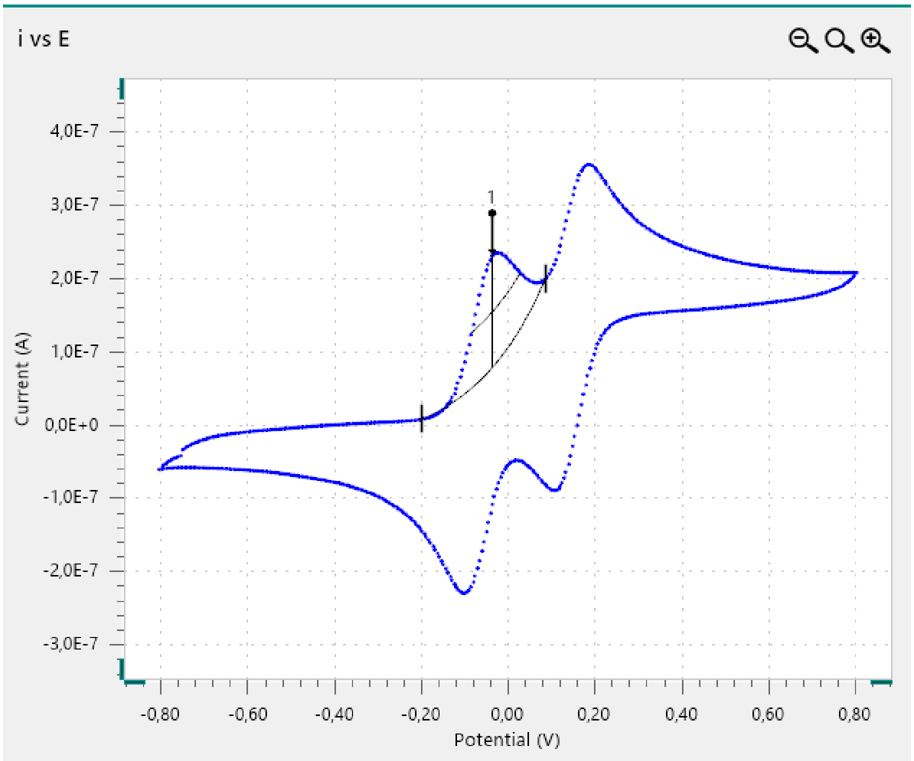


Figure 905 Manual peak search using the exponential baseline

12.2.2.2 Zero base

Using the **zero base** no baseline is used in the determination of the peak.

To define the baseline, click on the plot area to define the start point of the baseline and drag the mouse across the plot area to define the baseline. Click the mouse button again to define the end point of the baseline. When the start point and end point of the baseline have been defined the data point on Y axis, with the highest absolute value, located within the range defined by the start point and end point of the baseline is identified as a peak *(see Figure 906, page 741)*.

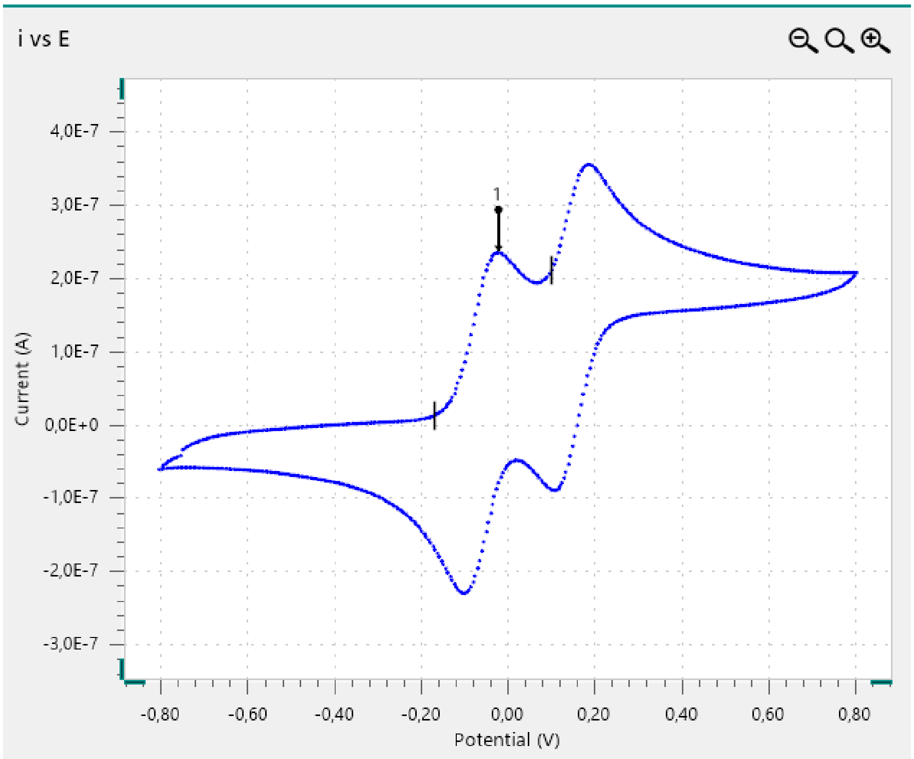


Figure 906 Manual peak search using the zero base baseline

NOTE

The zero base search method locates the absolute maximum value of the curve in the curve segment closest to the first point defining the **search window**.

12.2.2.3 Polynomial

This baseline uses a polynomial function in the determination of the peaks.

To define the baseline, click on the plot area to define the start point of the baseline. Drag the mouse across the plot and click again to add way-points for the polynomial function. This can be repeated as many times as required. To define the end point of the baseline, press the **[Enter]** key on the keyboard. The end point will be set to the last location of the mouse pointer.

When the end point has been defined, the X coordinates of the start point and end point will be used to find the closest points on the curve and the polynomial baseline will be drawn between these points *(see Figure 905, page 740)*.

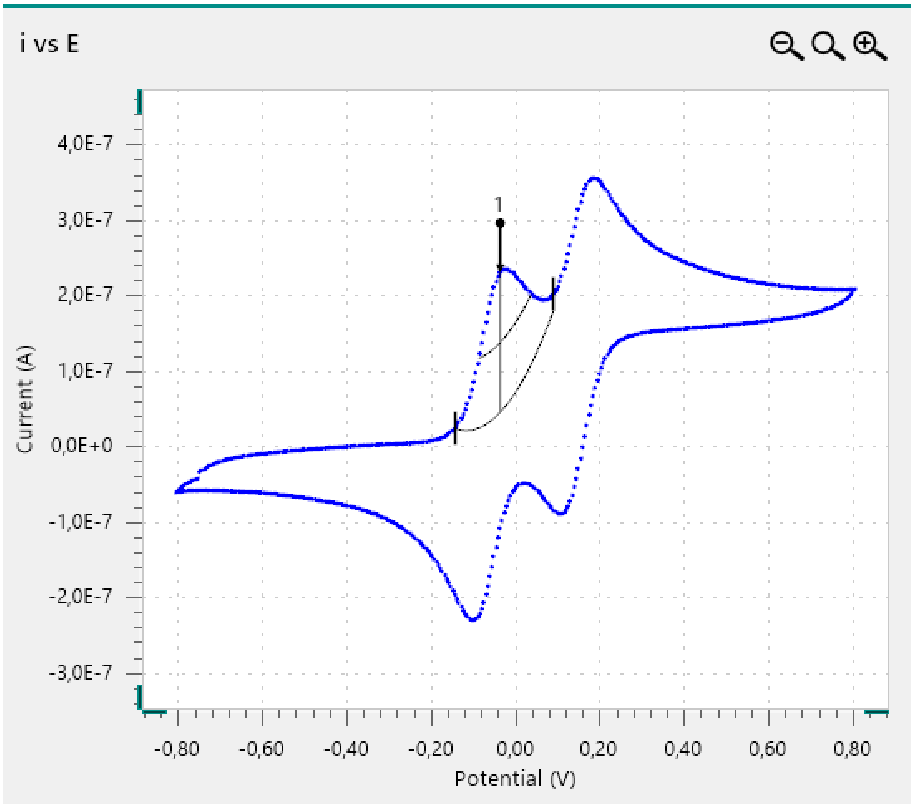


Figure 907 Manual peak search using the polynomial baseline

12.2.2.4 Linear curve cursor

This option uses a linear baseline in the determination of the peaks.

To define the baseline, click on the plot area to define the start point of the baseline and drag the mouse across the plot area to define the baseline. Click the mouse button again to define the end point of the baseline. When the start point and end point of the baseline have been defined, the X coordinates of these points will be used to find the closest points on the curve and the linear baseline will be drawn between these points *(see Figure 908, page 743)*.

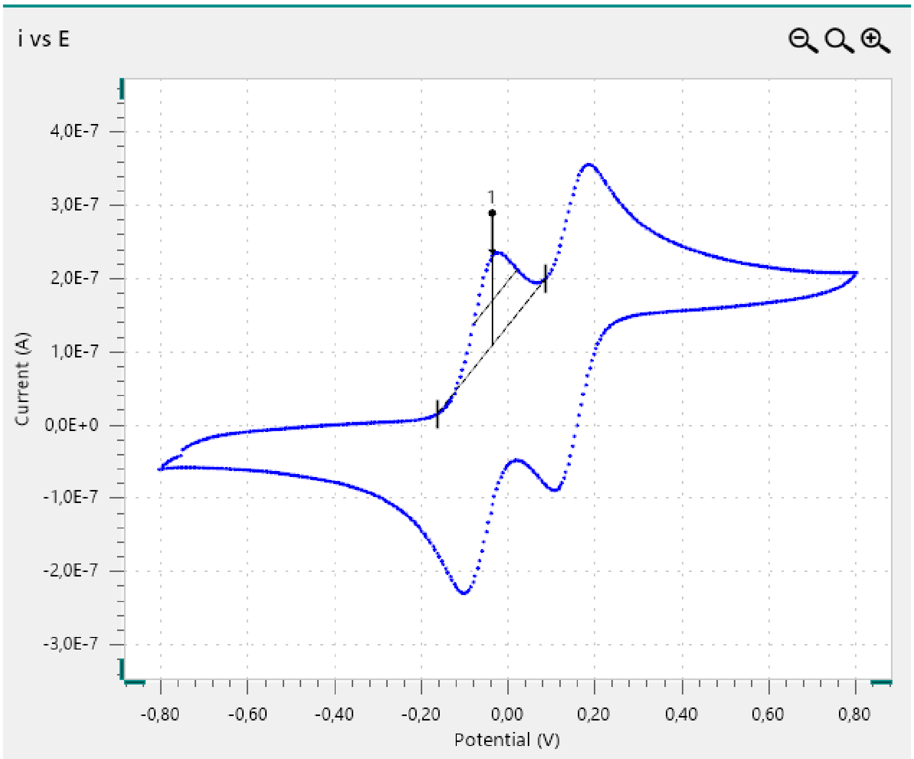


Figure 908 Manual peak search using the linear curve cursor baseline

12.2.2.5 Linear free cursor

This option uses a linear baseline in the determination of the peaks.

To define the baseline, click on the plot area to define the start point of the baseline and drag the mouse across the plot area to define the baseline. Click the mouse button again to define the end point of the baseline. This baseline is not connected to the nearest data points on the curve *(see Figure 909, page 744)*.

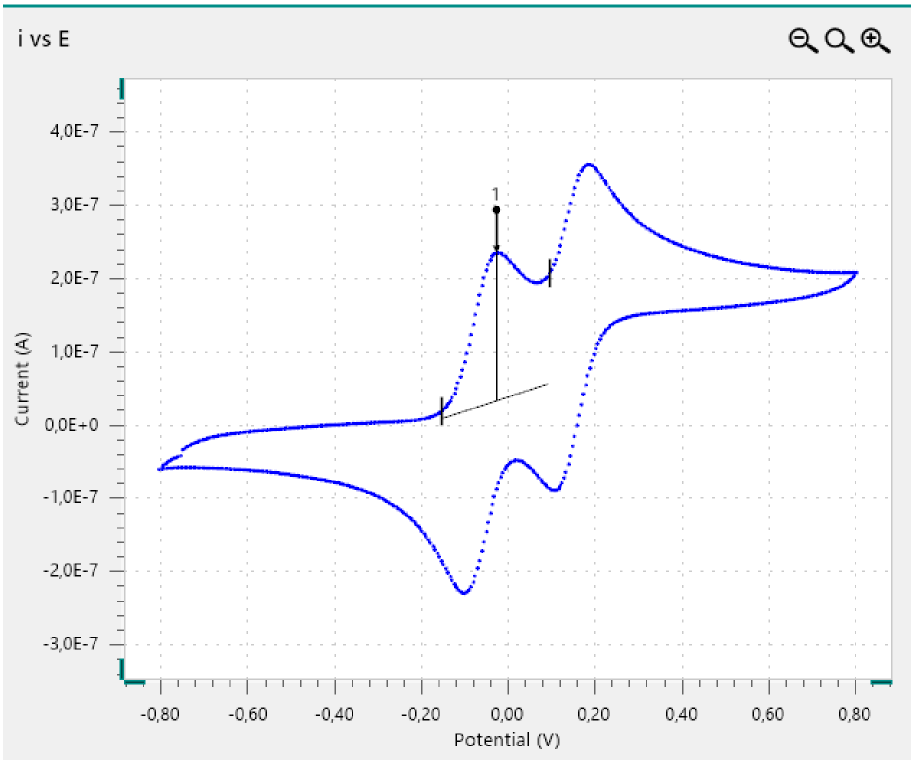


Figure 909 Manual peak search using the linear free cursor baseline

12.2.2.6 Linear front

This option finds peaks by extending a tangent baseline located in front of the peak.

To define the baseline, click on the plot area to define the start point of the baseline and drag the mouse across the plot area to define the baseline. Click the mouse button again to define the end point of the baseline. When the start point and end point of the baseline have been defined, the tangent is extended frontwards and the peak is located *(see Figure 910, page 745)*.

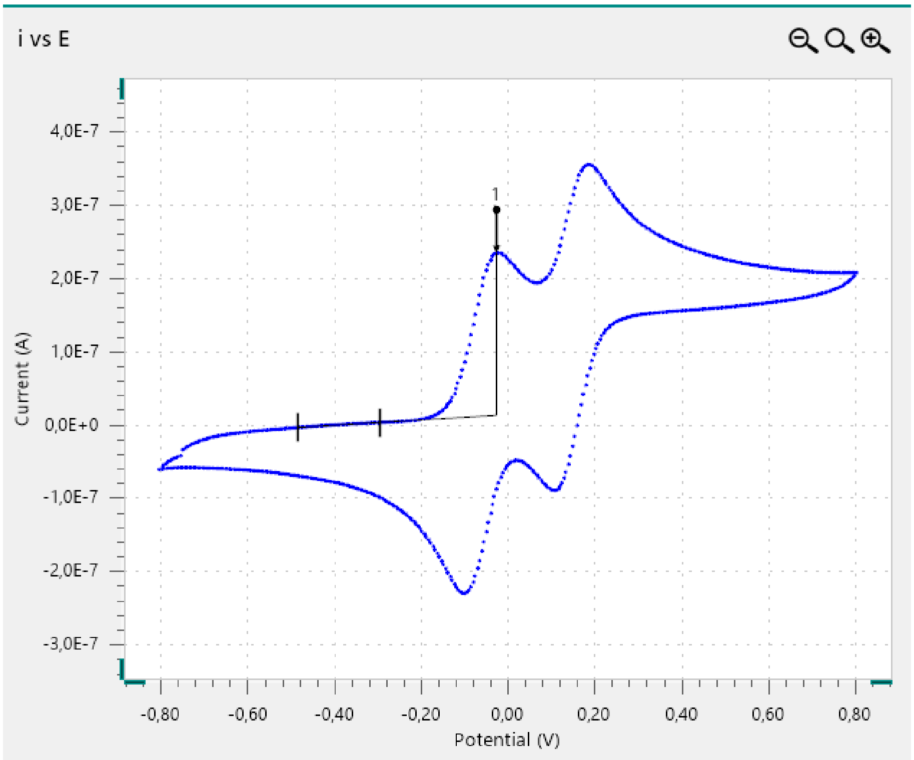


Figure 910 Manual peak search using the linear front baseline

12.2.2.7 Linear rear

This option finds peaks by extending the baseline located after the peak.

To define the baseline, click on the plot area to define the start point of the baseline and drag the mouse across the plot area to define the baseline. Click the mouse button again to define the end point of the baseline. When the start point and end point of the baseline have been defined, the tangent is extended backwards and the peak is located *(see Figure 911, page 746)*.

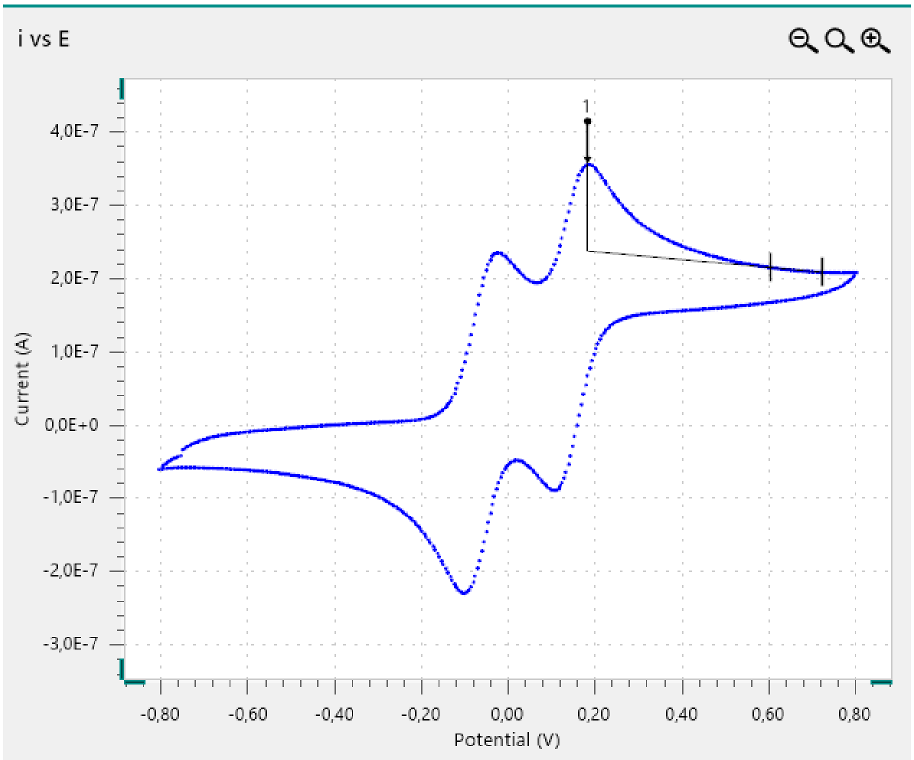


Figure 911 Manual peak search using the linear rear baseline

12.2.2.8 Linear front tangent

This option finds peaks by extending the baseline located in front of the peak.

To define the baseline, click on the plot area to define the start point of the baseline and drag the mouse across the plot area to define the baseline. Click the mouse button again to define the end point of the baseline. When the start point and end point of the baseline have been defined, the software automatically connects the baseline to the curve at the data point for which the first derivative is the closest to the slope of drawn baseline *(see Figure 912, page 747)*.

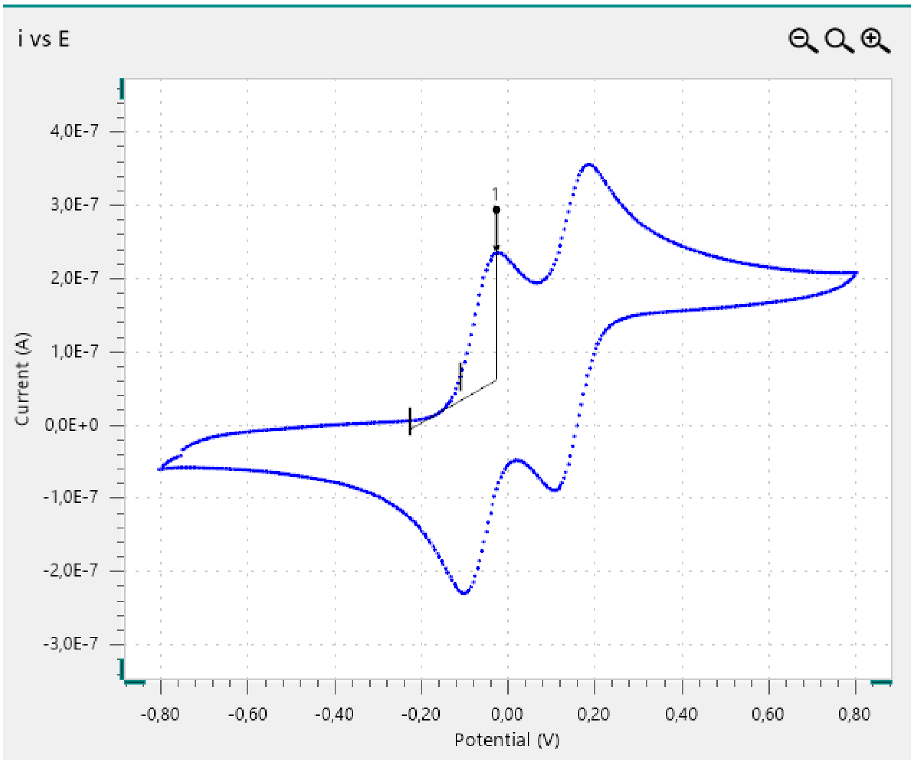


Figure 912 Manual peak search using the linear front tangent baseline

12.2.2.9 Linear rear tangent

This option finds peaks by extending the baseline located after the peak.

To define the baseline, click on the plot area to define the start point of the baseline and drag the mouse across the plot area to define the baseline. Click the mouse button again to define the end point of the baseline. When the start point and end point of the baseline have been defined, the software automatically connects the baseline to the curve at the data point for which the first derivative is the closest to the slope of drawn baseline *(see Figure 913, page 748)*.

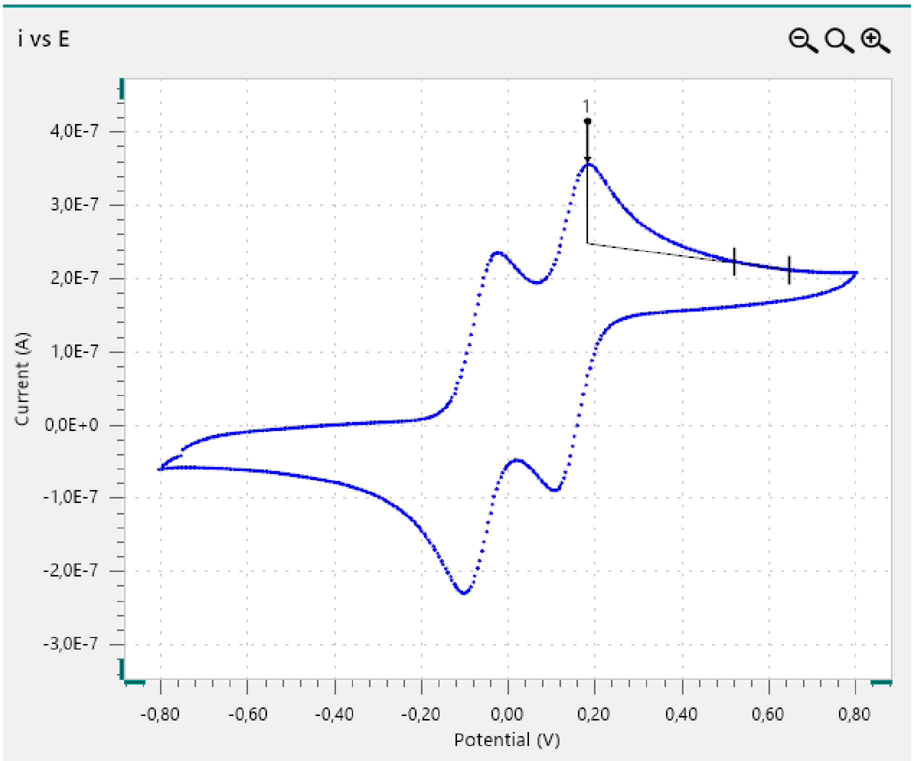


Figure 913 Manual peak search using the linear rear tangent baseline

12.2.3 Manual adjustments

Peaks found using the manual search mode can be finetuned at any time by moving one of the small vertical lines defining the location of the baseline on the plot *(see Figure 914, page 748)*.

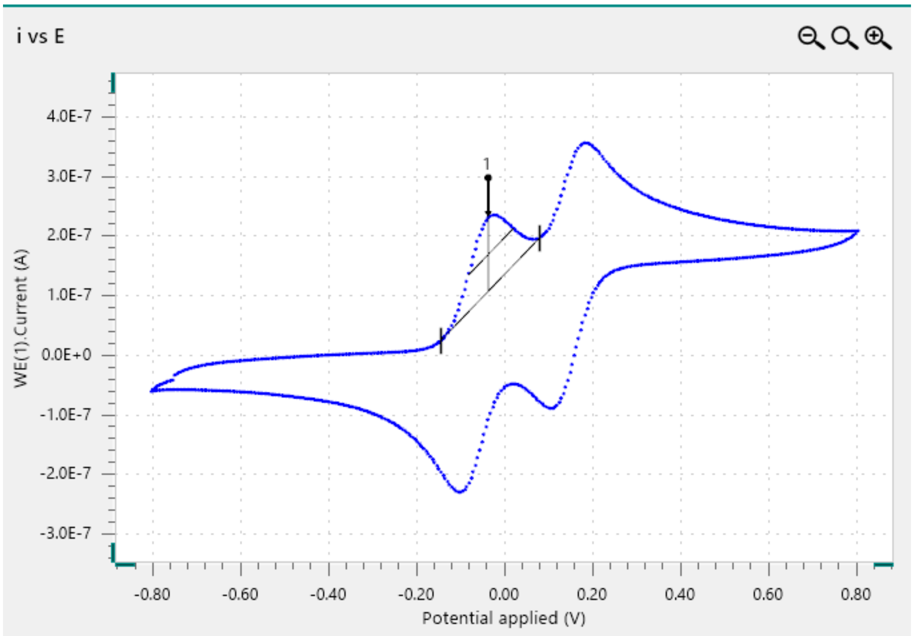


Figure 914 It is possible to modify the baseline points

To modify one of the baseline markers, click the marker and while holding the mouse button, move the marker left or right along the curve. While the mouse button is held, the coordinates of the mouse button on the curve are shown *(see Figure 915, page 749)*.

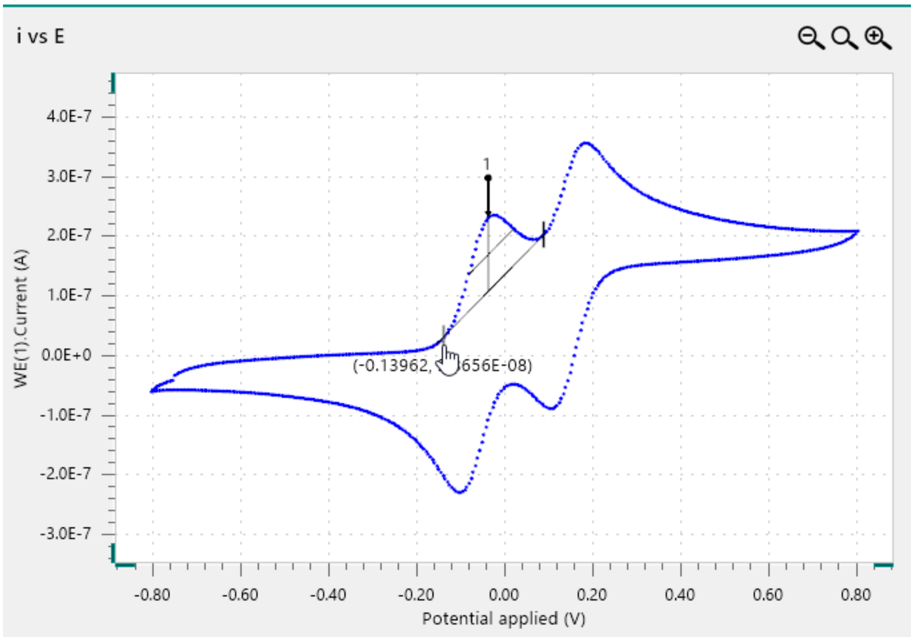


Figure 915 Moving a baseline marker shows the coordinates

When the mouse button is released, a new baseline is determined based on the new location of the displaced marker and the peak will be redetermined automatically. The results will also be automatically updated *(see Figure 916, page 749)*.

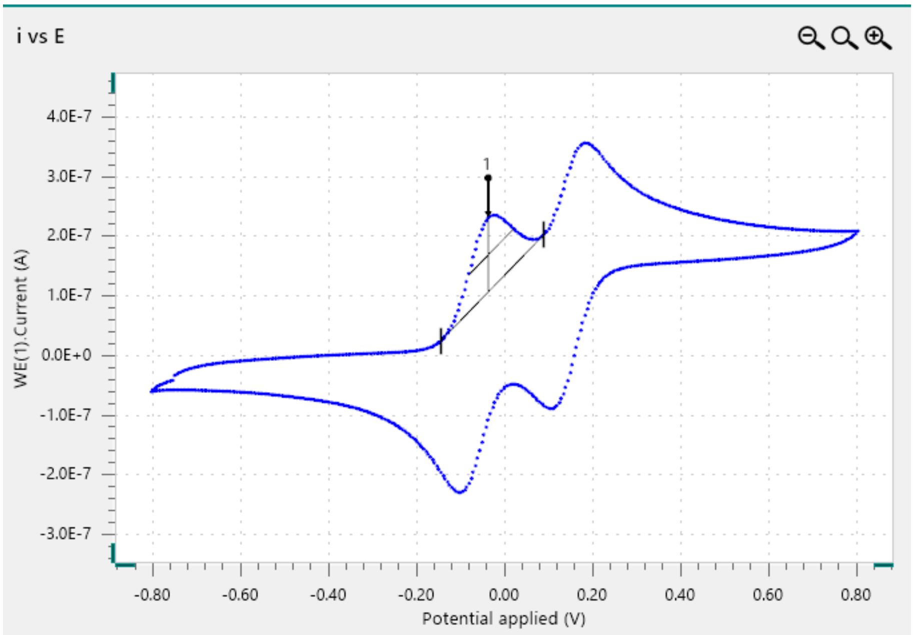


Figure 916 The new baseline is calculated and the peak is updated

12.2.4 Results

Regardless of the *Search mode* used, the peaks identified by the **Peak search** command are listed in the **Peaks** panel on the right-hand side of the plot *(see Figure 917, page 750)*.

Peaks

	Peak position	Peak height	Peak area	Base start	Base end	Peak width half height	Peak (1/2)	Peak sum of derivatives
1	-0,037079	1,2413E-07	1,3067E-08	-0,13962	0,089874	0,10276	0,045033	3,5521E-06
2	0,18265	1,5951E-07	3,3985E-08	0,06546	0,78812	0,16615	0,050395	3,3802E-06
3	-0,10056	-1,7938E-07	3,6045E-08	-0,76462	0,011749	0,15401	-0,051147	3,85E-06
4	0,11917	-1,2714E-07	1,3144E-08	-0,0028992	0,22171	0,1009	-0,044652	3,7511E-06

Reset

Figure 917 The results of the Peak search command are listed in the Peaks panel

The following results are listed for each peak:

- **Index:** this is a unique label used to identify the peak in the curve.
- **Peak position:** X axis position of the maximum Y value with respect to the baseline, in X units.
- **Peak height:** maximum Y value with respect to the baseline, in Y units.
- **Peak area:** the geometric area located between the identified peak and the baseline, in units of Y/X.
- **Base start:** X axis position of the beginning of the baseline used to locate the peak, in X units.
- **Base end:** X axis position of the end of the baseline used to locate the peak, in X units.
- **Peak width half height:** the width of the peak, in X axis units at half the value of the peak height.
- **Peak (1/2):** the difference between peak position and the peak position at half height, in X axis units.
- **Peak sum of derivatives:** the sum of the absolute values of the maximum and the minimum in the derivative of the Y signal with respect to the X signal, in Y/X units.

CAUTION

Depending on the type of base line mode used, some of the values reported in the **Results** panel may not be calculated. The *zero base* does not provide Peak area, Base start, Base end, Peak width half height, Peak (1/2) and Peak sum of derivatives. The *linear front*, *linear front tangent*, *linear rear* and *linear rear tangent* methods do not provide a value for Peak area, Peak width half height, and Peak (1/2).

12.3 Regression analysis

The **Regression** command provides additional controls that can be used when the command is used to analyze data. To use the **Regression** command, this command can be added to the procedure as a command, using the drag and drop method, or by using the button. In the latter case, a pop-out menu is displayed, providing a list of commands and possible plots on which these command can be applied *(see Figure 918, page 751)*.

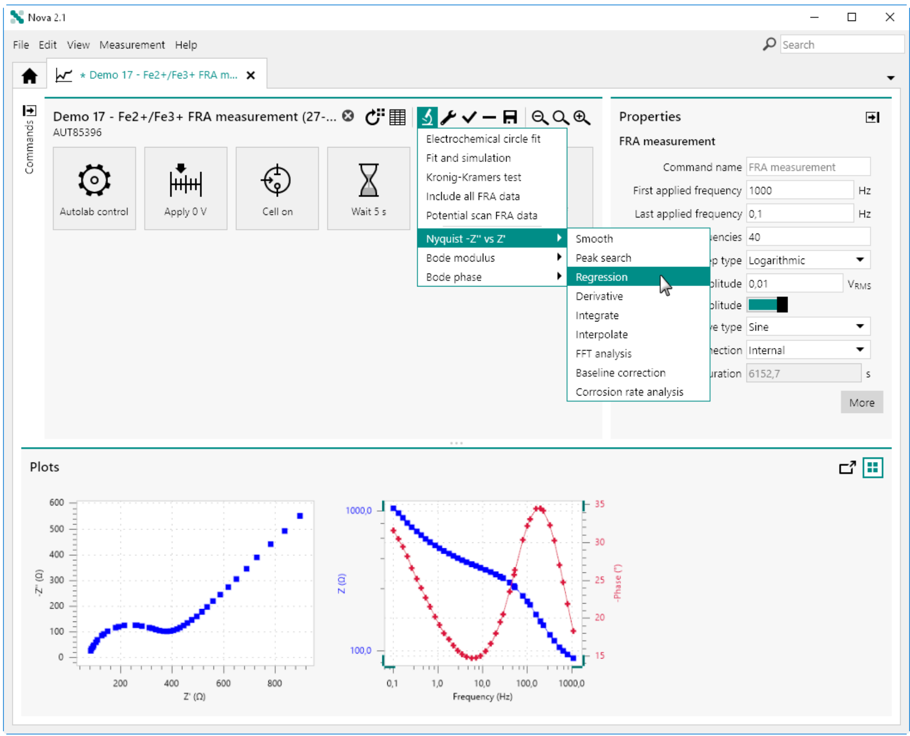


Figure 918 Adding a Regression command to the Nyquist plot

The **Regression** command is added to the procedure editor. Clicking the command shows the properties in the dedicated panel on the right hand side *(see Figure 919, page 752)*.

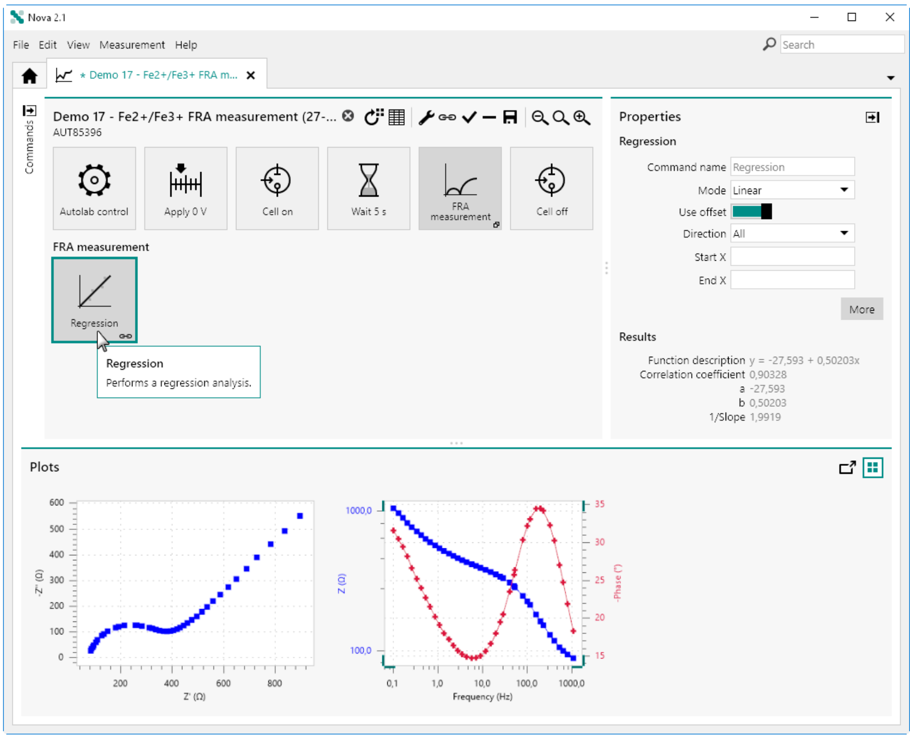


Figure 919 The Regression command is added to the procedure

NOTE

For more information on the properties of the **Regression** command, please refer to *Chapter 7.8.3*.

Clicking the More button opens a new screen in which the additional controls of the **Regression** command are shown for the scope of data analysis. The plot on the left hand side shows the source data and the curve drawn by the **Regression** command. The properties of the **Regression** command are all set to their default values *(see Figure 920, page 753)*.

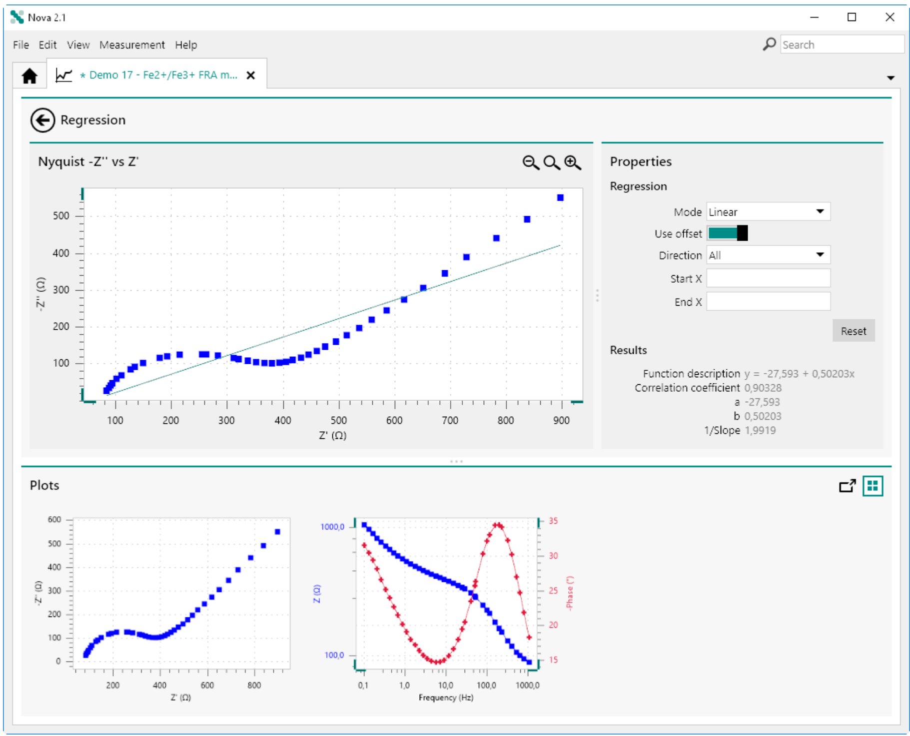


Figure 920 The additional controls of the Regression command

Using the mouse, it is possible to manually draw the area of the plot on which the **Regression** command should be executed. By clicking and holding the mouse button, a specific area can be drawn. This area will be delimited by two vertical lines and will be shown with a light green background *(see Figure 921, page 753)*.

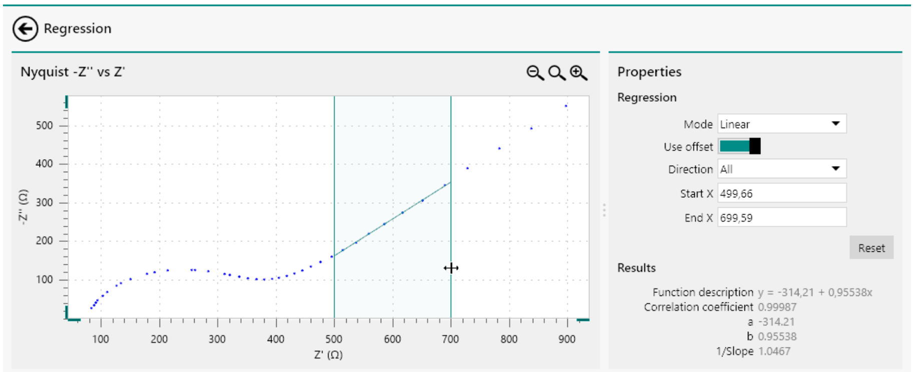


Figure 921 Manually defining boundaries for the Regression command

NOTE

The results of the **Regression** command are automatically recalculated each time one of the properties is modified or each time the **Regression** command is used on a specific area of the plot.

Once the boundaries of the **Regression** command have been specified, it is possible to manually adjust these boundaries by clicking either one of the boundary lines and dragging the line left or right *(see Figure 922, page 754)*.

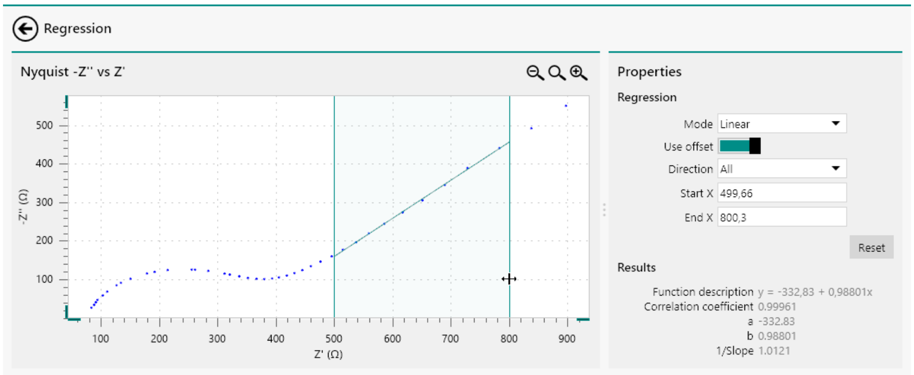


Figure 922 Adjusting the boundaries of the Regression command

It is also possible to fine tune the properties and the boundaries manually in the **Properties** panel on the right hand side *(see Figure 923, page 754)*.

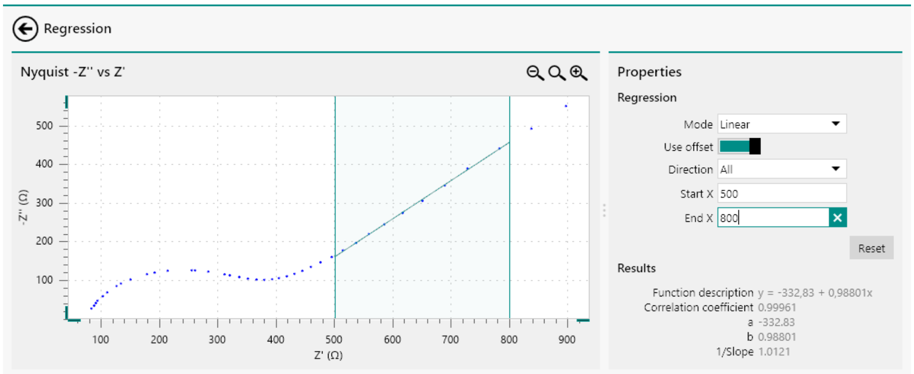


Figure 923 Finetuning the properties of the Regression command

Finally, clicking the Reset button resets all the properties of the **Regression** command back to the default values *(see Figure 924, page 754)*.

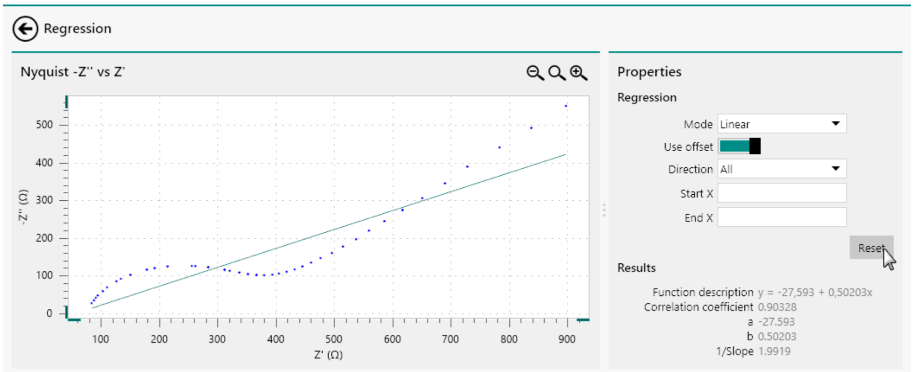


Figure 924 Resetting the properties of the Regression command

12.4 Integrate

The **Integrate** command provides additional controls that can be used when the command is used to analyze data. To use the **Integrate** command, this command can be added to the procedure as a command, using the drag and drop method, or by using the button. In the latter case, a pop-out menu is displayed, providing a list of commands and possible plots on which these command can be applied *(see Figure 925, page 755)*.

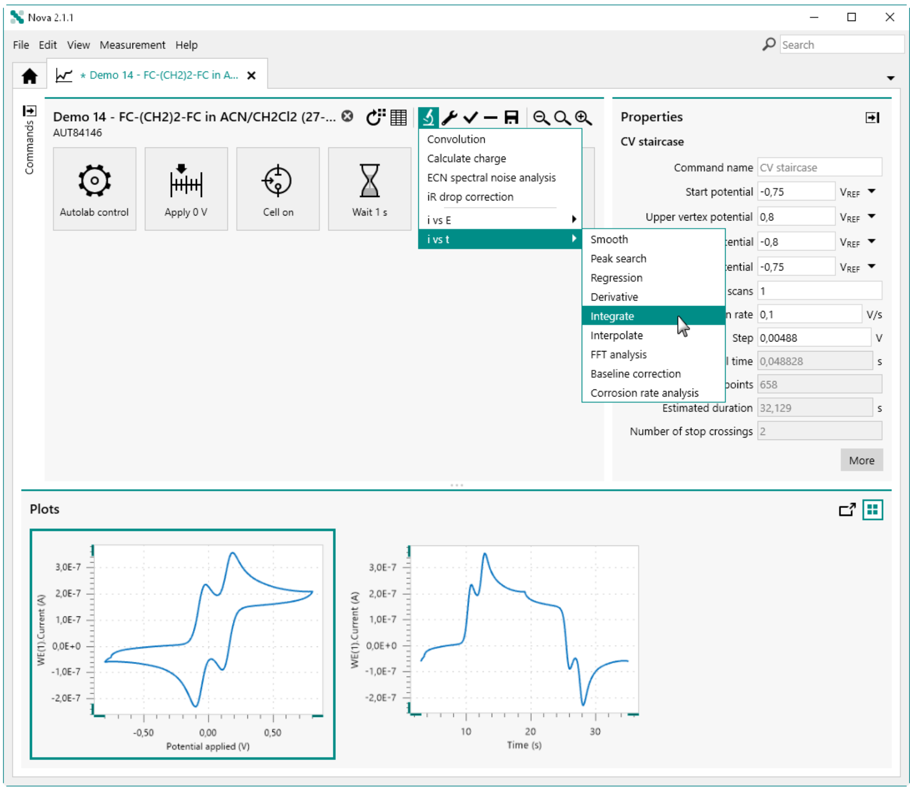


Figure 925 Adding a Integrate command to the i vs t plot

The **Integrate** command is added to the procedure editor. Clicking the command shows the properties in the dedicated panel on the right hand side *(see Figure 926, page 756)*.

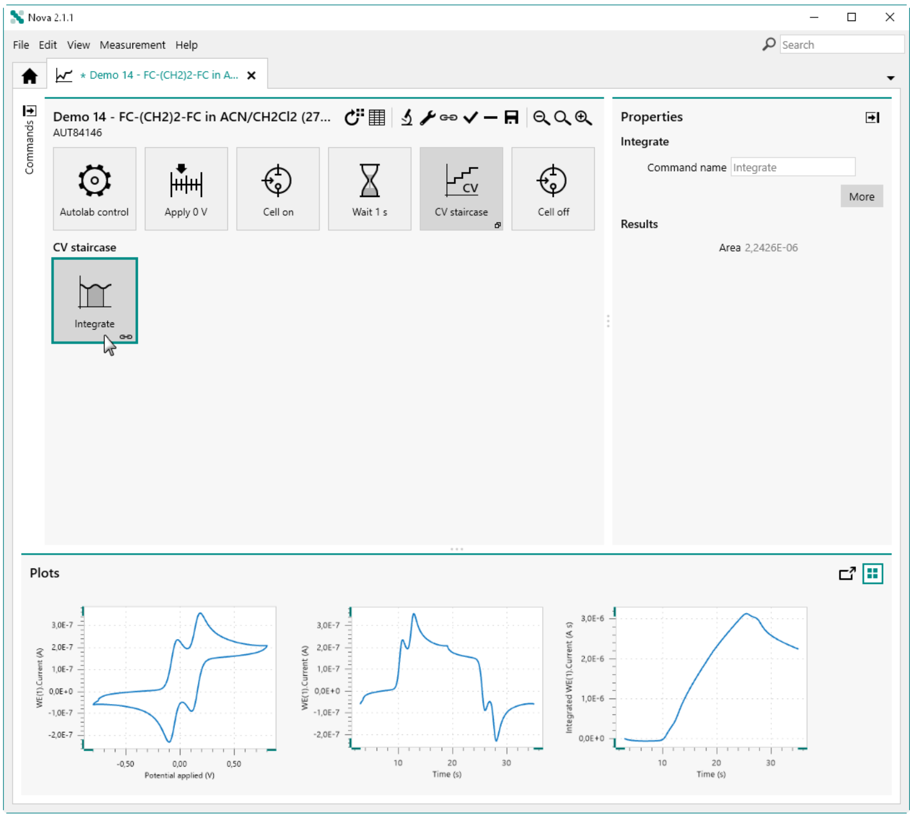


Figure 926 *The Integrate command is added to the procedure*

NOTE

For more information on the properties of the **Integrate** command, please refer to *Chapter 7.8.5*.

Clicking the More button opens a new screen in which the additional controls of the **Integrate** command are shown for the scope of data analysis. The plot on the left hand side shows the source data and the area calculated by the **Integrate** command is shown on the right-hand side of the plot *(see Figure 927, page 757)*.

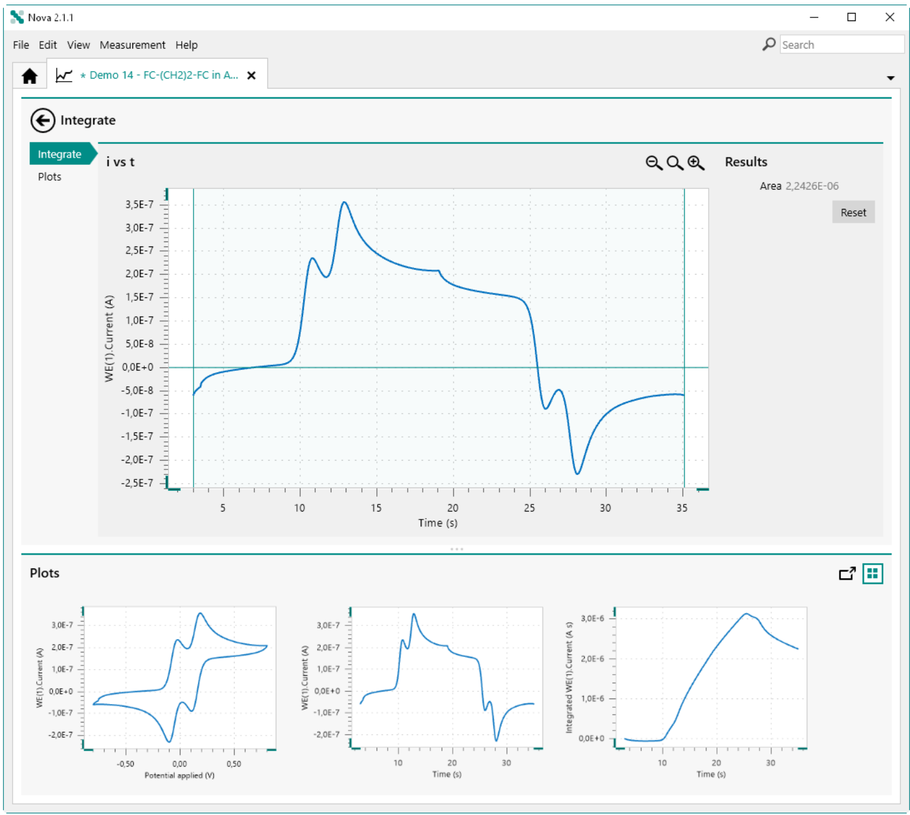


Figure 927 The additional controls of the Integrate command

By default, the whole plot is integrated. The boundaries used for the integration of the data are represented by vertical lines on either side of the plot *(see Figure 927, page 757)*. Using the mouse, it is possible to manually adjust these boundaries by clicking either one of the boundary lines and dragging the line left or right *(see Figure 928, page 758)*.

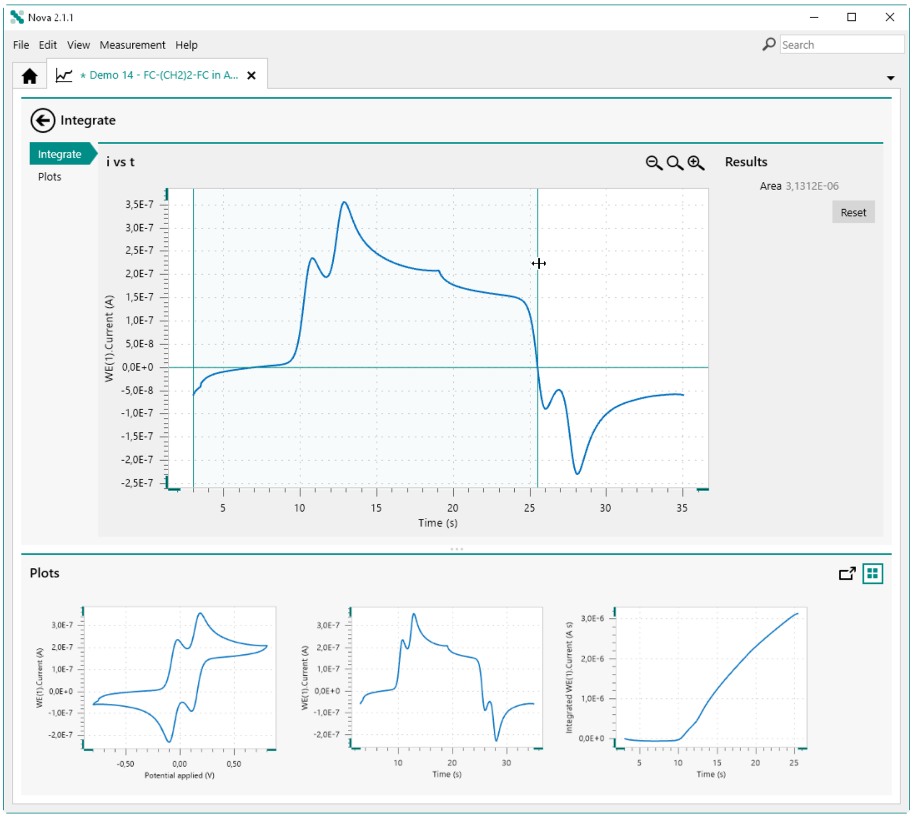


Figure 928 Adjusting the boundaries of the Integrate command

NOTE

The results of the **Integrate** command are automatically recalculated each time one of the properties is modified or each time the **Integrate** command is used on a specific area of the plot.

Finally, clicking the Reset button resets all the properties of the **Integrate** command back to the default values *(see Figure 929, page 759)*.

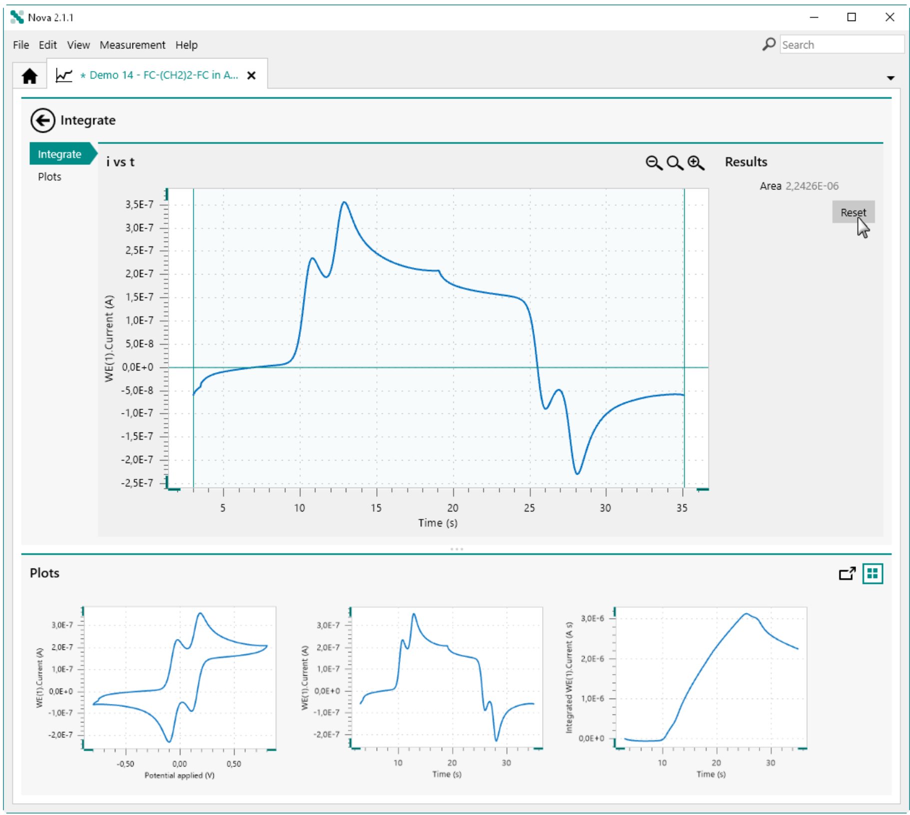


Figure 929 Resetting the properties of the Integrate command

12.5 Interpolate

The **Interpolate** command provides additional controls that can be used when the command is used to analyze data. To use the **Interpolate** command, this command can be added to the procedure as a command, using the drag and drop method, or by using the button. In the latter case, a pop-out menu is displayed, providing a list of commands and possible plots on which these command can be applied *(see Figure 930, page 760)*.

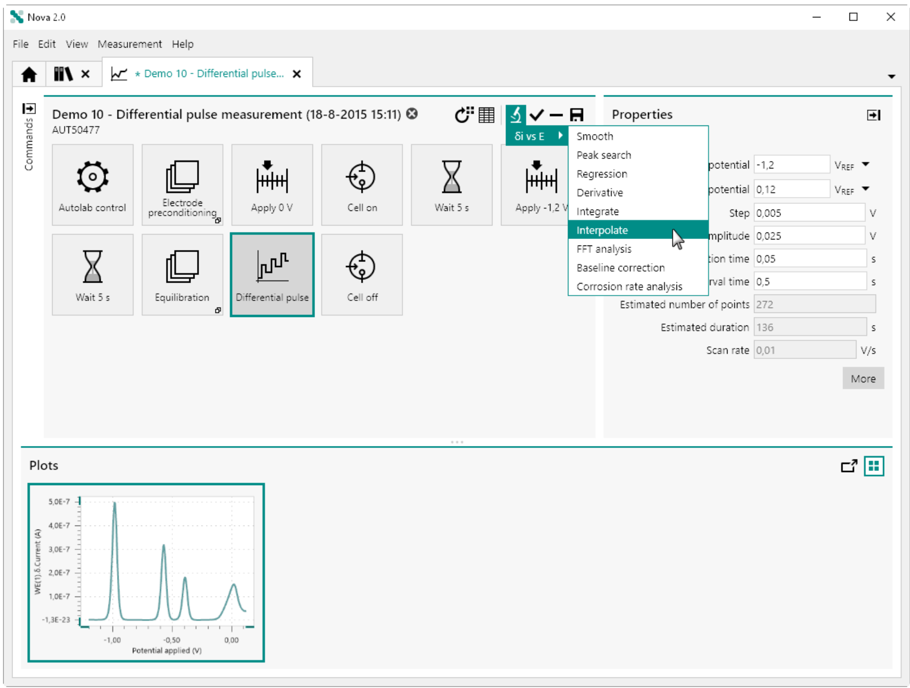


Figure 930 Adding a Interpolate command to the δi vs E plot

The **Interpolate** command is added to the procedure editor. Clicking the command shows the properties in the dedicated panel on the right hand side *(see Figure 931, page 760)*.

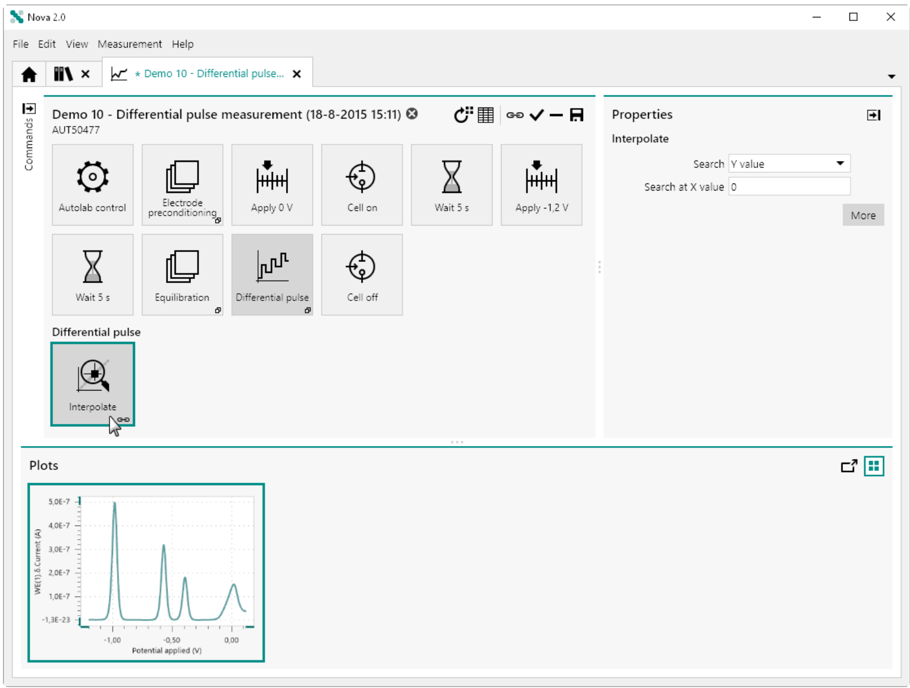


Figure 931 The Interpolate command is added to the procedure

NOTE

For more information on the properties of the **Interpolate** command, please refer to *Chapter 7.8.6*.

Clicking the More button opens a new screen in which the additional controls of the **Interpolate** command are shown for the scope of data analysis. The plot on the left hand side shows the source data and the properties of the **Interpolate** command are shown on the right-hand side of the plot *(see Figure 932, page 761)*.

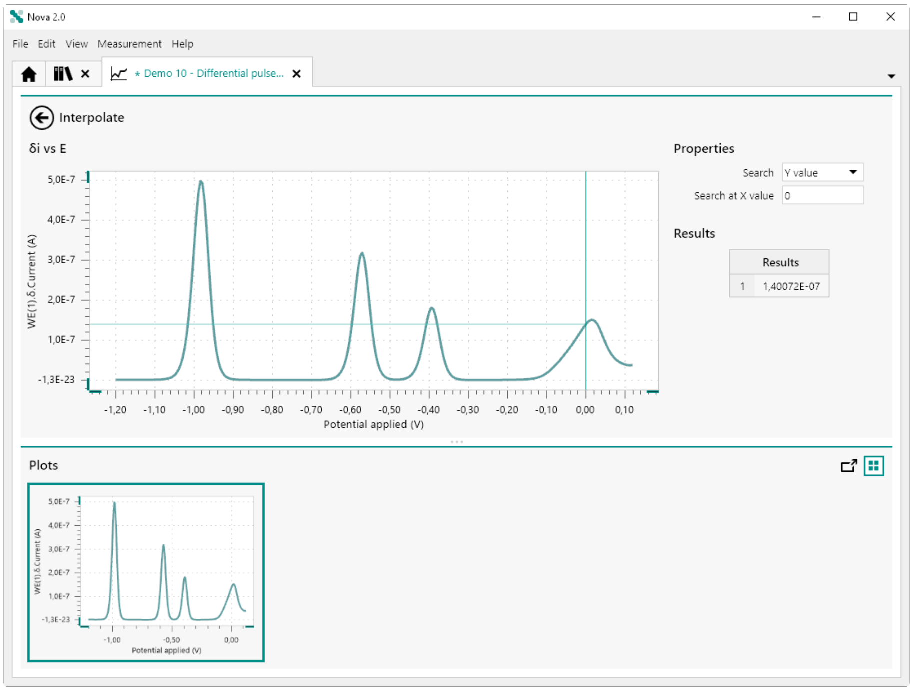


Figure 932 The additional controls of the Interpolate command

By default, the **Interpolate** command is executed, searching for Y value at a X position of 0. The results that match this search criteria are listed in the **Results** panel and indicated by the lines on the plot *(see Figure 933, page 762)*.

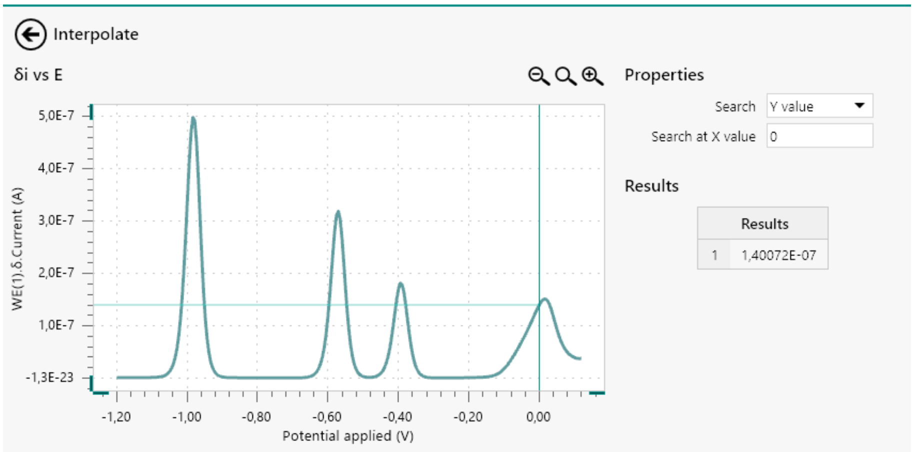


Figure 933 It is possible to adjust the properties of the Interpolate command

The dark green line indicates the location of the position at which the **Interpolate** command is carried out. Th light green line indicates the position of the value(s) found by the **Interpolate** command . It is possible to change the position at which the Interpolate command is carried out by changing the value in the provided field in the **Properties** panel *(see Figure 934, page 762)*.

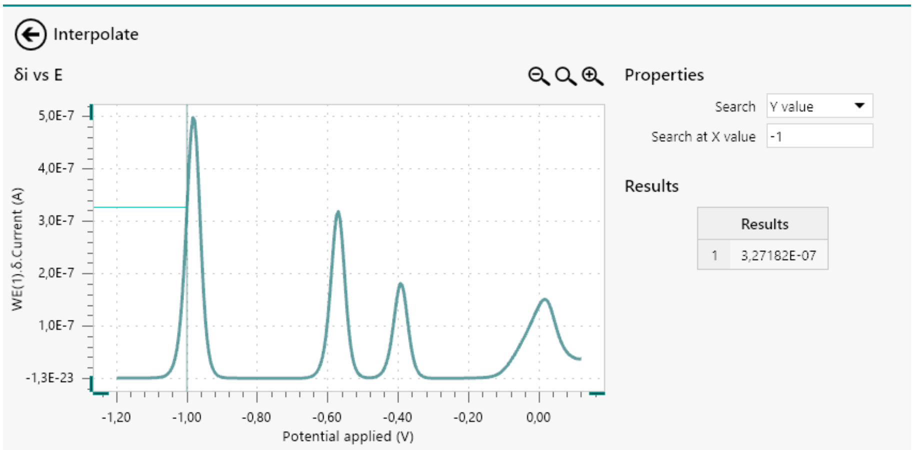


Figure 934 The Interpolate command is updated when the properties are changed

The command will be updated and the new results will be displayed graphically and in the **Results** panel. It is also possible to move the dark green line indicating the position at which the **Interpolate** command is carried out using the mouse and dragging the line across the plot *(see Figure 935, page 763)*.

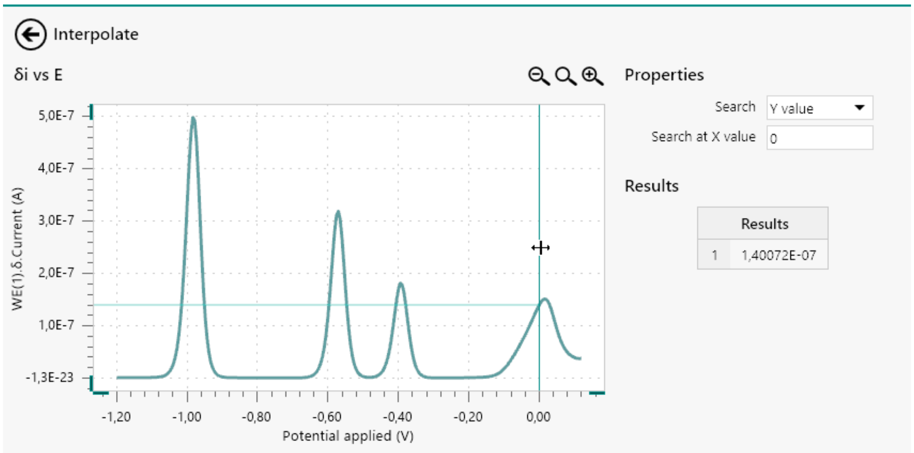


Figure 935 It is possible to move the line indicating the position at which the Interpolate command is carried out

The command will be be updated when the mouse button is released and the results will be updated as indicated above *(see Figure 936, page 763)*.

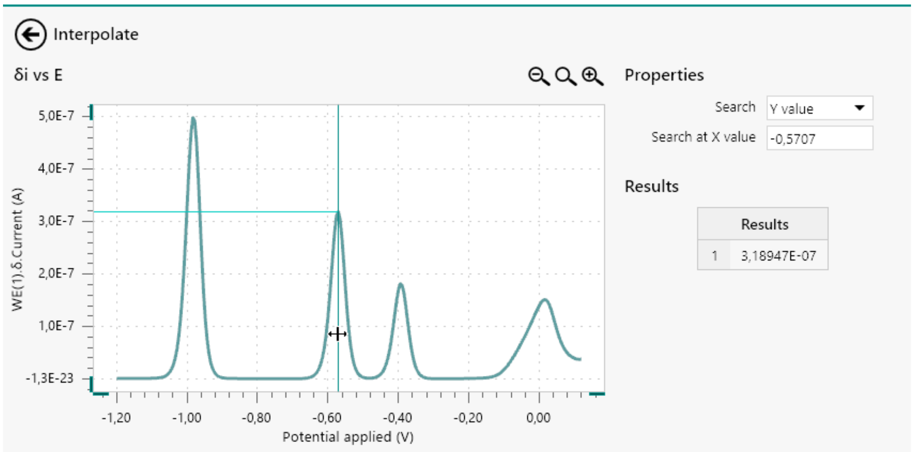


Figure 936 The Interpolate command is updated when the properties are changed

When the **Interpolate** command is able to find more than one value, as shown in *Figure 937*, each of the values found will be listed in the **Results** panel and will be indicated graphically by light green lines on the plot.

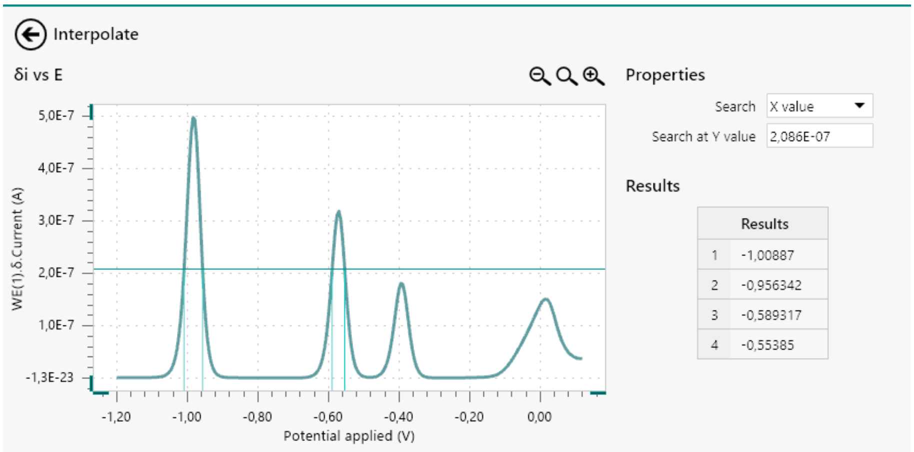


Figure 937 More than one value can be found by the Interpolate command

12.6 Hydrodynamic analysis

The **Hydrodynamic analysis** command provides additional controls that can be used when the command is used to analyze data. To use the **Hydrodynamic analysis** command, this command can be added to the procedure as a command, using the drag and drop method, or by using the button. In the latter case, a pop-out menu is displayed, providing a list of commands and possible plots on which these command can be applied *(see Figure 938, page 765)*.

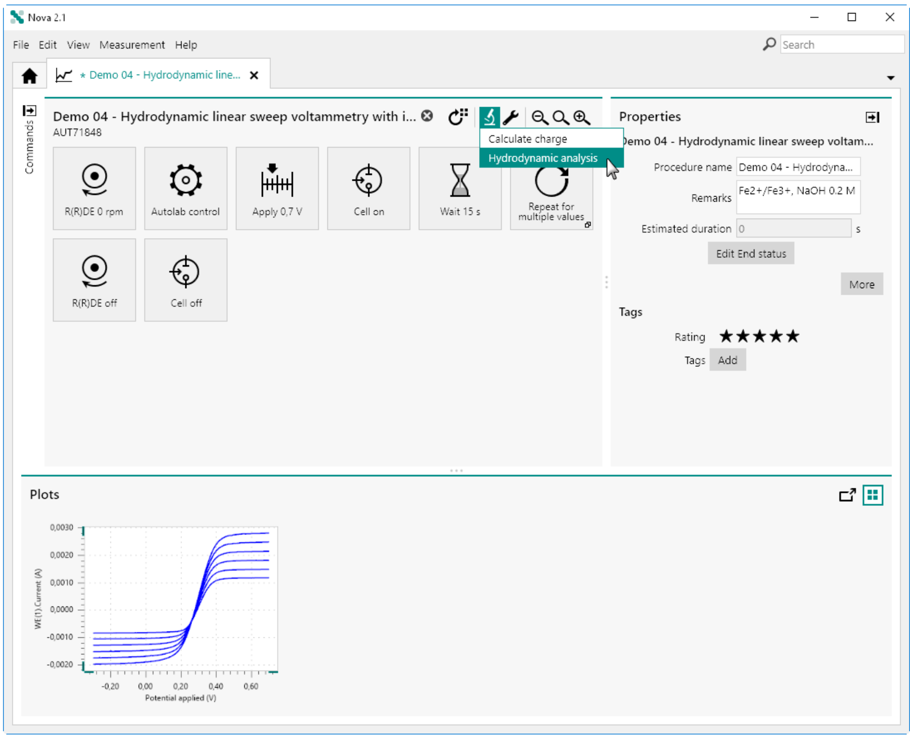


Figure 938 Adding a Hydrodynamic analysis command to the measurement

The **Hydrodynamic analysis** command is added to the procedure editor. Clicking the command shows the properties in the dedicated panel on the right hand side *(see Figure 939, page 766)*.

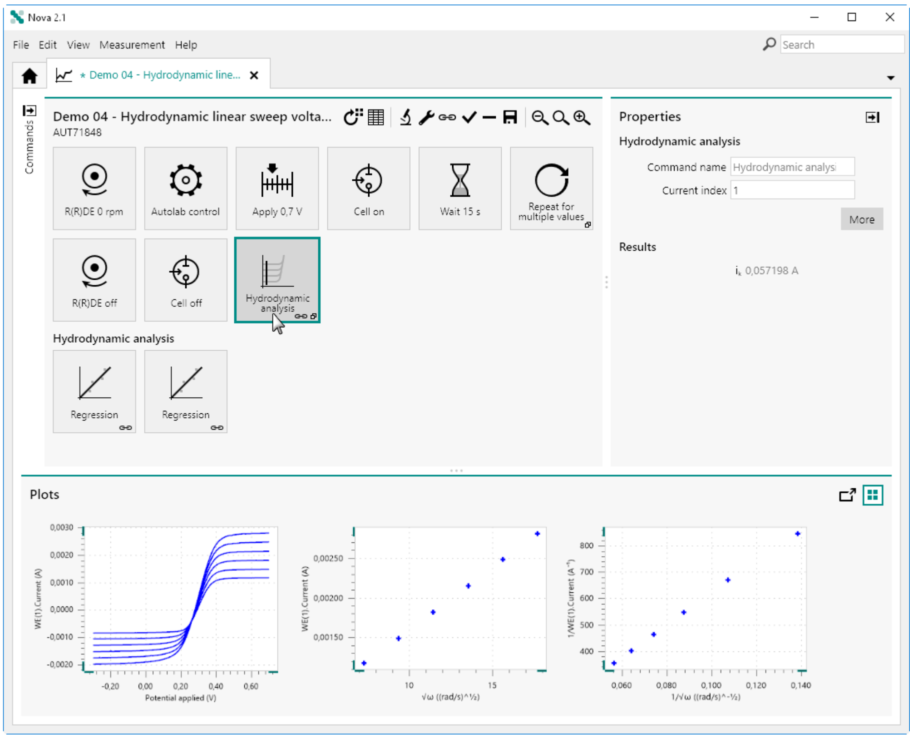


Figure 939 The Hydrodynamic analysis command is added to the procedure

> **NOTE**
>
> For more information on the properties of the **Hydrodynamic analysis** command, please refer to *Chapter 7.8.10*.

Clicking the More button opens a new screen in which the additional controls of the **Hydrodynamic analysis** command are shown for the scope of data analysis. The plot on the left hand side shows the source data. The plots on the right hand side show the regression lines generated by the command based on the selected current values *(see Figure 940, page 767)*.

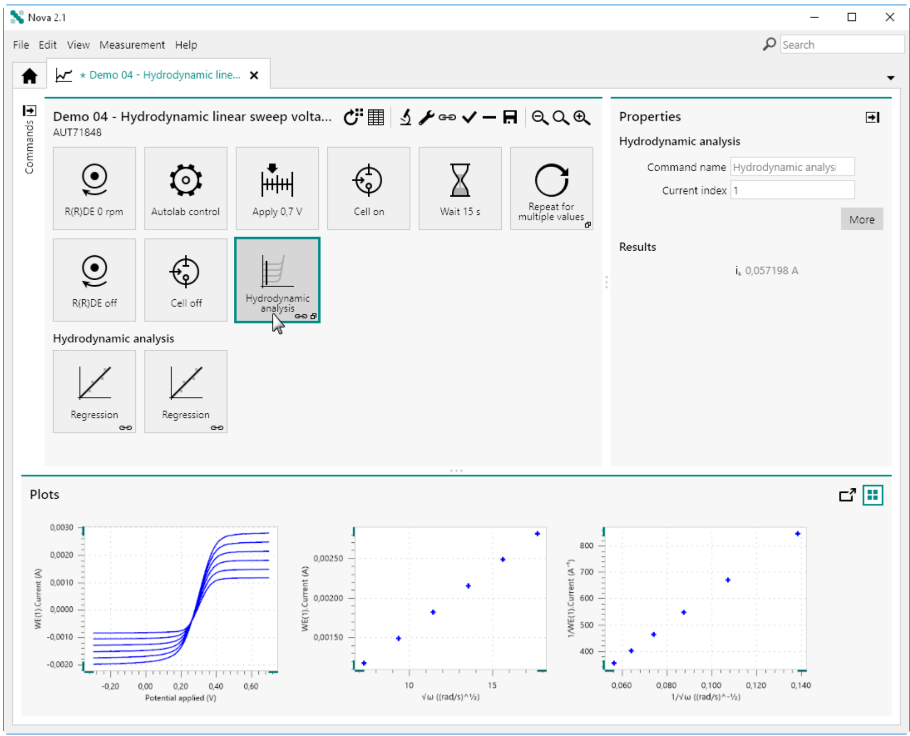


Figure 940 The additional controls of the Hydrodynamic analysis command

The currents are selected using the vertical green line show in the plot on the left hand side. By default, the line is drawn at the position of the first data point (corresponding to index 1) of each curve. To reposition the line, click the line and while holding the mouse button, slide the line across the plot area *(see Figure 941, page 767)*.

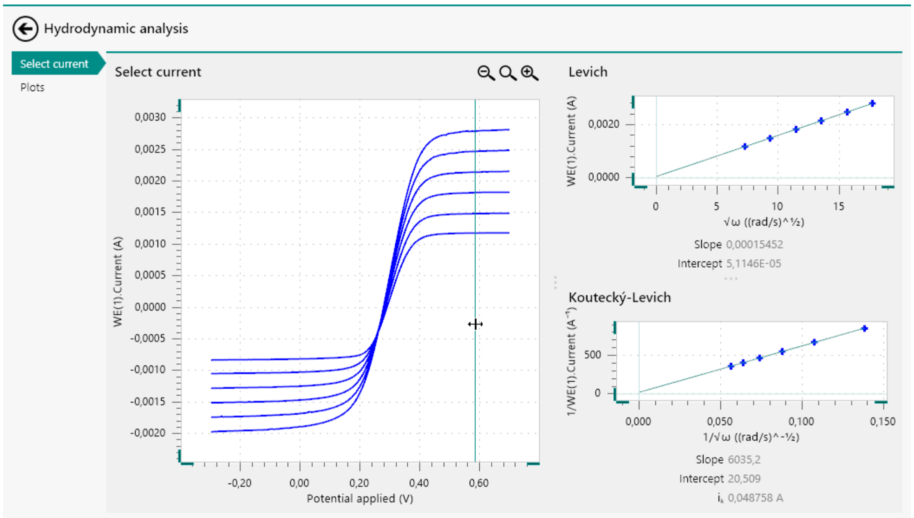


Figure 941 Moving the vertical line to specifying the current

The selected current are updated as the line is moved. Releasing the mouse button validates the selection of the limiting currents *(see Figure 942, page 768)*.

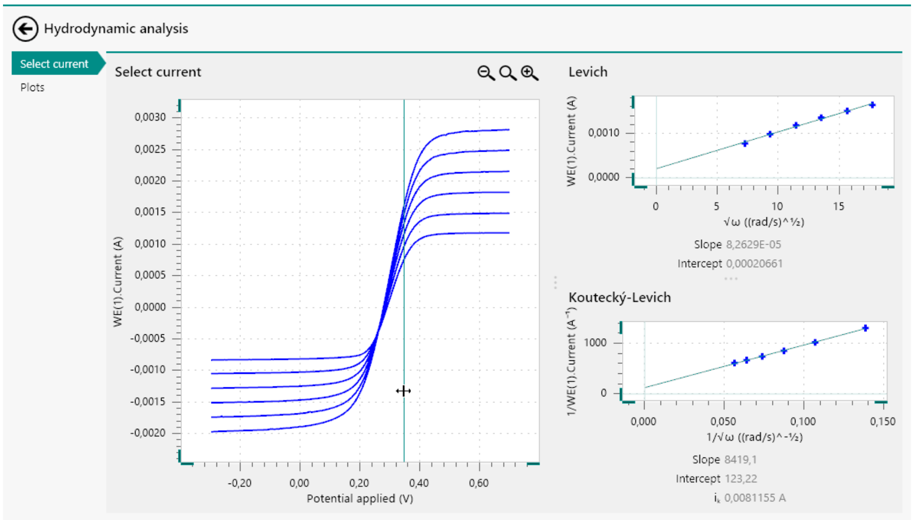


Figure 942 The current values are updated

The **Hydrodynamic analysis** command automatically carries out a Levich analysis (which is normally carried out on the mass-transport limited current values) and a Koutecký-Levich analysis (which is normally carried out in the mixed kinetic - mass-transport regime). A linear regression is carried out on both these analysis methods and the results are displayed below the corresponding plots on the right-hand side.

For the Levich plot, the Slope and Intercept are provided. For the Koutecký-Levich plot, the same information is provided, as swell as the extrapolated kinetic current, i_k, obtained from the intercept on the plot.

12.7 Baseline correction

The **Baseline correction** command provides additional controls that can be used when the command is used to analyze data. To use the **Baseline correction** command, this command can be added to the procedure as a command, using the drag and drop method, or by using the button. In the latter case, a pop-out menu is displayed, providing a list of commands and possible plots on which these command can be applied *(see Figure 943, page 769)*.

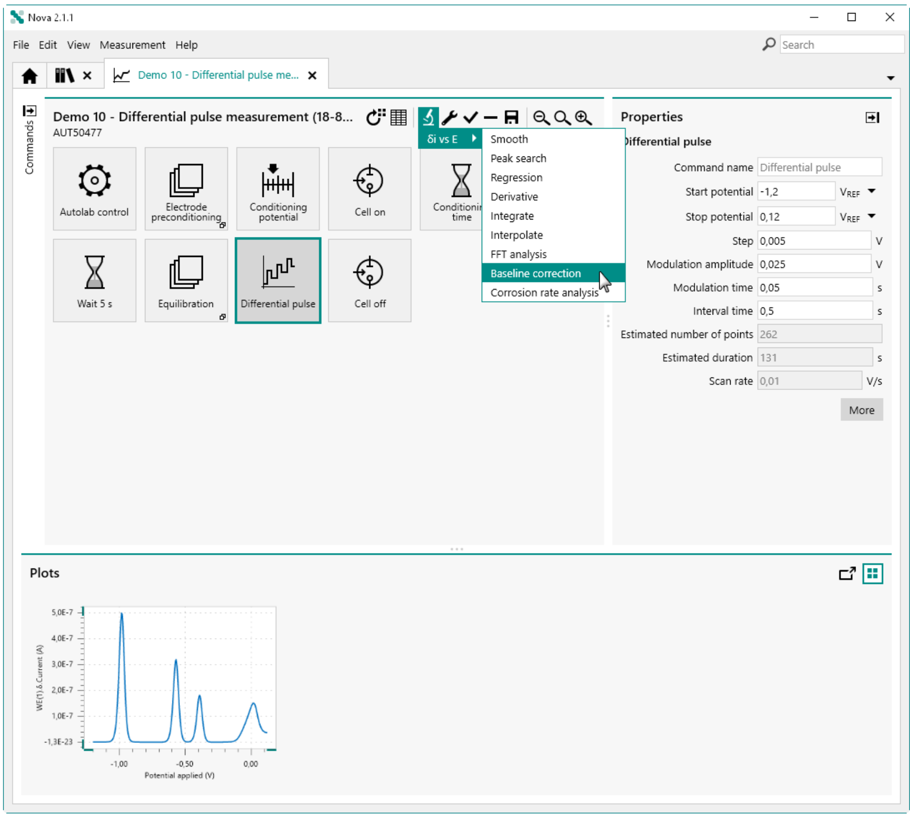


Figure 943 Adding a Baseline correction command to the Differential pulse command

The **Baseline correction** command is added to the procedure editor. Clicking the command shows the properties in the dedicated panel on the right hand side *(see Figure 944, page 770)*.

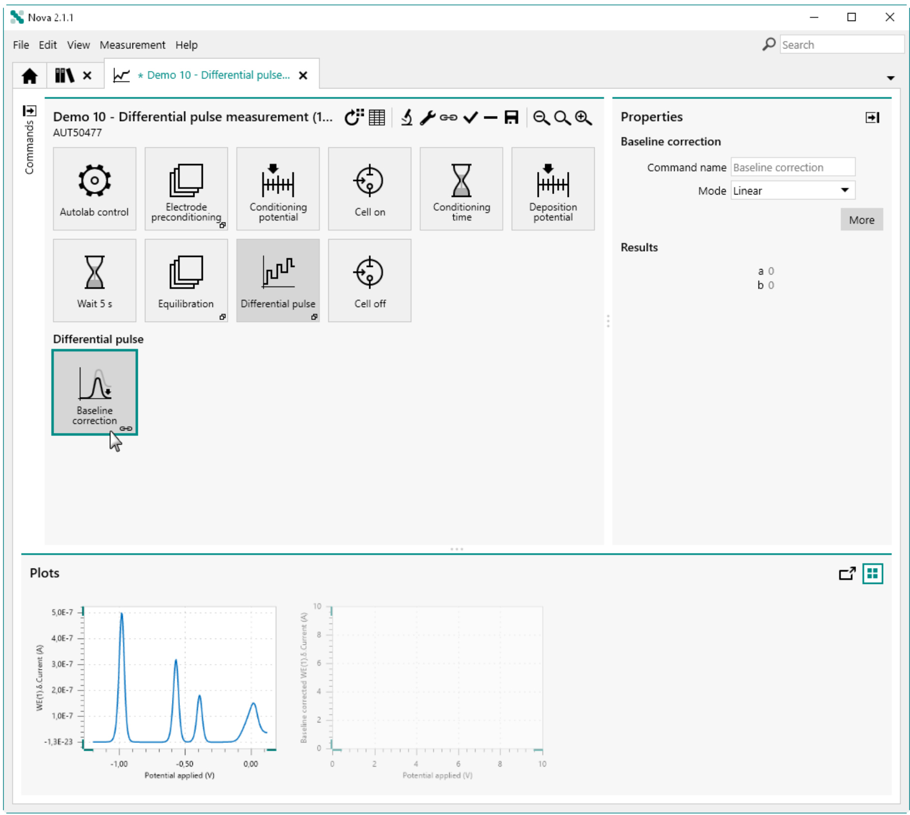


Figure 944 The Baseline correction command is added to the procedure

NOTE

For more information on the properties of the **Baseline correction** command, please refer to *Chapter 7.8.13*.

Clicking the More button opens a new screen in which the additional controls of the **Baseline correction** command are shown for the scope of data analysis. The plot on the left hand side shows the source data. The properties of the **Baseline correction** command are all set to their default values *(see Figure 945, page 771)*.

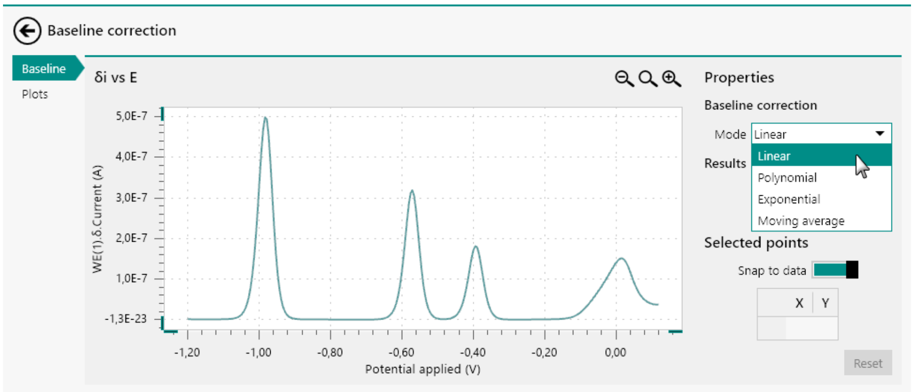


Figure 945 The additional controls of the Baseline correction command

The mode and properties of the **Baseline correction** command can be adjusted in the **Properties** panel. Using the mouse, it is possible to click the plot to define a point defining the baseline. Depending on the mode selected for the **Baseline correction** command, two or more data points are necessary to define the baseline. When sufficient data points have been defined on the plot, the baseline will be drawn *(see Figure 946, page 771)*.

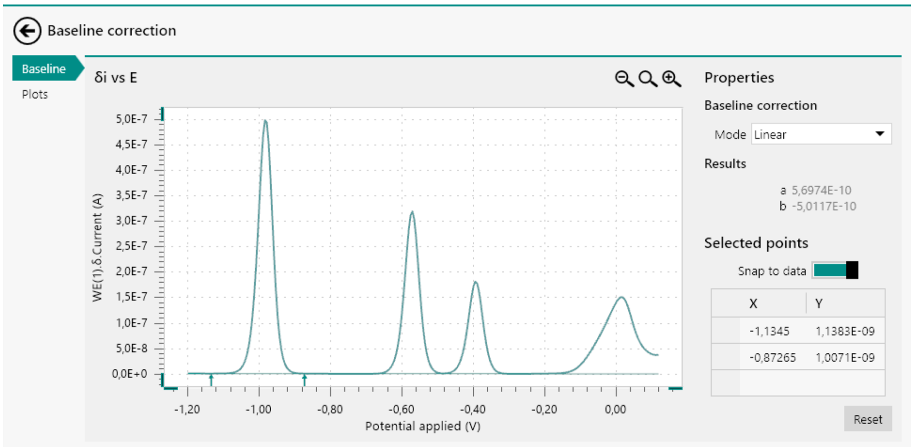


Figure 946 The baseline is drawn as soon as enough data points have been specified

NOTE

The coordinates of the selected points are added to the table on the right-hand side. The table allows data points to be modified manually or removed.

Once the baseline is defined, the residual plot is automatically created in the **Plots** frame *(see Figure 947, page 772)*.

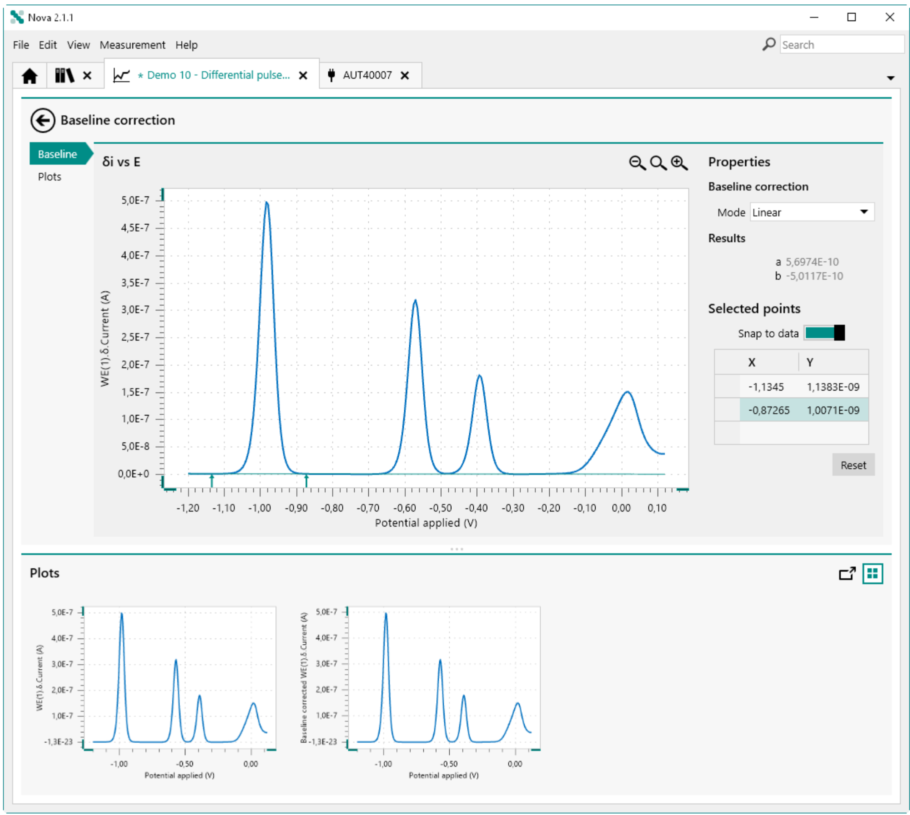


Figure 947 The residual plot is automatically created in the Plots frame when the baseline is defined

It is possible to add extra points to define the baseline by clicking additional points on the plot. Each new point added to the plot forces the baseline to be recalculated *(see Figure 948, page 772)*.

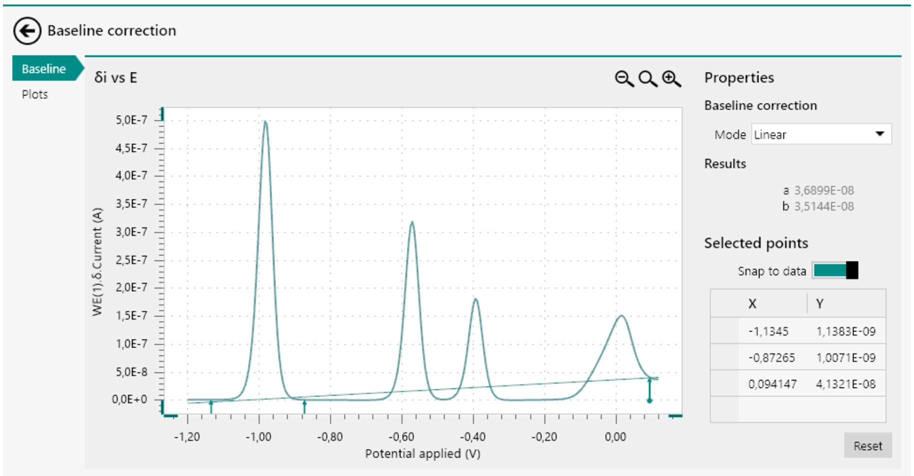


Figure 948 The baseline is update each time a point is added to the plot

Each change to the drawn baseline in turn forces the residual plot to be updated in the **Plots** frame *(see Figure 949, page 773)*.

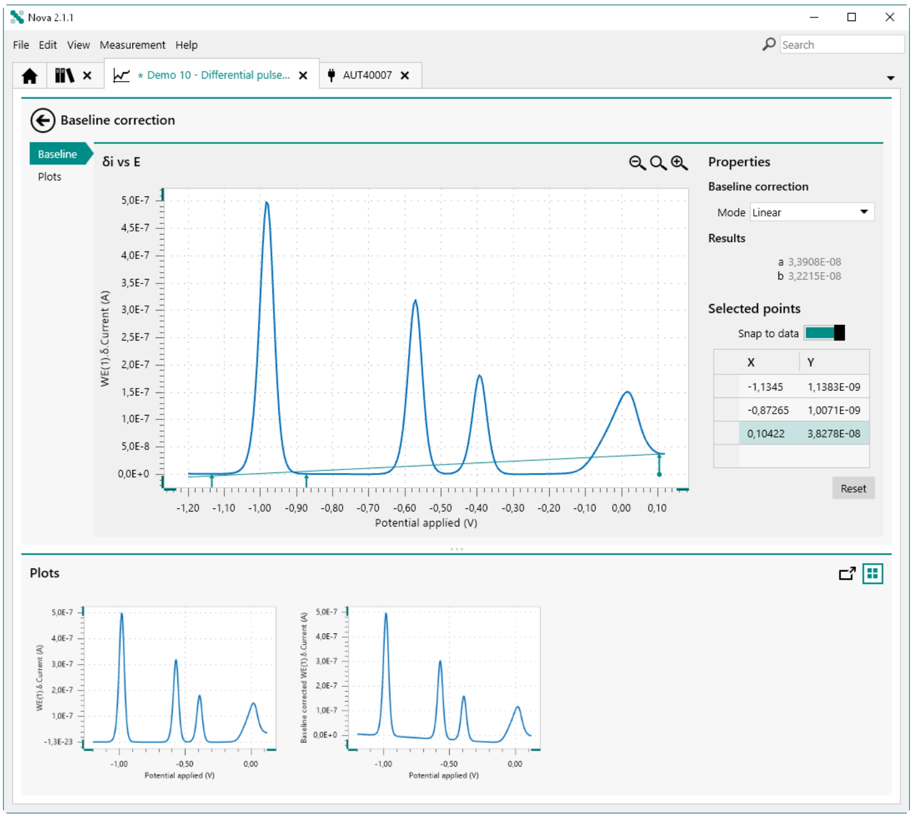


Figure 949 Changing the baseline triggers the residual plot to be updated

NOTE

Adding extra markers to a specific area of the plot increases the relative importance of that specific area of the plot in the baseline correction.

12.7.1 Zooming in/out

If needed, it is possible to use the controls located in the top right corner of the plot to zoom in () or out () or to rescale () the plot. It is also possible to use the controls provided in the **View** menu or the associated keyboard shortcuts *(see Figure 950, page 774)*.

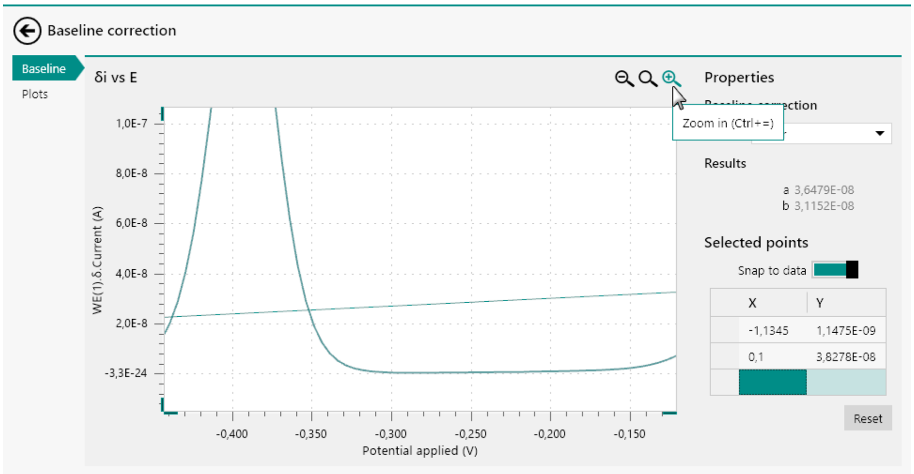


Figure 950 It is possible to zoom in or out

Clicking the 🔍 button or pressing the **[F4]** key rescales the complete plot *(see Figure 951, page 774)*.

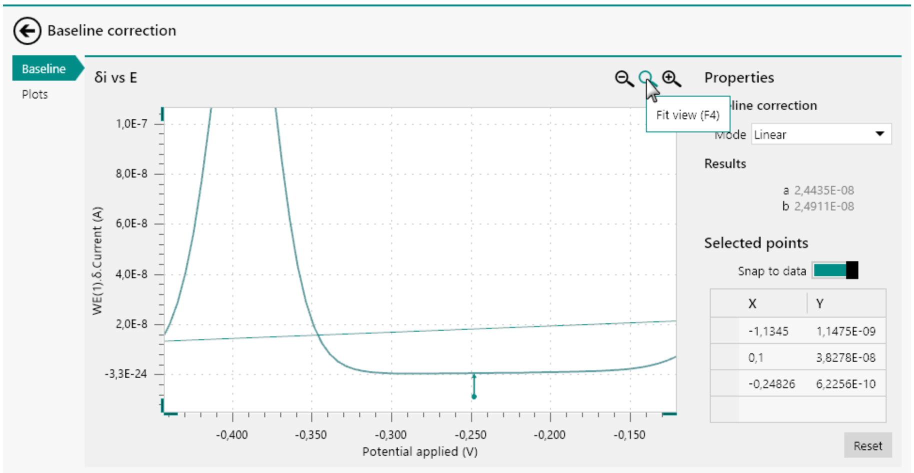


Figure 951 Rescaling the plot

NOTE

When working with a mouse fitted with a wheel, it is possible to zoom in or out using the wheel.

12.7.2 Fine tuning the baseline correction

If needed, it is possible to fine tune the location of the points using the table located in the **Properties** panel. To edit the location of one of the points, click the X or Y cell of the point to edit and click it again to edit the value *(see Figure 952, page 775)*.

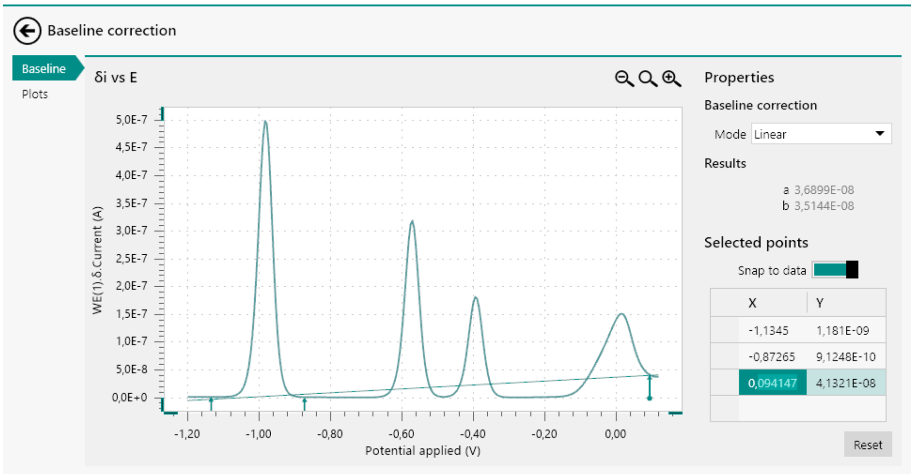


Figure 952 Editing the location of a point

Type the new value in the selected cell *(see Figure 953, page 775)*.

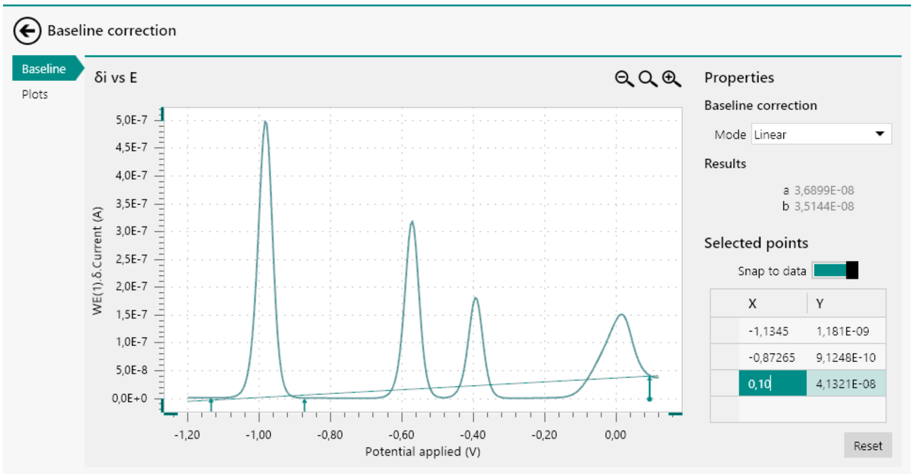


Figure 953 Fine tuning the location of the selected point

Clicking away from the cell or pressing the **[Enter]** key or **[Tab]** key will validate the new location of the point.

If needed, a point marker can be deleted. To delete a point, click the cell located at the left of the X and Y cell of the point. This will select the complete row of the table. Press the **[Delete]** key to delete this point *(see Figure 954, page 776)*.

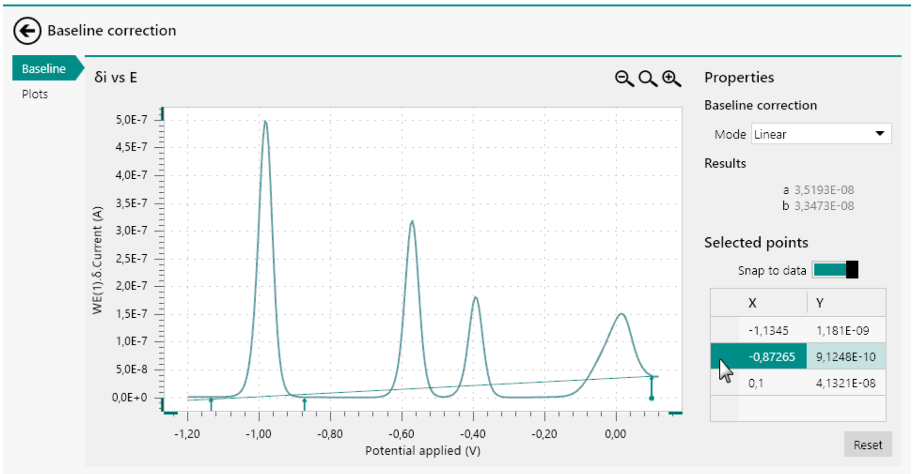


Figure 954 Selecting the point to delete

The point will be removed *(see Figure 955, page 776)*.

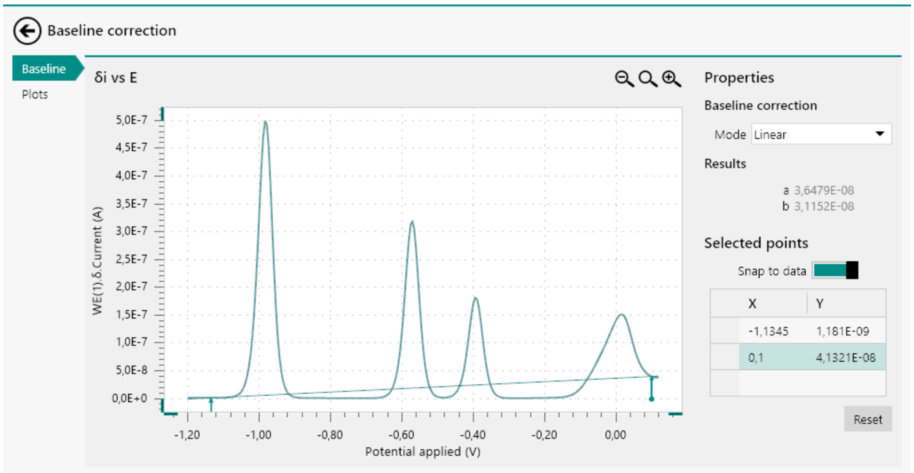


Figure 955 The selected point is deleted

12.8 Corrosion rate analysis

The **Corrosion rate analysis** command provides additional controls that can be used when the command is used to analyze data.

CAUTION

The **Corrosion rate analysis** command is intended to be used on current data (WE(1).Current) plotted against potential data (Potential applied).

To use the **Corrosion rate analysis** command, this command can be added to the procedure as a command, using the drag and drop method, or by using the button. In the latter case, a pop-out menu is displayed, providing a list of commands and possible plots on which these command can be applied *(see Figure 956, page 777)*.

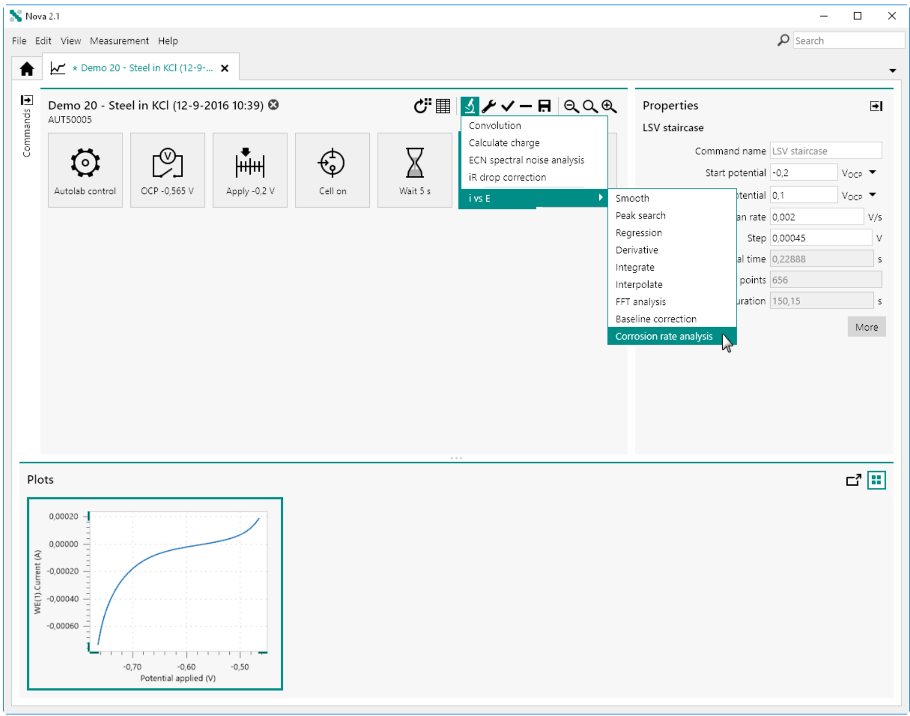


Figure 956 Adding a Corrosion rate analysis command to the linear polarization data

The **Corrosion rate analysis** command is added to the procedure editor. Clicking the command shows the properties and the results in the dedicated panel on the right hand side *(see Figure 957, page 778)*.

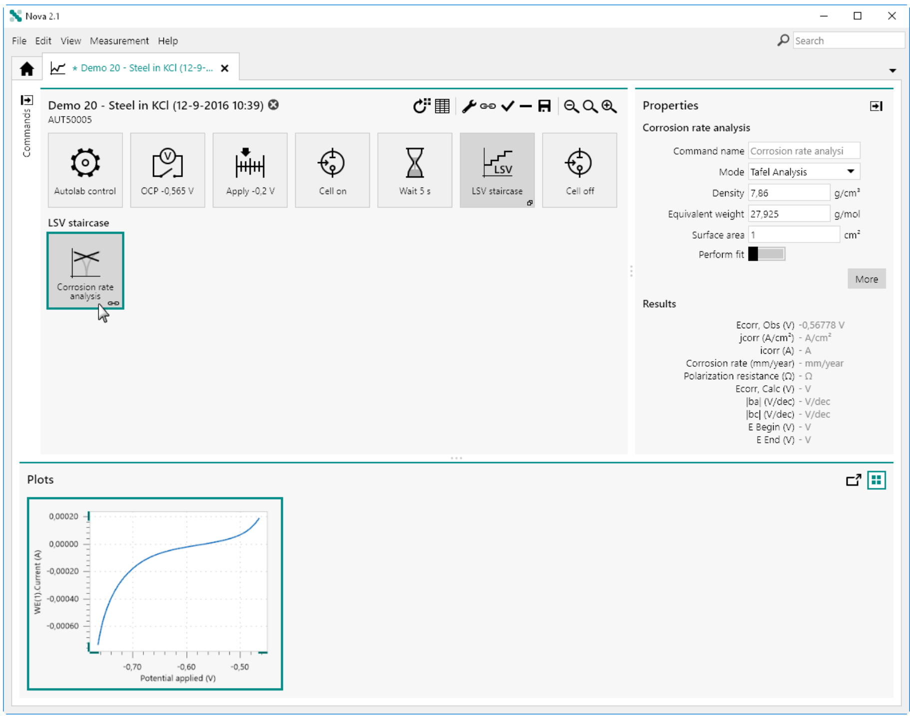


Figure 957 The Corrosion rate analysis command is added to the procedure

NOTE

For more information on the properties of the **Corrosion rate analysis** command, please refer to *Chapter 7.8.14*.

Clicking the More button opens a new screen in which the additional controls of the **Corrosion rate analysis** command are shown for the scope of data analysis. The plot on the left hand side shows the source data, plotted on a logarithmic scale. The properties of the **Corrosion rate analysis** command are all set to their default values *(see Figure 958, page 779)*.

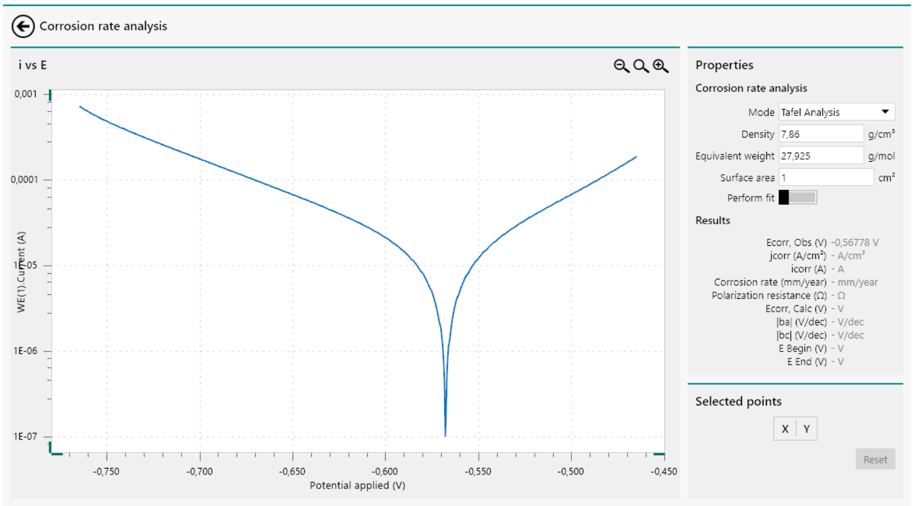


Figure 958 The additional controls of the Corrosion rate analysis command

Since the **Corrosion rate** command has two different modes, each mode provides dedicated controls. The *Mode* drop-down list can be used to change the mode of the command.

12.8.1 Tafel Analysis

In *Tafel Analysis* mode, it is necessary to define two points on the *anodic* part of the Tafel plot and two points on the *cathodic* part of the Tafel plot.

To define a point, click on the plot. The software will automatically select the closest point of the measured data. When two points are defined, the anodic Tafel slope will be drawn on the plot *(see Figure 959, page 780)*.

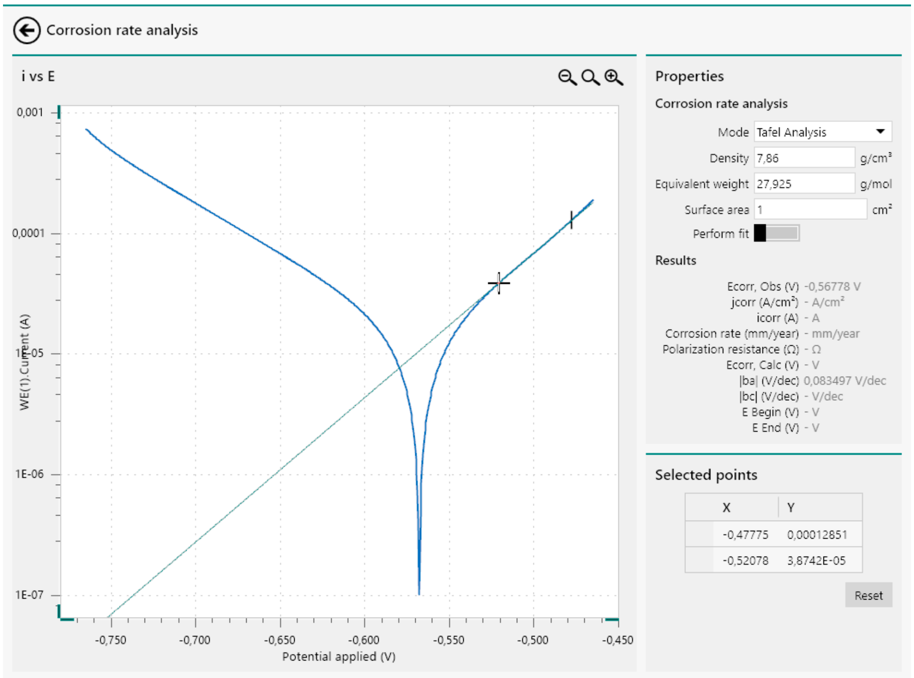


Figure 959 Defining the points for the anodic Tafel slope

The same can be done for the cathodic branch. When the two points are defined, the cathodic Tafel slope is plotted and the intercept of both lines is used to determine the corrosion potential, the exchange current and current density, the polarization resistance and the corrosion rate *(see Figure 960, page 780)*.

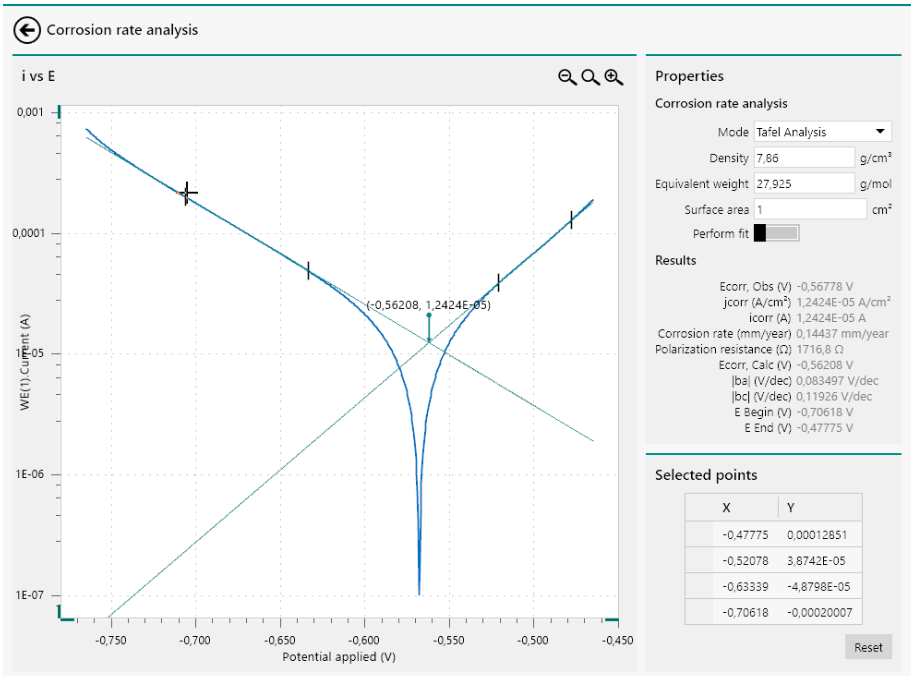


Figure 960 The two slopes are used to determine the corrosion rate

Depending on the position of the **Perform fit** toggle, the **Corrosion rate analysis** command will either:

1. **Perform fit off:** the command will perform the calculations of the command based on the location of the intercept. This will lead to an approximation of the corrosion data *(see Figure 960, page 780)*.
2. **Perform fit on:** the intercept will be used as a starting point for the fitting of the data using the Butler-Volmer equation. The complete curve will be fitted using this equation and the corrosion data will be determined by the results of the fit. This leads to a more accurate determination of the corrosion date, as shown in *Figure 961*.

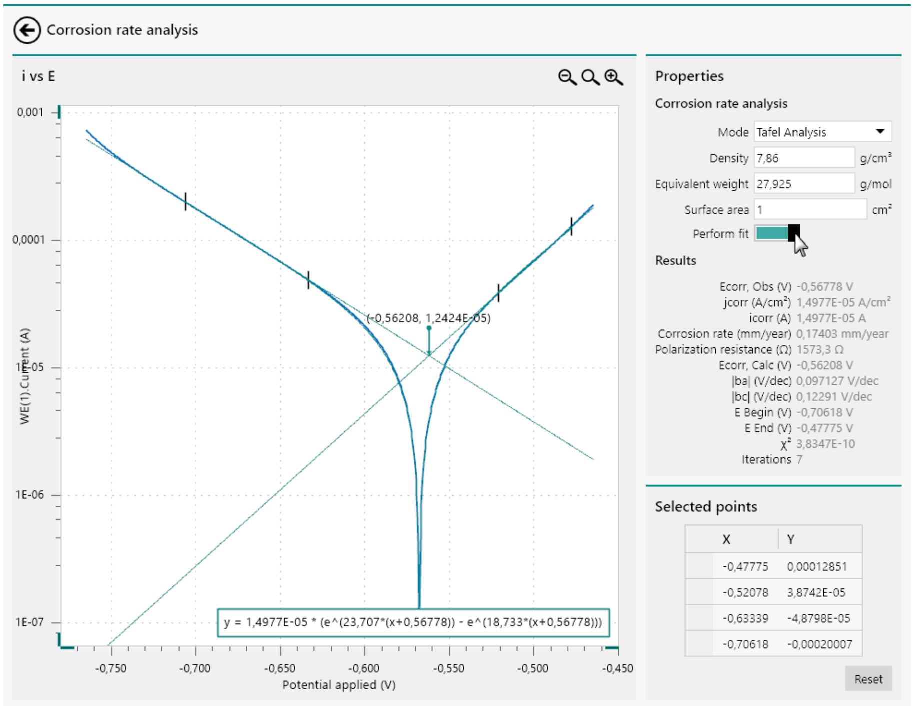


Figure 961 The measured data is fitted with the Butler-Volmer equation

Once the four points required by the **Corrosion rate analysis** command have been specified, it is possible to manually adjust the location of these points by clicking a vertical line defining the location of a point and dragging the line left or right *(see Figure 962, page 782)*.

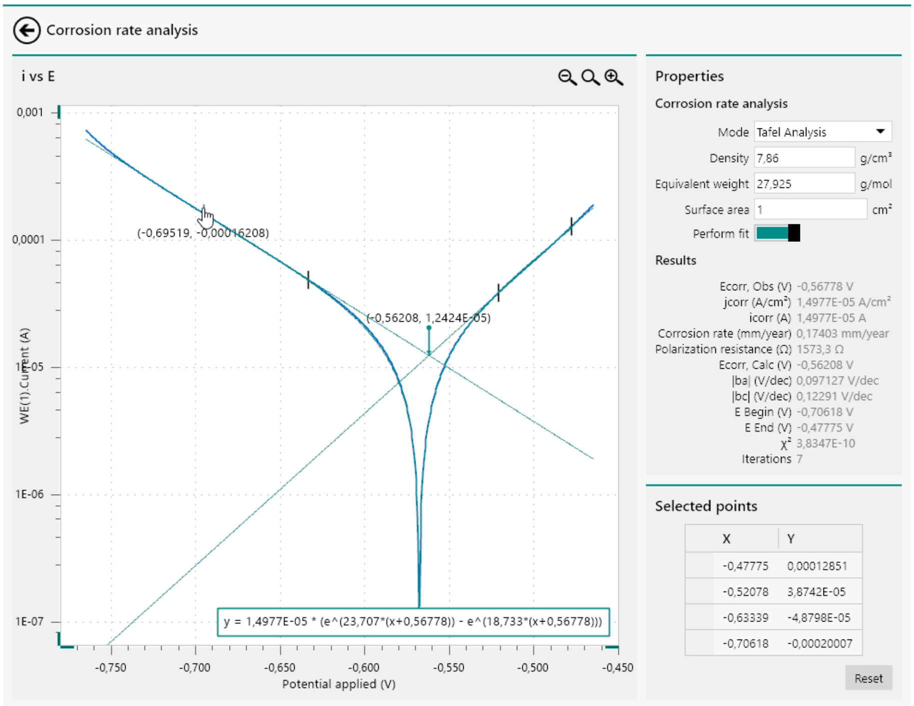


Figure 962   Adjusting the points of the Corrosion rate analysis command

As soon as the point is relocated, the calculation of the **Corrosion rate analysis** command will be updated and the new results will be displayed in the Results sub-panel *(see Figure 963, page 783)*.

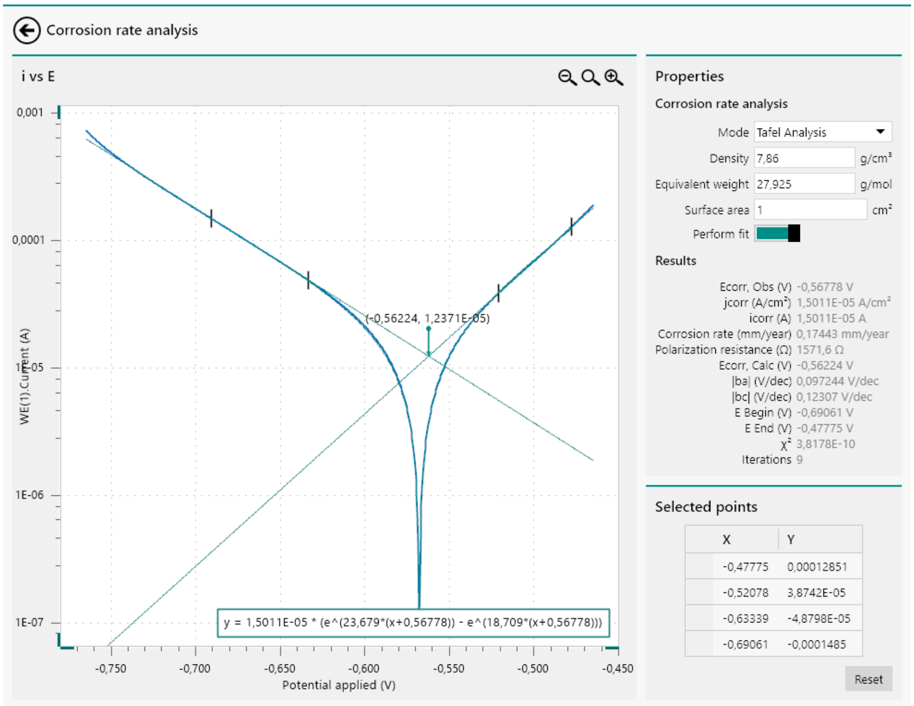


Figure 963 The calculation is refreshed as soon as one of the points is modified

NOTE

It is also possible to finetune the location of the points used in the **Corrosion rate analysis** command by using the **Selected points** table, located below the **Properties** panel.

Finally, clicking the Reset button resets all the properties of the **Corrosion rate analysis** command back to the default values and clears the selected points.

12.8.2 Polarization Resistance

In *Polarization Resistance* mode, no inputs are required. The calculations are automatically carried out with the specified settings. The potential range in which the analysis is carried out is shown in the plot *(see Figure 964, page 784)*.

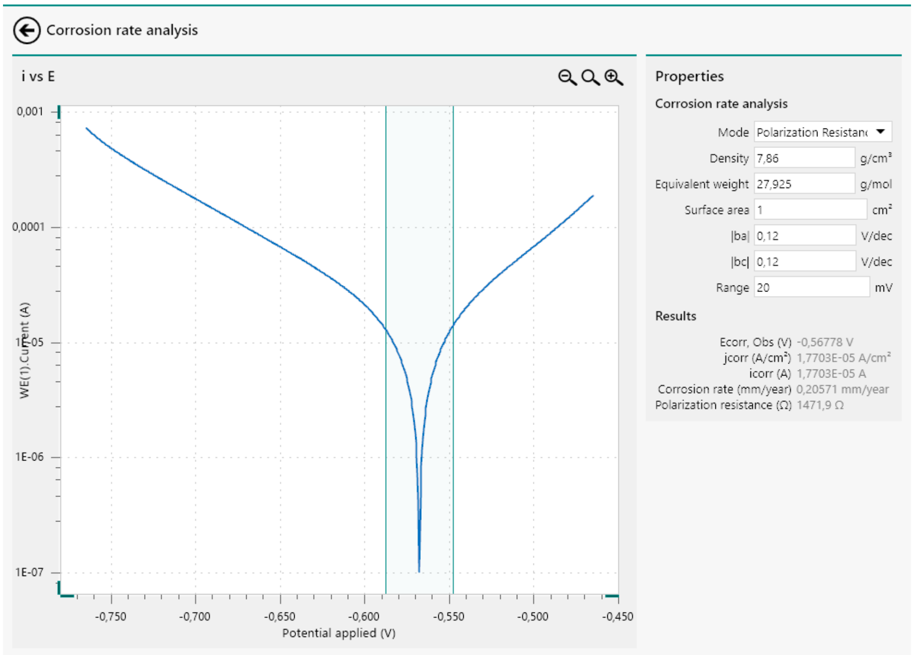


Figure 964 The Polarization Resistance analysis is carried in out in the highlighted range

If needed, the **Range** value can be adjusted. When the value is modified, the calculation is updated and the range is adjusted on the plot *(see Figure 965, page 784)*.

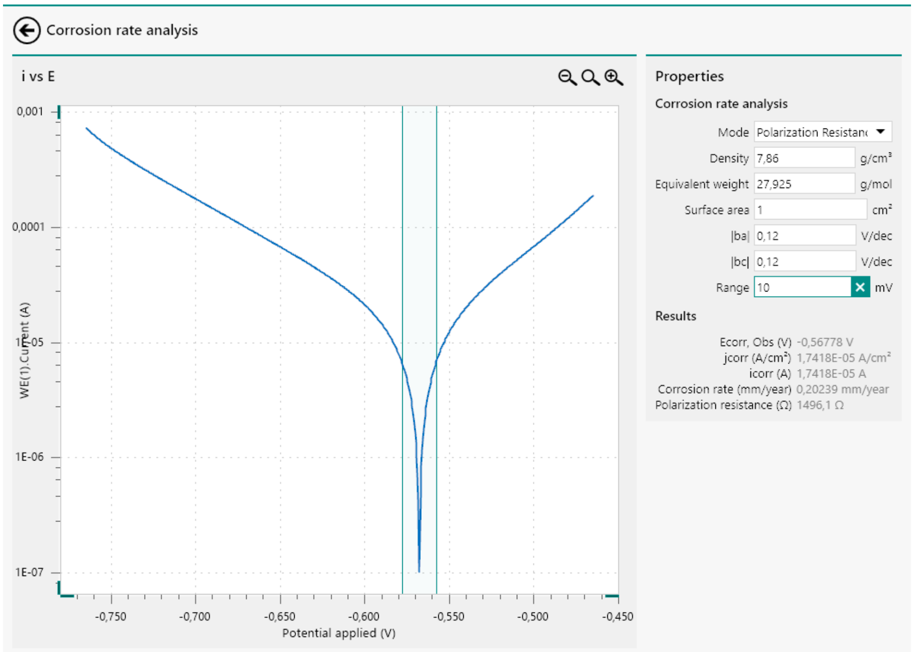


Figure 965 The plot is updated when the Range value is change

Changing the value of any other property used by the command will also force the calculation to update.

12.9 Electrochemical circle fit

The **Electrochemical circle fit** command provides additional controls that can be used when the command is used to analyze data.

CAUTION

The **Electrochemical circle fit** command is intended to be used on impedance spectroscopy data.

To use the **Electrochemical circle fit** command, this command can be added to the procedure as a command, using the drag and drop method, or by using the button. In the latter case, a pop-out menu is displayed, providing a list of commands and possible plots on which these command can be applied *(see Figure 966, page 785)*.

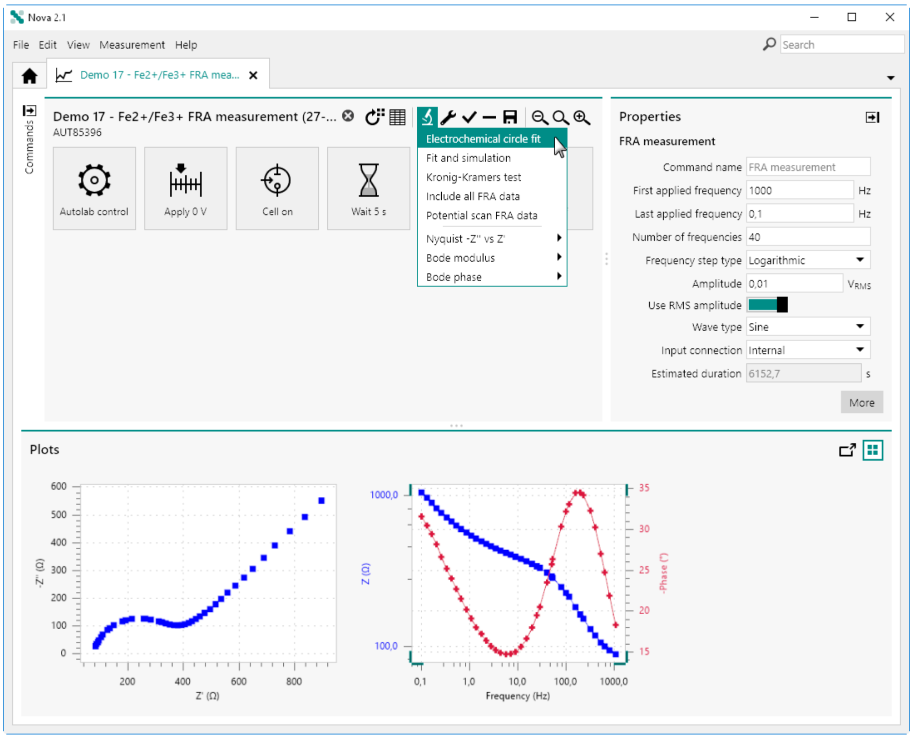


Figure 966 Adding a Electrochemical circle fit command to impedance data

The **Electrochemical circle fit** command is added to the procedure editor. Clicking the command shows the properties in the dedicated panel on the right hand side *(see Figure 967, page 786)*.

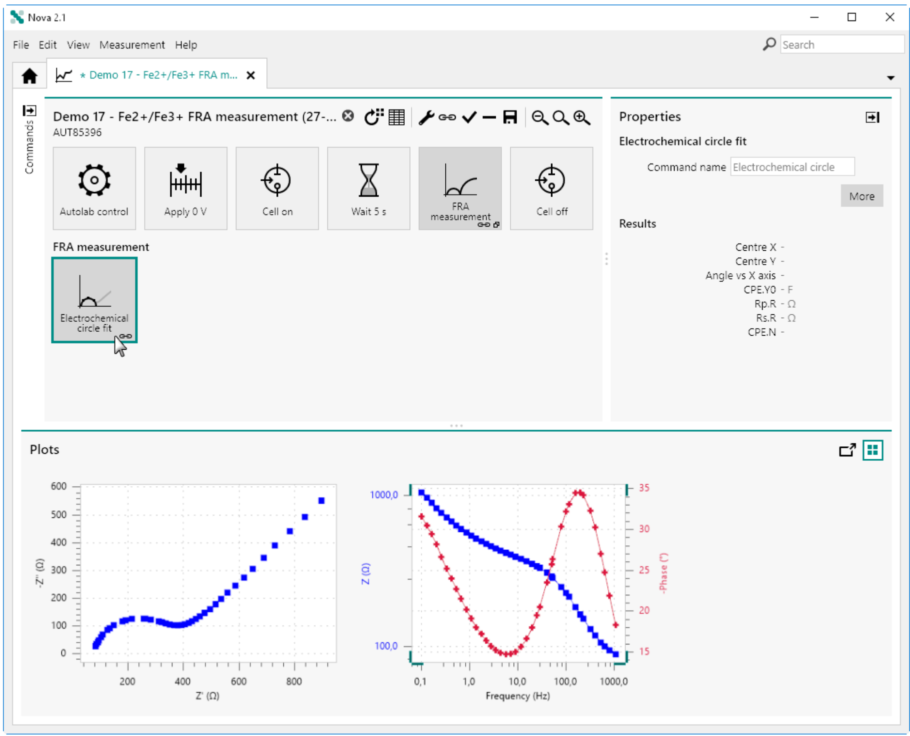


Figure 967 The Electrochemical circle fit command is added to the procedure

NOTE

For more information on the properties of the **Electrochemical circle fit** command, please refer to *Chapter 7.9.1*.

Clicking the More button opens a new screen in which the additional controls of the **Electrochemical circle fit** command are shown for the scope of data analysis. The plot on the left hand side shows the source data, presented in a Nyquist plot. The results of the **Electrochemical circle fit** command are shown in the panel on the right hand side *(see Figure 967, page 786)*.

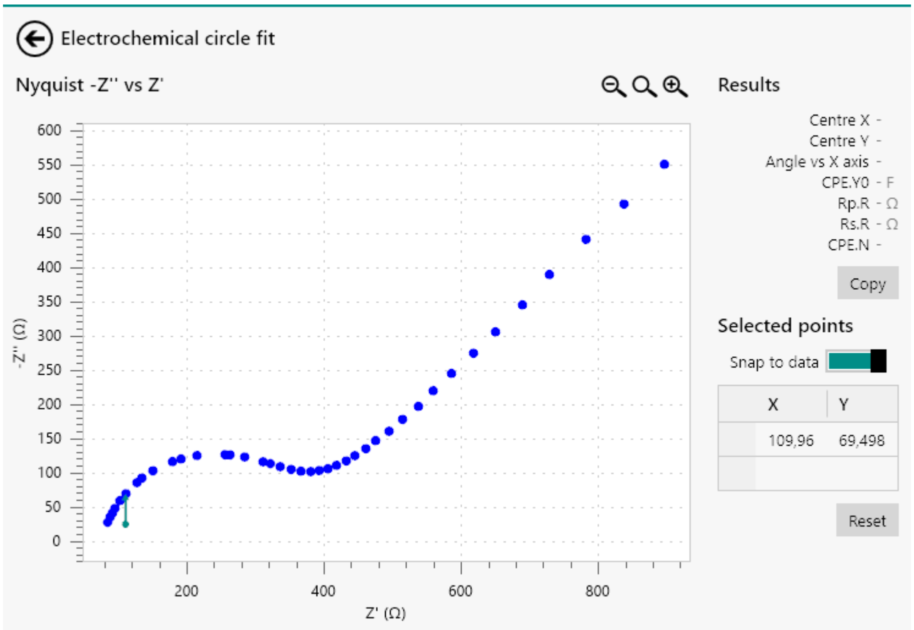


Figure 968 The additional controls of the Electrochemical circle fit command

NOTE

The coordinates of the selected points are added to the table on the right-hand side. The table allows data points to be modified manually or removed.

To use the **Electrochemical circle fit** command, it is necessary to define three or more points along a visible semi-circle in the Nyquist plot to draw a half-circle and determine the properties of the apparent time constant of this part of the plot.

To define a point, click on the plot. The software will automatically select the closest point of the measured data (if the *Snap to data* option is on) or the point will be located where the mouse is clicked (in the *Snap to data* option is off). When three points are defined, the semi-circle will be drawn on the plot and the results will be updated in the panel on the right-hand side *(see Figure 969, page 788)*.

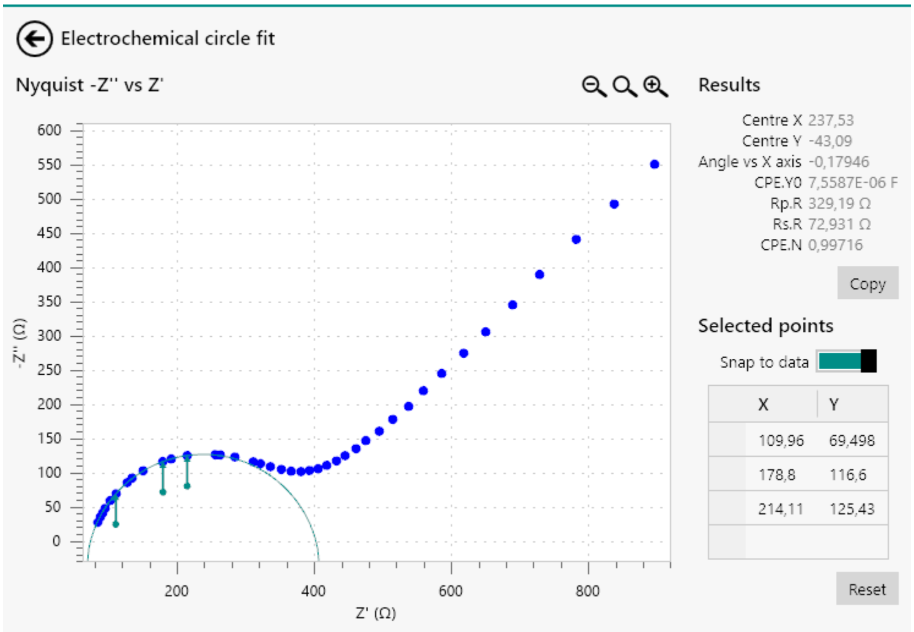


Figure 969 The semi-circle is drawn when three or more points are selected

It is possible to add extra points to the plot. The calculation will automatically be refreshed whenever a point is added to the plot.

NOTE

Adding extra points to a specific area of the plot increases the relative importance of that specific area of the plot in the electrochemical circle fit.

NOTE

It is also possible to finetune the location of the points used in the **Electrochemical circle fit** command by using the **Selected points** table, located below the **Results** panel.

Finally, clicking the Reset button resets all the properties of the **Electrochemical circle fit** command back to the default values and clears the selected points.

12.9.1 Zooming in/out

If needed, it is possible to use the controls located in the top right corner of the plot to zoom in ( ) or out ( ) or to rescale ( ) the plot. It is also possible to use the controls provided in the **View** menu or the associated keyboard shortcuts *(see Figure 970, page 789)*.

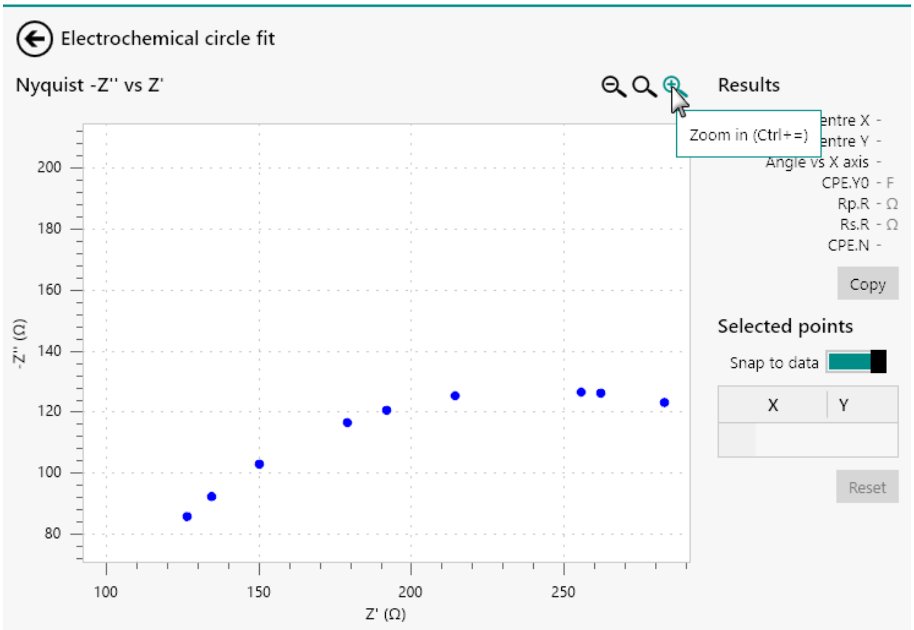


Figure 970 It is possible to zoom in or out

Clicking the button or pressing the **[F4]** key rescales the complete plot *(see Figure 971, page 789)*.

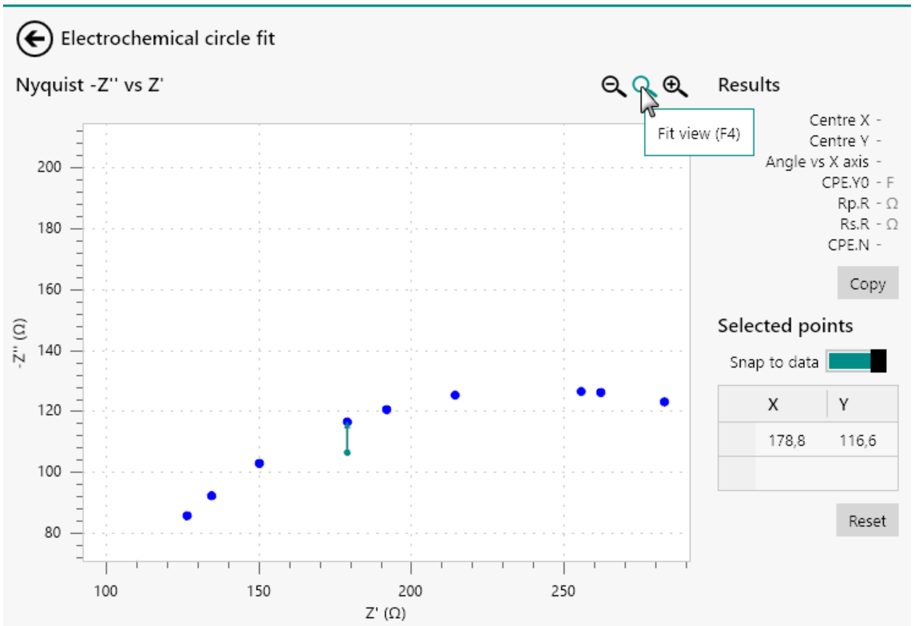


Figure 971 Rescaling the plot

NOTE

When working with a mouse fitted with a wheel, it is possible to zoom in or out using the wheel.

12.9.2 Fine tuning the baseline correction

If needed, it is possible to fine tune the location of the points using the table located in the **Properties** panel. To edit the location of one of the points, click the X or Y cell of the point to edit and click it again to edit the value *(see Figure 972, page 790)*.

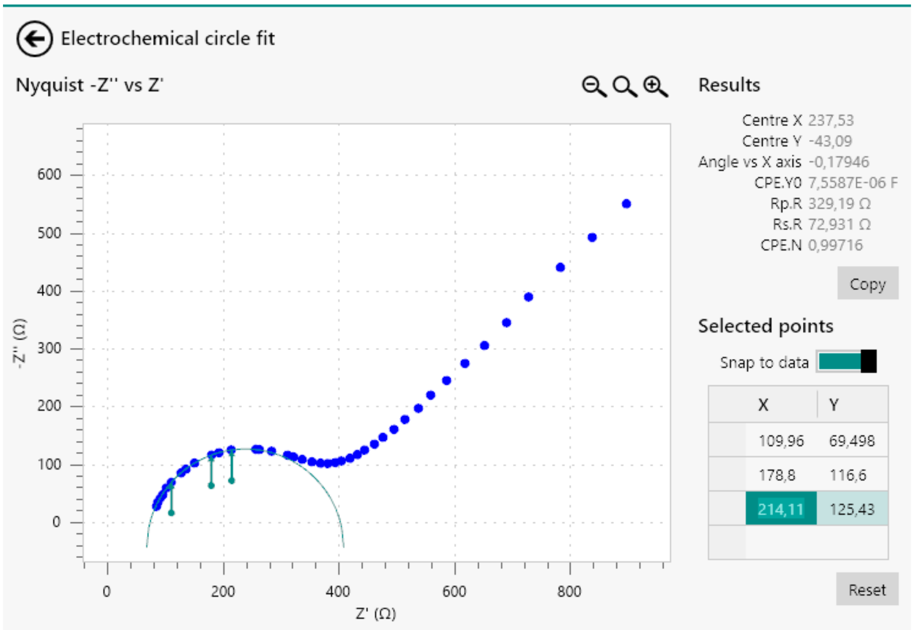


Figure 972 Editing the location of a point

Type the new value in the selected cell *(see Figure 973, page 791)*.

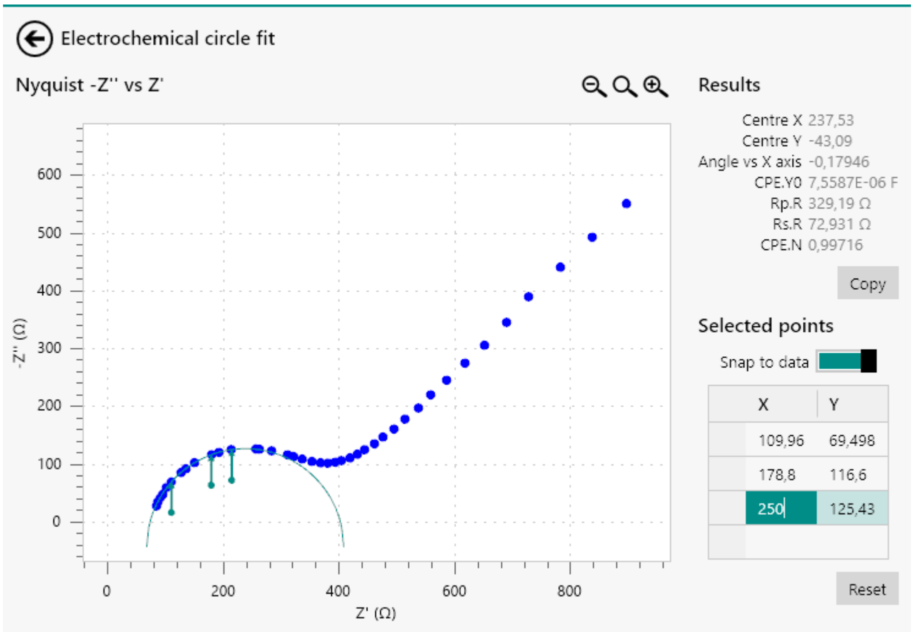


Figure 973 Fine tuning the location of the selected point

Clicking away from the cell or pressing the **[Enter]** key or **[Tab]** key will validate the new location of the point.

If needed, a point marker can be deleted. To delete a point, click the cell located at the left of the X and Y cell of the point. This will select the complete row of the table. Press the **[Delete]** key to delete this point *(see Figure 974, page 791)*.

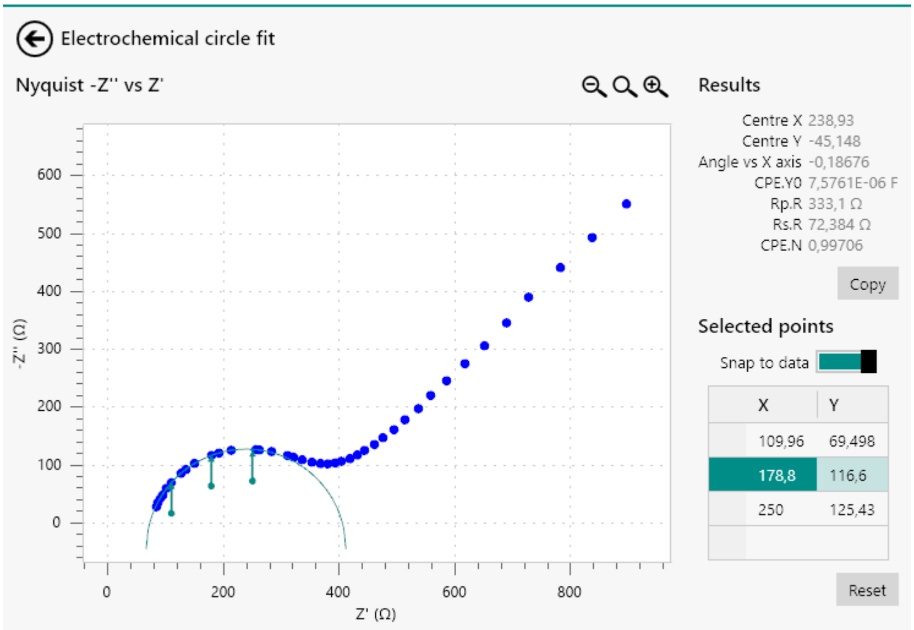


Figure 974 Selecting the point to delete

The point will be removed *(see Figure 975, page 792)*.

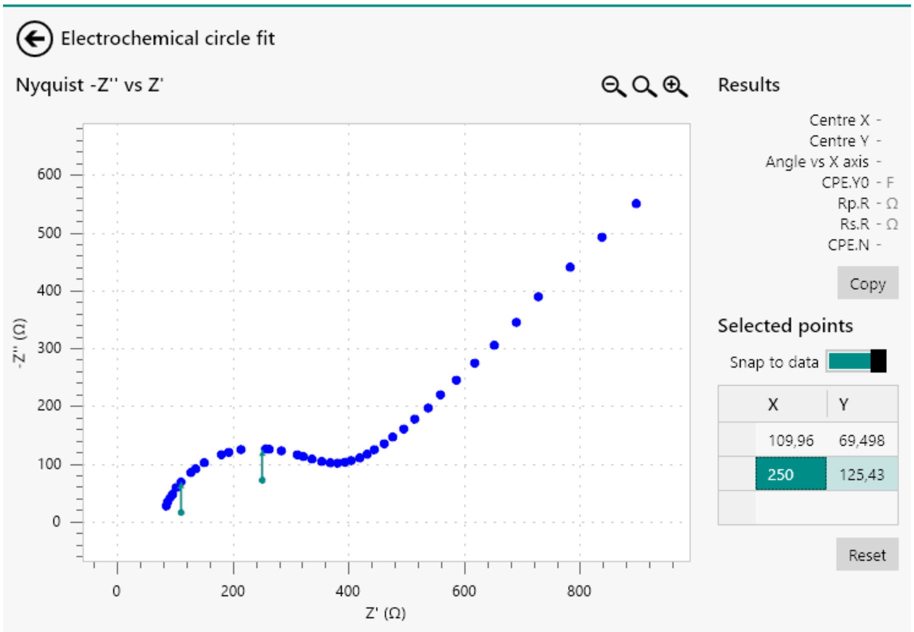


Figure 975 The selected point is deleted

12.9.3 Copy as equivalent circuit

It is possible to copy the results from the **Electrochemical circle fit** analysis tool to the clipboard as an equivalent circuit. This circuit can then be used in the **Fit and Simulation** command or analysis tool *(see Chapter 12.10, page 793)*.

To copy the results, click the Copy button located in the **Results** panel *(see Figure 976, page 792)*.

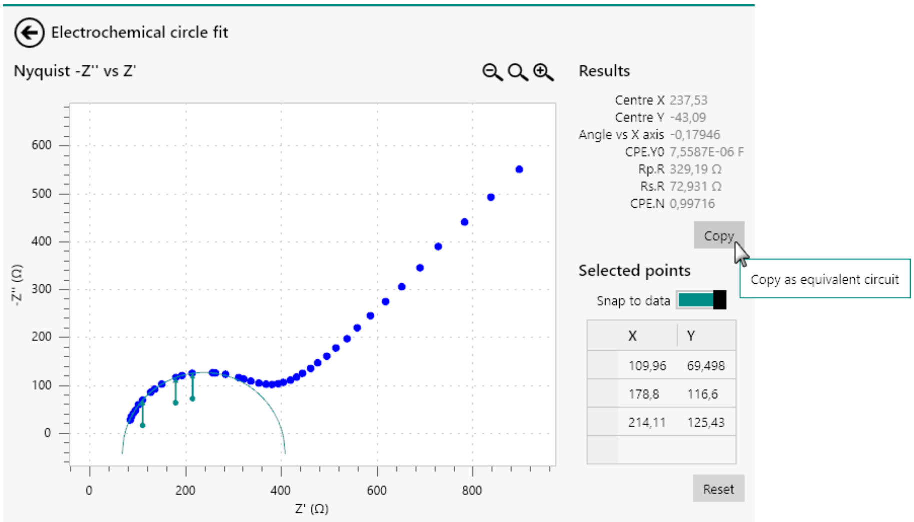


Figure 976 Copy the results as an equivalent circuit

The results of the **Electrochemical circle fit** analysis tool will be copied to the clipboard as a R(RQ) equivalent circuit.

12.10 Fit and simulation

The **Fit and simulation** command provides additional controls that can be used when the command is used to analyze data.

CAUTION

The **Fit and simulation** command is intended to be used on impedance spectroscopy data.

To use the **Fit and simulation** command, this command can be added to the procedure as a command, using the drag and drop method, or by using the button. In the latter case, a pop-out menu is displayed, providing a list of commands and possible plots on which these command can be applied *(see Figure 977, page 793)*.

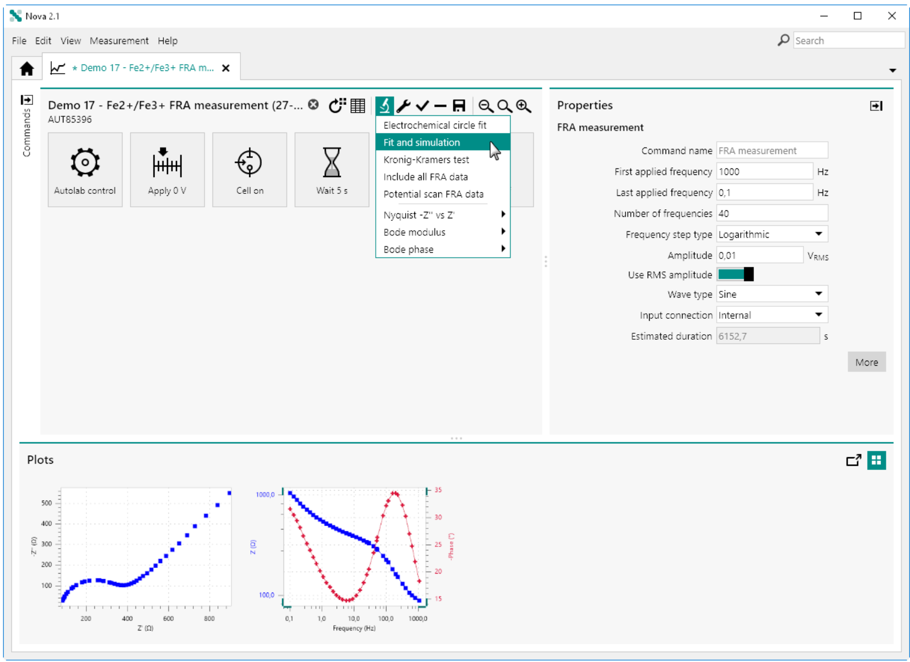


Figure 977 Adding a Fit and simulation command to impedance data

The **Fit and simulation** command is added to the procedure editor. Clicking the command shows the properties in the dedicated panel on the right hand side *(see Figure 978, page 794)*.

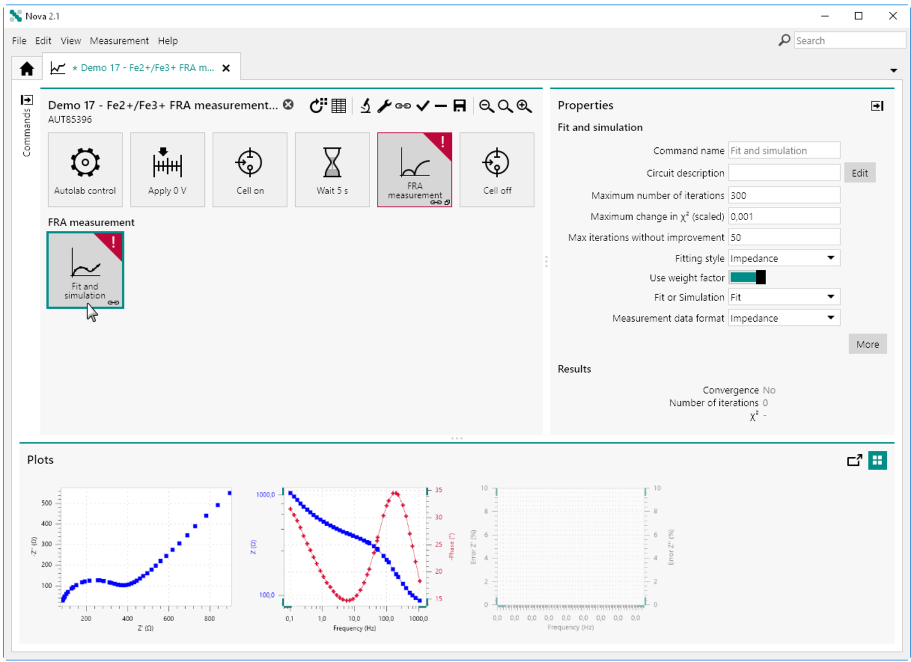


Figure 978 The Fit and simulation command is added to the procedure

NOTE

For more information on the properties of the **Fit and simulation** command, please refer to *Chapter 7.9.2*.

For data analysis purposes, it is possible to use the **Fit and simulation** tool in two different ways:

- Direct fitting or simulation *(see Chapter 12.10.1, page 794)*
- Fitting or simulating using the dedicated editor *(see Chapter 12.10.2, page 796)*

12.10.1 Direct fitting or simulation

It is possible to use the **Fit and simulation** tool to directly fit or simulate the impedance data. Using this method, it is only necessary to specify the equivalent circuit to use, as a CDC string in the **Circuit description** field of the **Properties** panel *(see Figure 979, page 795)*.

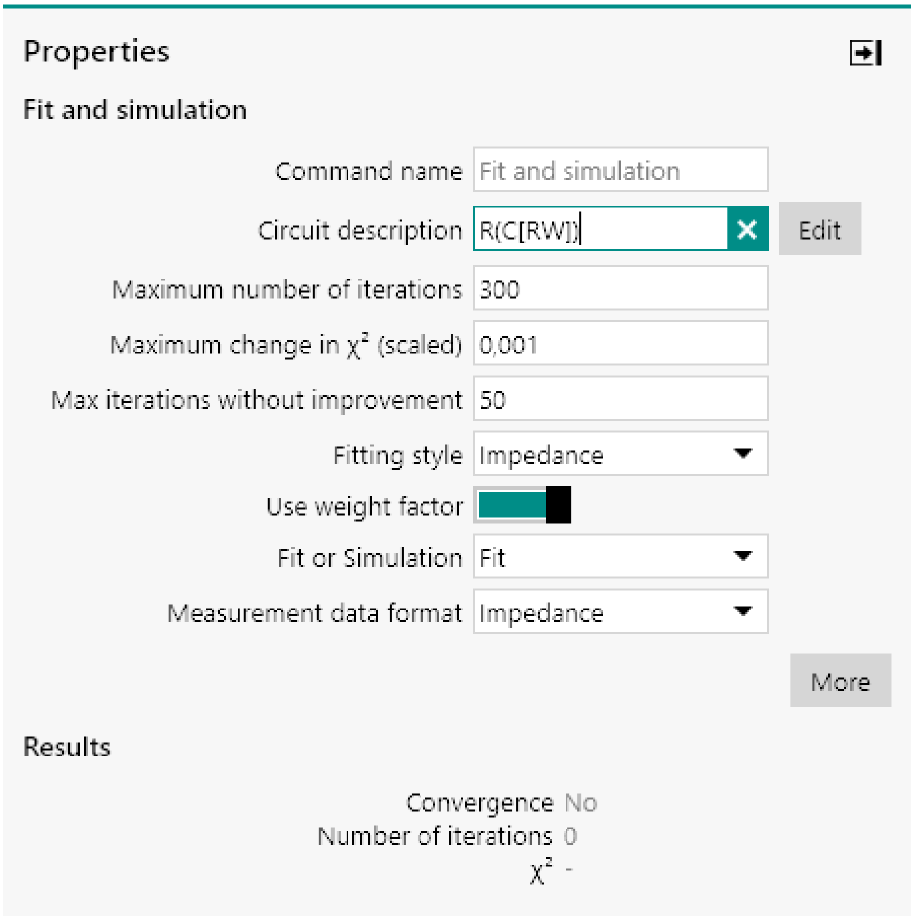


Figure 979 Typing a CDC string in Properties panel

When the string is validated, by pressing the **[Enter]** key or by unselecting the input field, the fitting or simulation will start immediately, using the default values for all the circuit elements and using all the other properties specified in the **Properties** panel *(see Figure 980, page 796)*.

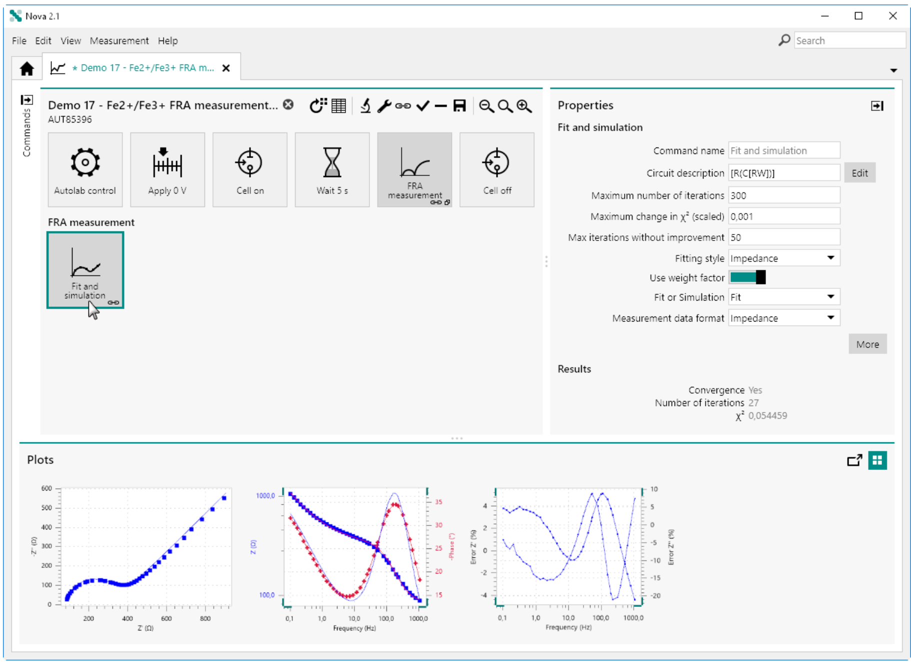


Figure 980 *The data is fitted or simulated using the specified properties and the default element values*

The fitting or the simulation of the data is automatically updated whenever one of the properties provided in the **Properties** panel is modified.

NOTE

More information on the CDC format of the equivalent circuits can be found in *Chapter 7.9.2.3*.

12.10.2 Fitting or simulation using the dedicated editor

The direct fitting method provided by the **Fit and simulation** command uses the default values for the circuit element. For a more customized analysis of the data, it is possible to used to the dedicated editor instead. To use the dedicated editor, click the

button next to the **Circuit description** field of the **Properties** panel *(see Figure 981, page 797)*.

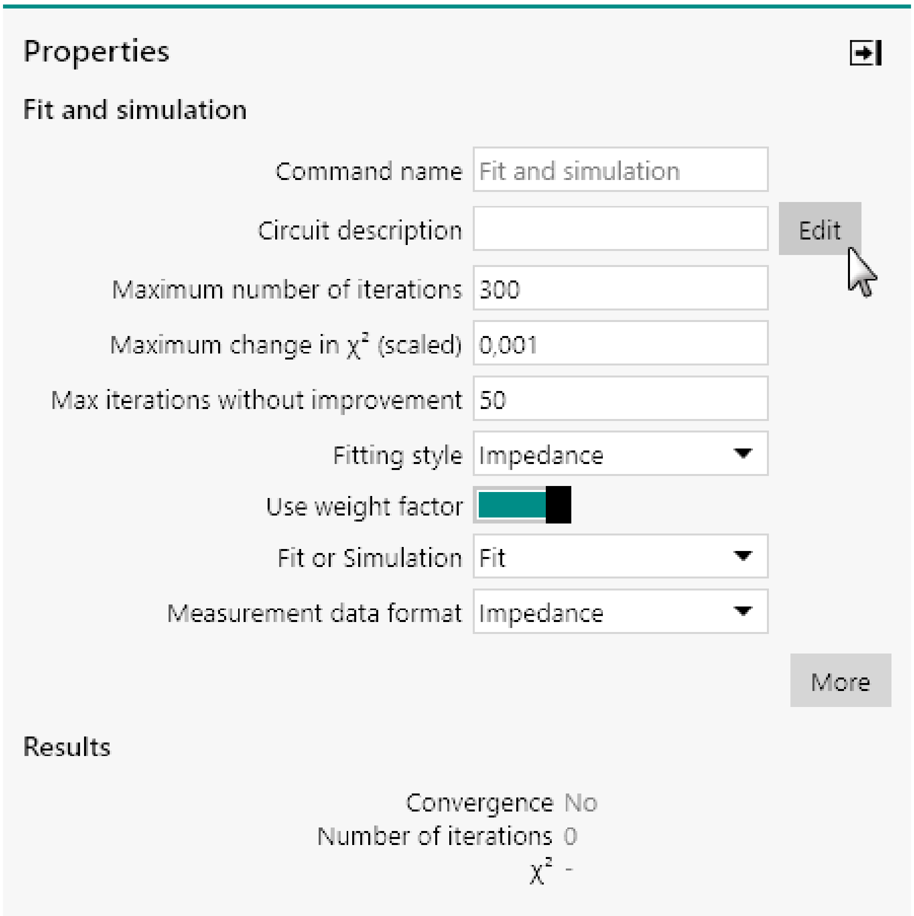


Figure 981 Opening the dedicated editor

The dedicated **Equivalent Circuit Editor** will be displayed. This editor provides the means to draw the equivalent circuit using the supported element and to specify the properties of each element *(see Figure 982, page 798)*.

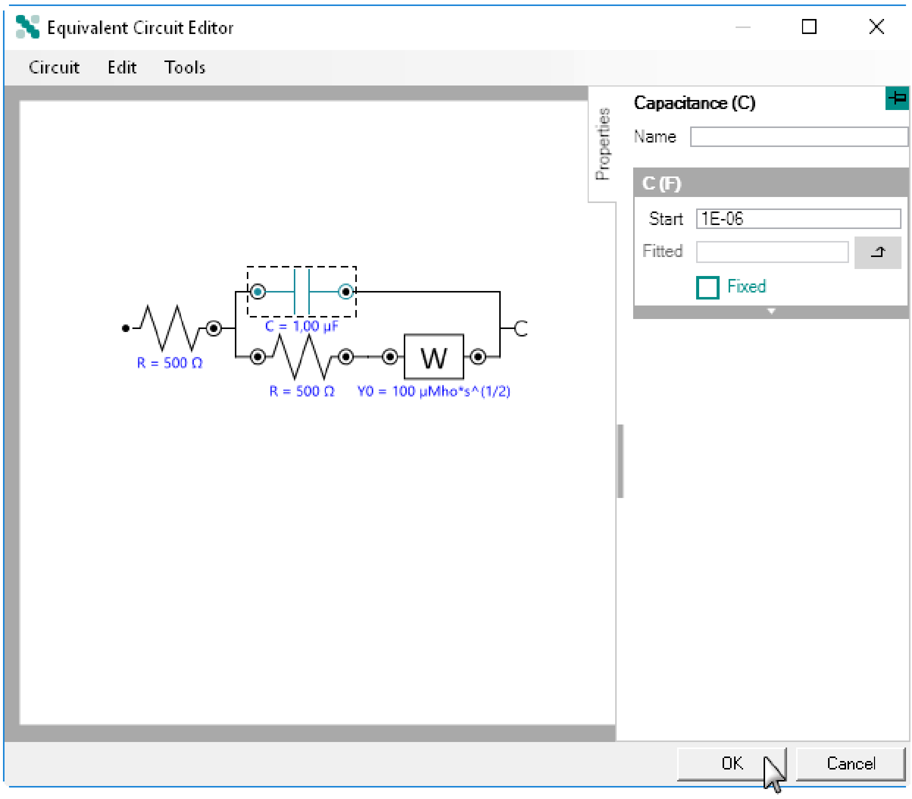


Figure 982 The equivalent circuit can be specified in the dedicated editor

NOTE

More information on the use of the **Equivalent Circuit Editor** can be found in *Chapter 7.9.2*.

When all of the properties have been defined, it is possible to run the calculation in two different ways:

- **By closing the editor:** clicking the OK button in the **Equivalent Circuit Editor** window closes the editor and triggers the calculation to run using the specified properties. The fitted or simulated data will be plotted *(see Figure 983, page 799)*.

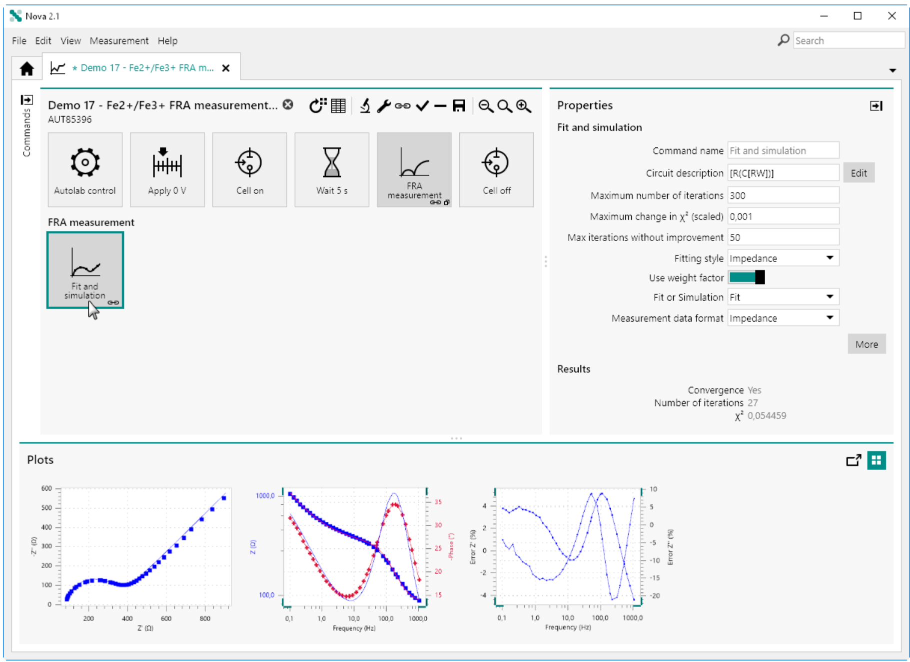


Figure 983 The data is fitted or simulated

- **By using the Tools menu in the Equivalent Circuit Editor:** the Tools menu provides the possibility to *Run Fit and simulation* (**[F5]** shortcut key) or *Resume Fit and simulation* (**[F9]** shortcut key), as shown in *Figure 984*.

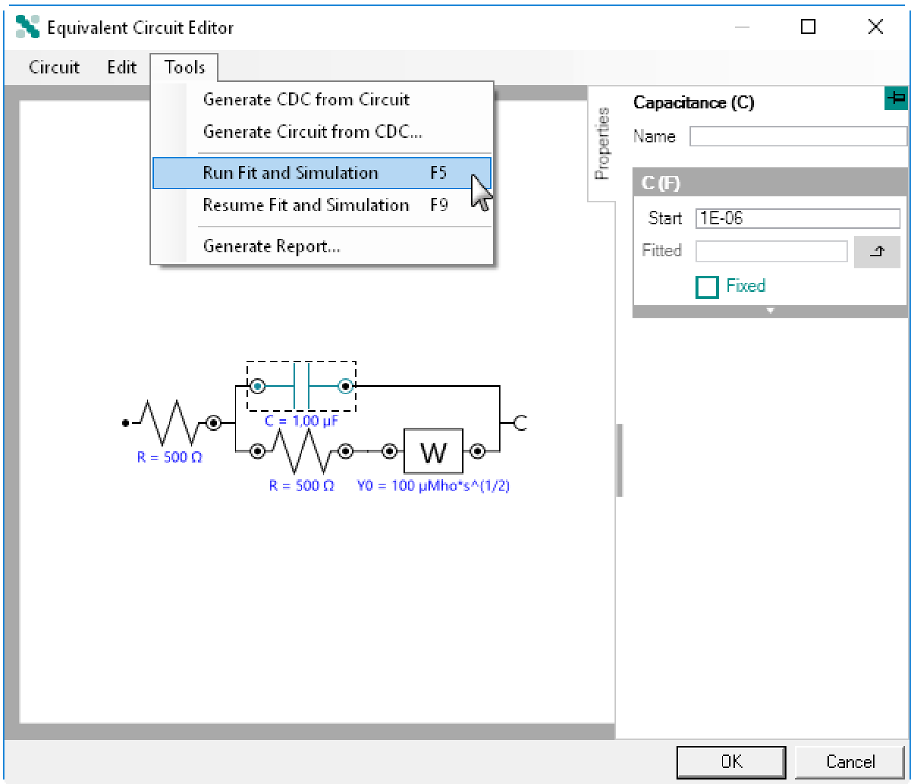


Figure 984 Running the calculation within the Equivalent Circuit Editor

While the calculation is running, a progress dialog will be shown *(see Figure 985, page 801)*.

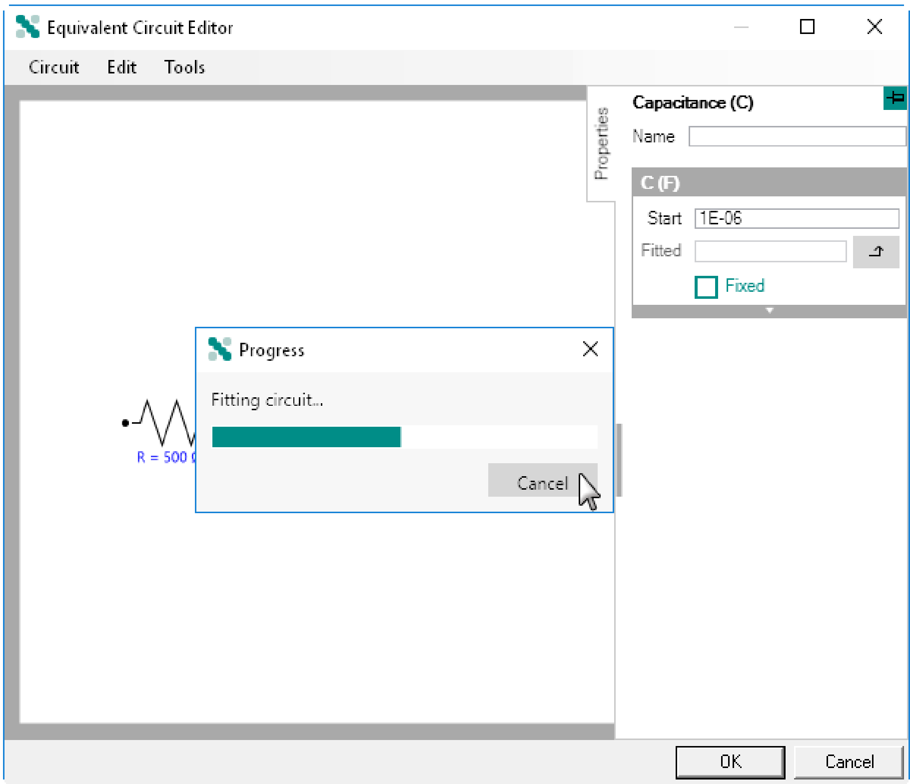


Figure 985 A progress dialog is shown during the calculation

If needed, the calculation can be stopped by clicking the Cancel button.

At the end of the calculation, if the command is used to fit the data, the fitted values are shown for each element in the **Equivalent Circuit Editor** *(see Figure 986, page 802)*.

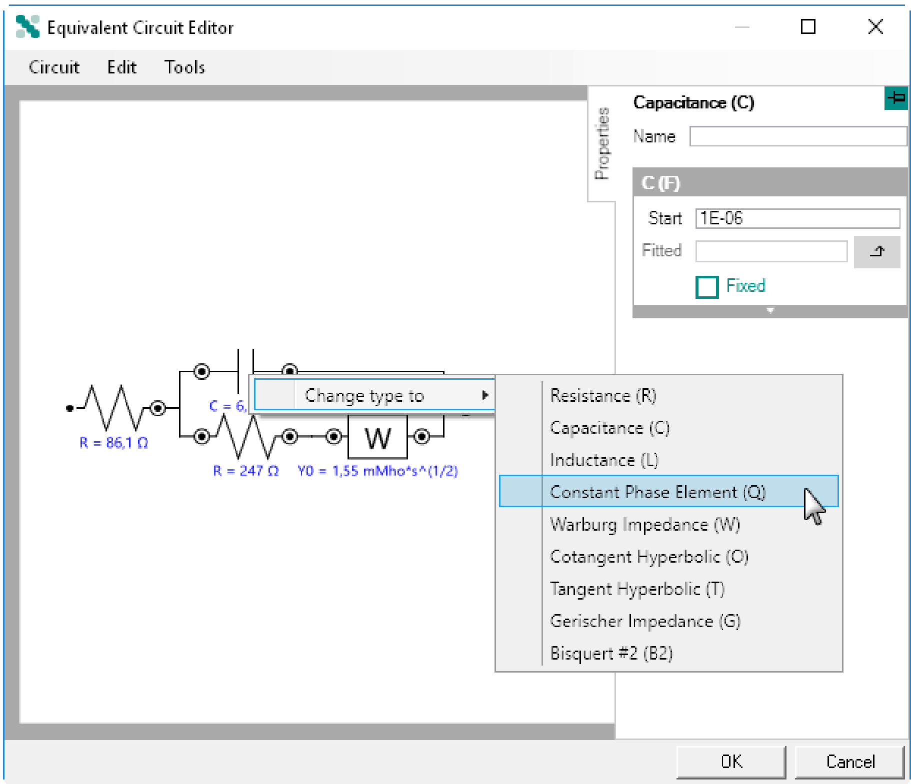


Figure 986 After the calculation, the calculated values are shown in the editor

If needed, the circuit can be modified, as shown in *Figure 986*. At any time, it is possible to restart the calculation, using the **Tools** menu or the **[F5]** shortcut key or resume the calculation, using the same menu or the **[F9]** shortcut key *(see Figure 987, page 803)*.

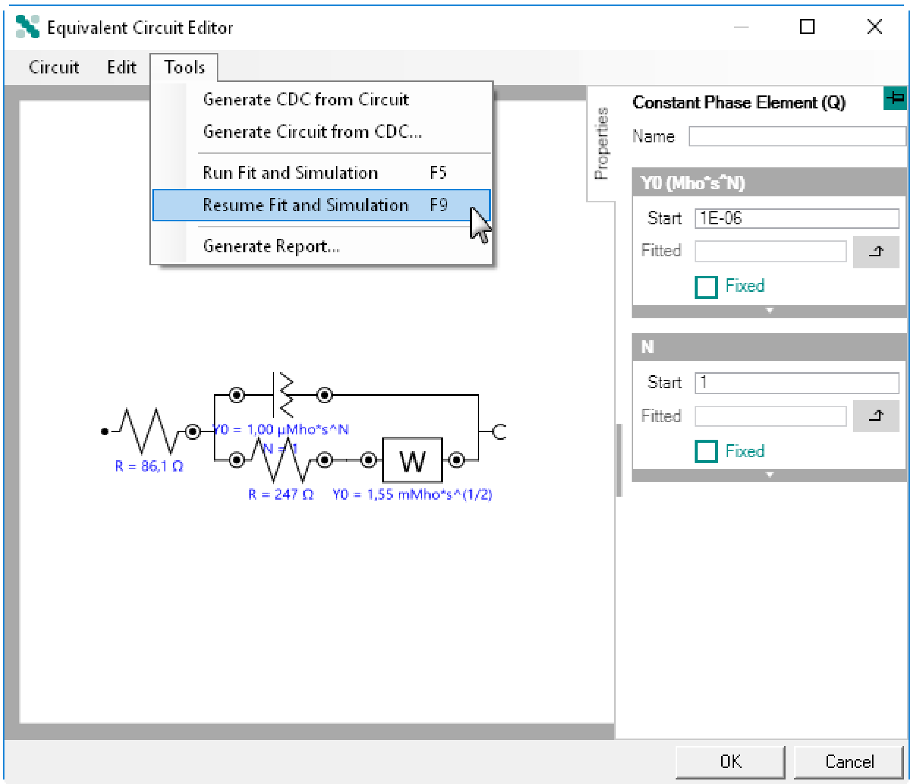


Figure 987 Resuming the calculation

Using the *Run Fit and simulation* (**[F5]**) option will trigger the calculation to restart using the *Start* value of each element. Using the *Resume Fit and simulation* (**[F9]**) option will trigger the calculation to continue using the *Fitted* values of each element.

NOTE

It is possible to assign the *Fitted* value for a circuit element as a new *Start* value by clicking the button in the **Properties** panel .

12.10.3 Viewing the result

When the calculation is complete, it is possible to view the details by clicking the Edit button in the **Properties** panel *(see Figure 988, page 804)*.

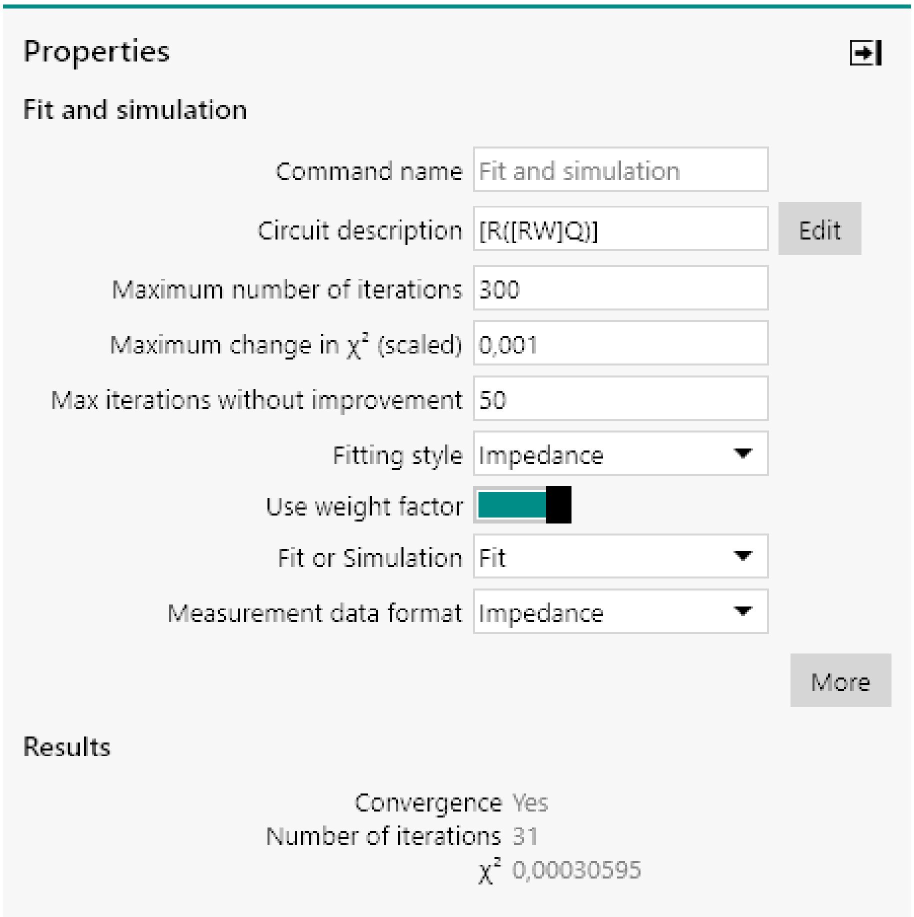


Figure 988 Opening the Equivalent Circuit Editor window

The **Equivalent Circuit Editor** will be shown, displaying the final values for each circuit element. A report can be generated by selecting the *Generate Report* option from the Tools menu *(see Figure 989, page 805)*.

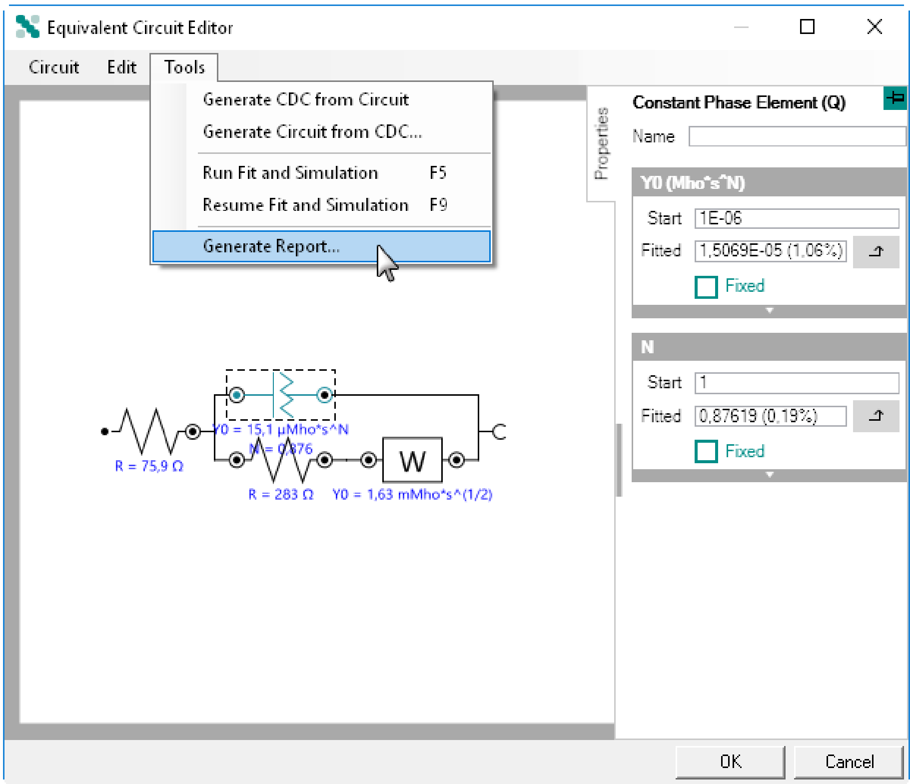
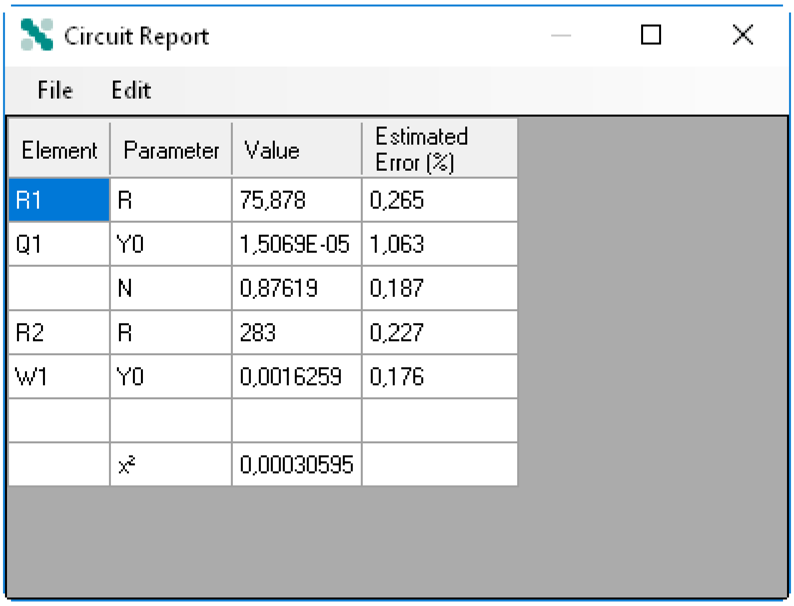


Figure 989 Generating the report

The report will display the fitted values for each element as well as the estimated error and the total χ^2 value *(see Figure 990, page 805)*.


Circuit Report

File Edit

Element	Parameter	Value	Estimated Error (%)
R1	R	75,878	0,265
Q1	Y0	1,5069E-05	1,063
	N	0,87619	0,187
R2	R	283	0,227
W1	Y0	0,0016259	0,176
	χ^2	0,00030595	

Figure 990 The Circuit Report

NOTE

The data in the **Circuit report** can be exported to ASCII using the **File** menu or can be copied to the Clipboard using the **Edit** menu.

The **Circuit Report** provides the following information:

- **Element:** this is the identification of the circuit element. If a unique name has been specified in the **Equivalent Circuit Editor**, this name will be used instead.
- **Parameter:** indicates the fitted or calculated property of the circuit element.
- **Value:** indicates the fitted or calculated property of the circuit element property.
- **Estimated error:** indicates the estimated error for the element property. This value is indicated in %.

NOTE

The **Estimated error** is calculated by testing marginal variations of the fitted or calculated value near the convergence. For example, if the best value for a particular resistor is 100 Ohms, the value is increased/decreased until the goodness of fit starts to decrease. If 98 and 102 Ohms produces a very similar goodness of fit, but 97 and 103 Ohms produces a poorer fit, the Error is reported as 2/100 * 100 = 2%. Very large error estimates are typically a result of an incorrect model - often one that contains more elements than are represented by the data. If the model contains too many elements, the 'extra' element has no effect on the goodness of fit.

13 Data handling

When data has been measured, it is possible to use the data handling commands provided in NOVA to process and handle the data. To apply data handling command to data acquired in NOVA, it is necessary to add the required command to the measured procedure and apply the function of these commands on the measure data.

NOTE

Data handling commands can be added to the initial procedure or the procedure after the measurement is finished.

To add a data handling command to a measured procedure, two methods can be used:

- Drag and drop the data handling command in the procedure
- Use the contextual shortcut button, located in the top right corner of the procedure editor

The functionality of the data handling commands commands is explained in the previous chapters and will not be detailed again in this chapter. This chapter focuses on the use of these commands on **measured** data. Only the commands that provide controls that are used in a specific way on data are detailed in this chapter.

The following commands are detailed:

- Get item
- Shrink data

13.1 Get item

The **Get item** command provides additional controls that can be used when the command is used to analyze data. To use the **Get item** command, this command can be added to the procedure as a command, using the drag and drop method, or by using the button. In the latter case, a pop-out menu is displayed, providing a list of commands that can be applied on the selected command *(see Figure 991, page 808)*.

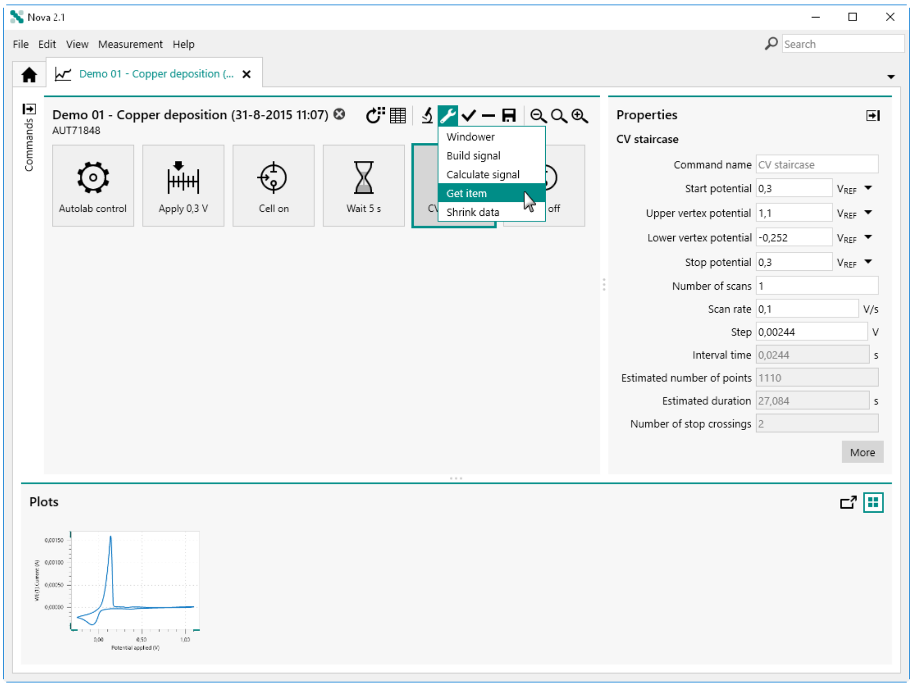


Figure 991 *Adding a Get item command to the CV staircase command*

The **Get item** command is added to the procedure editor. The **Edit links** screen will be show immediately after the command is added *(see Figure 992, page 808)*.

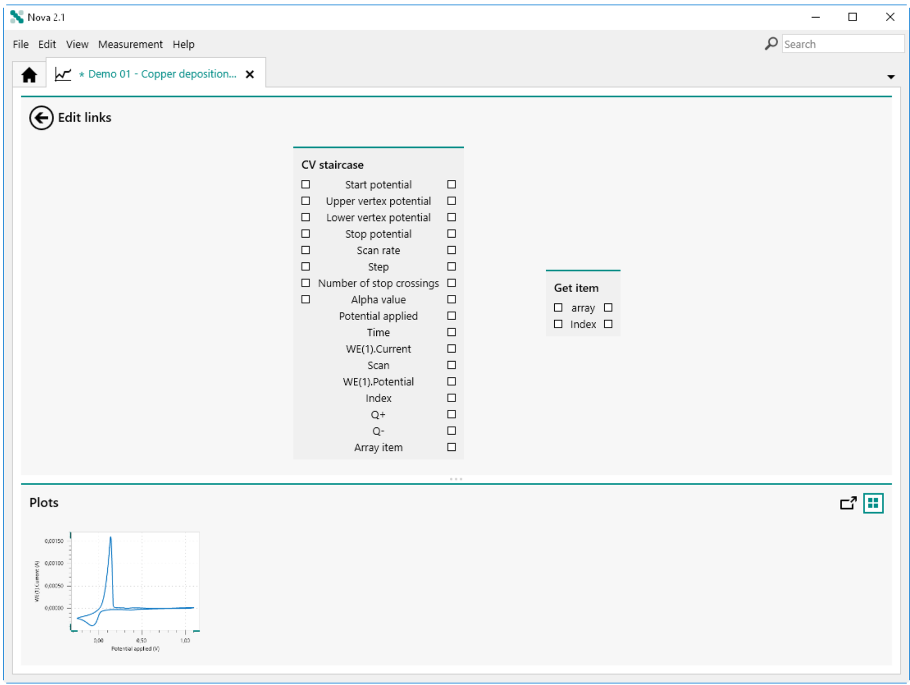


Figure 992 *The Edit links screen is automatically shown when the Get item command is added to the procedure*

Using the method described in *Chapter 10.13*, the links required for the **Get item** command can be edited *(see Figure 993, page 809)*.

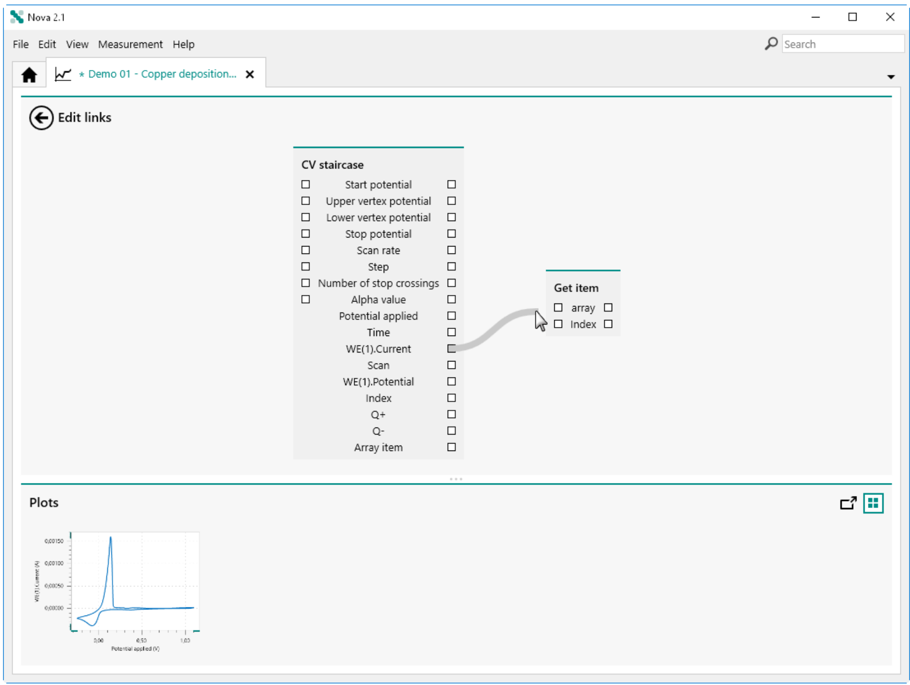


Figure 993 The links required by the Get item command can be edited

Clicking the button closes the **Edit links** screen and returns to the procedure editor. The properties of the **Get item** command can now be edited in the **Properties** panel *(see Figure 994, page 809)*.

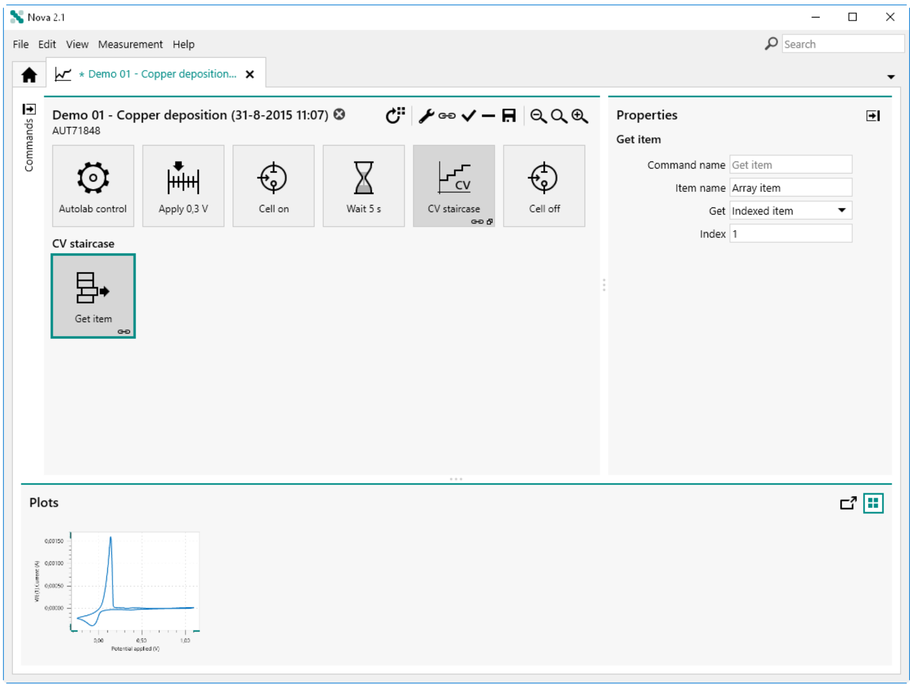


Figure 994 The properties of the Get item command can be set

If the links required by the **Get item** command are not properly, an error will be displayed in the procedure editor *(see Figure 995, page 810)*.

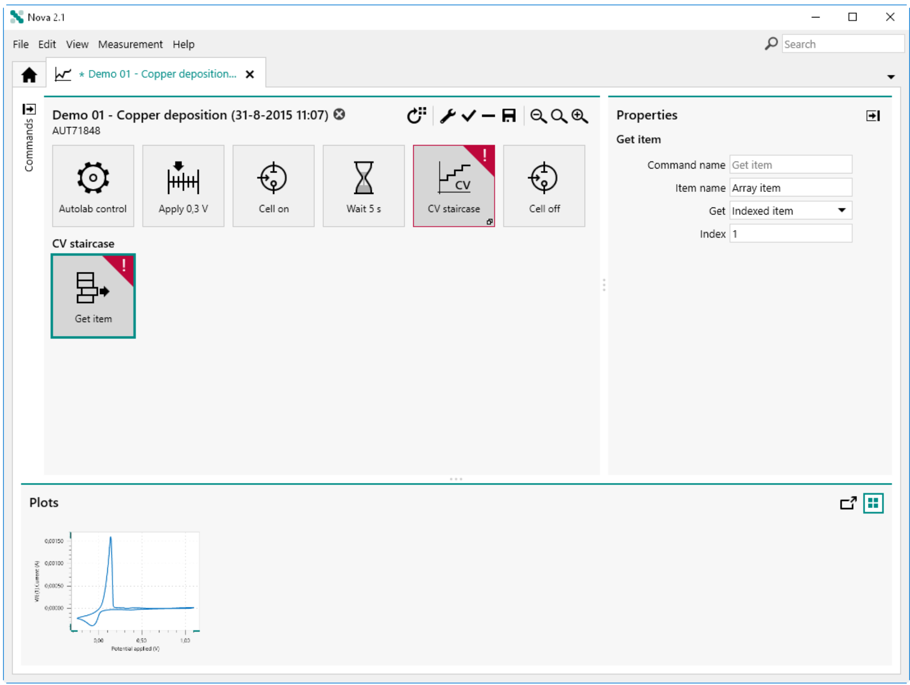


Figure 995 The procedure validation will trigger an error when the links are not set properly

NOTE

For more information on the properties of the **Get item** command, please refer to *Chapter 7.7.4*.

13.2 Shrink data

The **Shrink data** command provides additional controls that can be used when the command is used to handle data. To use the **Shrink data** command, this command can be added to the procedure as a command, using the drag and drop method, or by using the button. In the latter case, a pop-out menu is displayed, providing a list of commands that can be applied on the selected command *(see Figure 996, page 811)*.

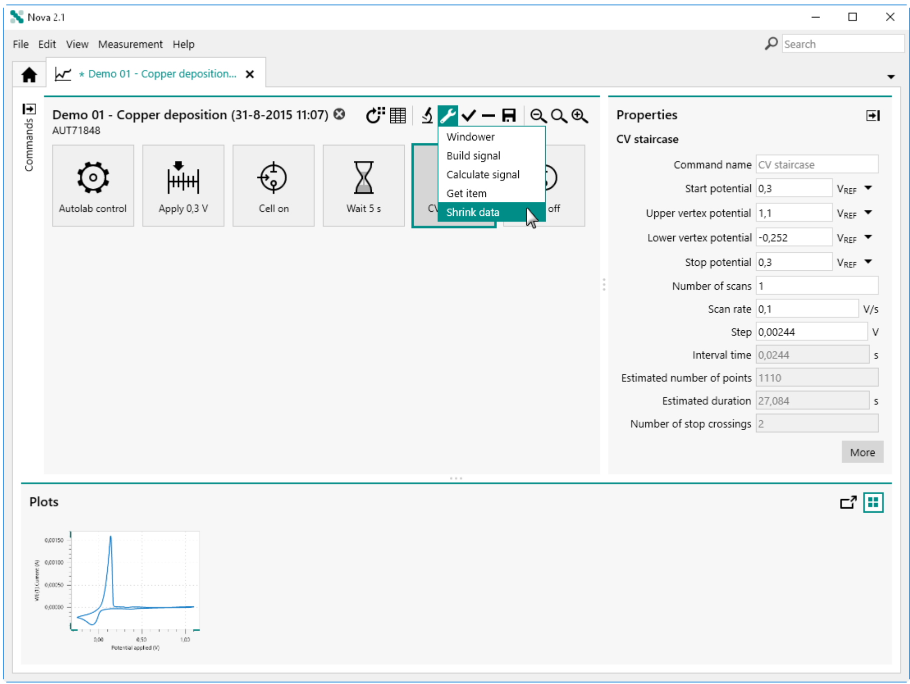


Figure 996 Adding a Shrink data command to the CV staircase command

The **Shrink data** command is added to the procedure editor. The **Edit links** screen will be show immediately after the command is added *(see Figure 997, page 811)*.

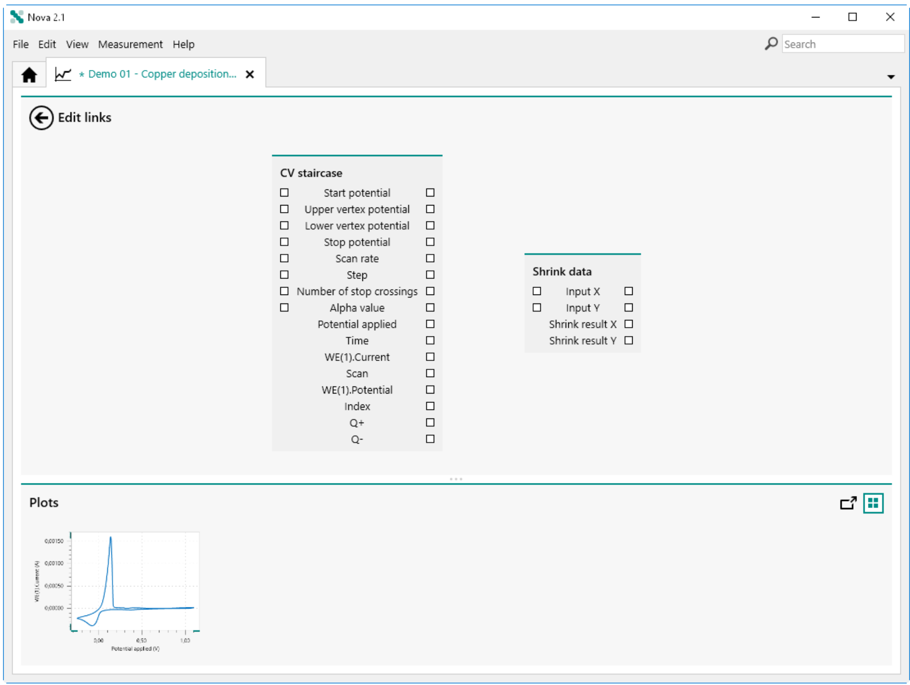


Figure 997 The Edit links screen is automatically shown when the Shrink data command is added to the procedure

Using the method described in *Chapter 10.13*, the links required for the **Shrink data** command can be edited *(see Figure 998, page 812)*.

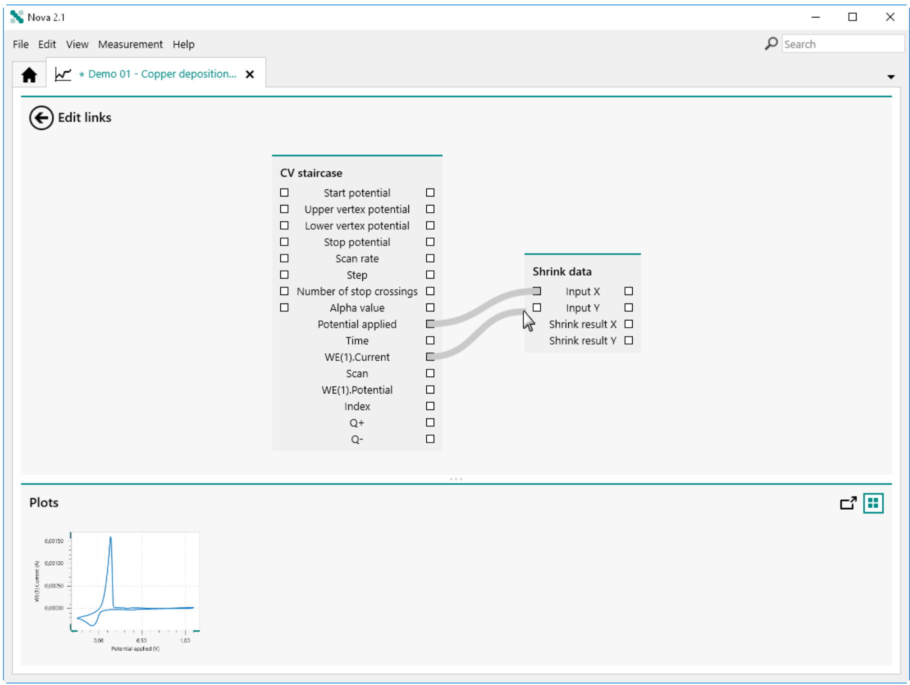


Figure 998 Setting the links for the Shrink data command

Clicking the button closes the **Edit links** screen and returns to the procedure editor. The properties of the **Shrink data** command can now be edited in the **Properties** panel *(see Figure 999, page 812)*.

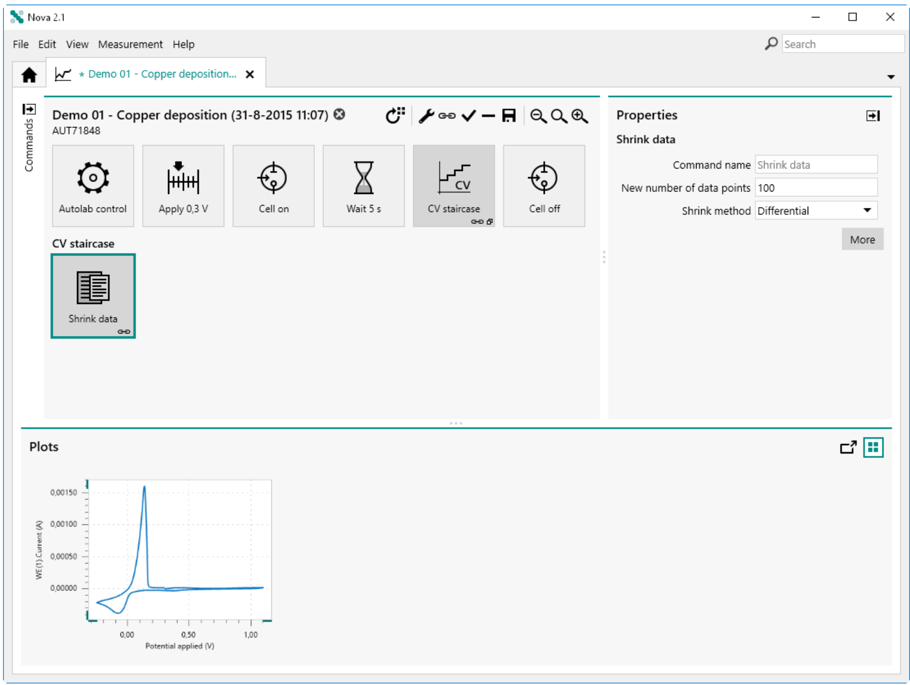


Figure 999 The properties of the Shrink data command can be set

If the links required by the **Shrink data** command are not properly, a warning will be displayed in the procedure editor *(see Figure 995, page 810)*.

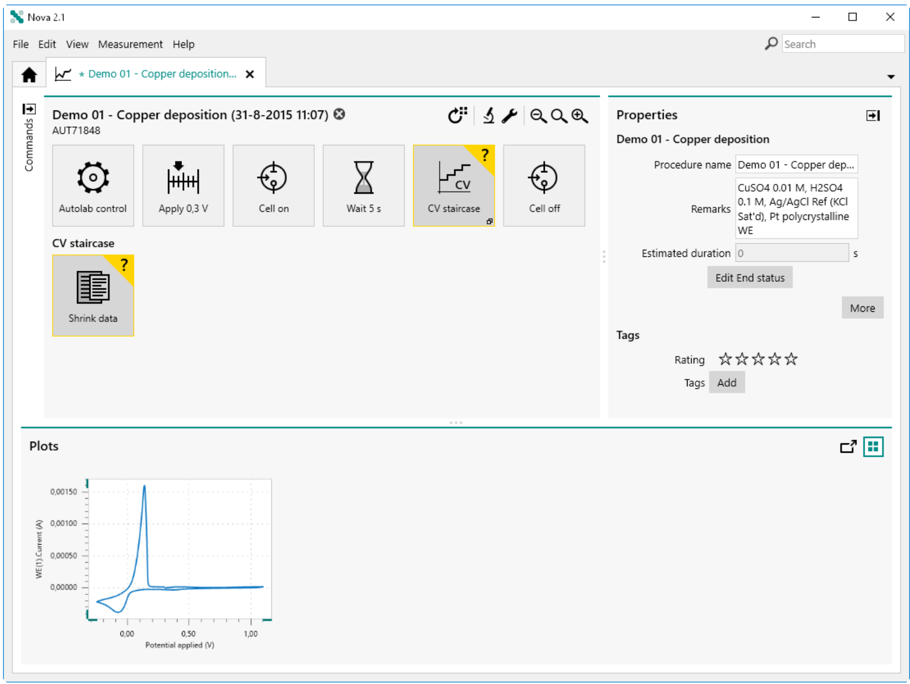


Figure 1000 The procedure validation will trigger a warning when the links are not set properly

NOTE

For more information on the properties of the **Shrink data** command, please refer to *Chapter 7.7.8*.

14 Data overlays

NOVA provides the means to create data overlays at any time during or after a measurement. This provides the means to compare data from different experiments in a convenient way. Data overlays in NOVA are created in a separate tab and are volatile, which means that these overlays are not saved and that the content of each overlay is discarded when NOVA is closed.

It is possible to create as many overlays as needed and it is possible to add as many plots as required to any overlay.

This chapter explains the following controls of the data overlays in NOVA:

1. Creating an overlay *(see Chapter 14.1, page 814)*
2. Adding data to an overlay *(see Chapter 14.2, page 817)*
3. Hiding and showing data in an overlay *(see Chapter 14.4, page 821)*
4. Editing the data plotted in an overlay *(see Chapter 14.3, page 818)*
5. Removing data from an overlay *(see Chapter 14.5, page 824)*

14.1 Create an overlay

To create an overlay, right-click a plot of an open measurement or data and select the *Add to new overlay* option from the context menu *(see Figure 1001, page 815)*.

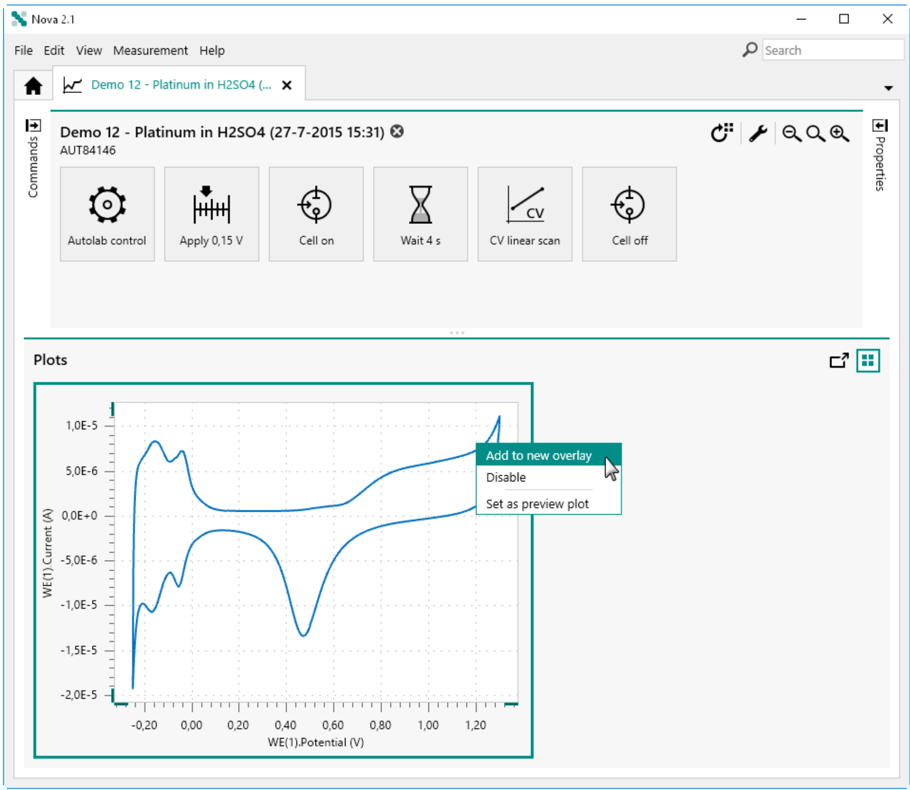


Figure 1001 Right-click a plot to create a new overlay

NOTE

It is possible to right-click a plot from a saved file or from an ongoing measurement.

A new overlay will be created in a new tab and the data from the source dataset will be added to this overlay *(see Figure 1002, page 816)*.

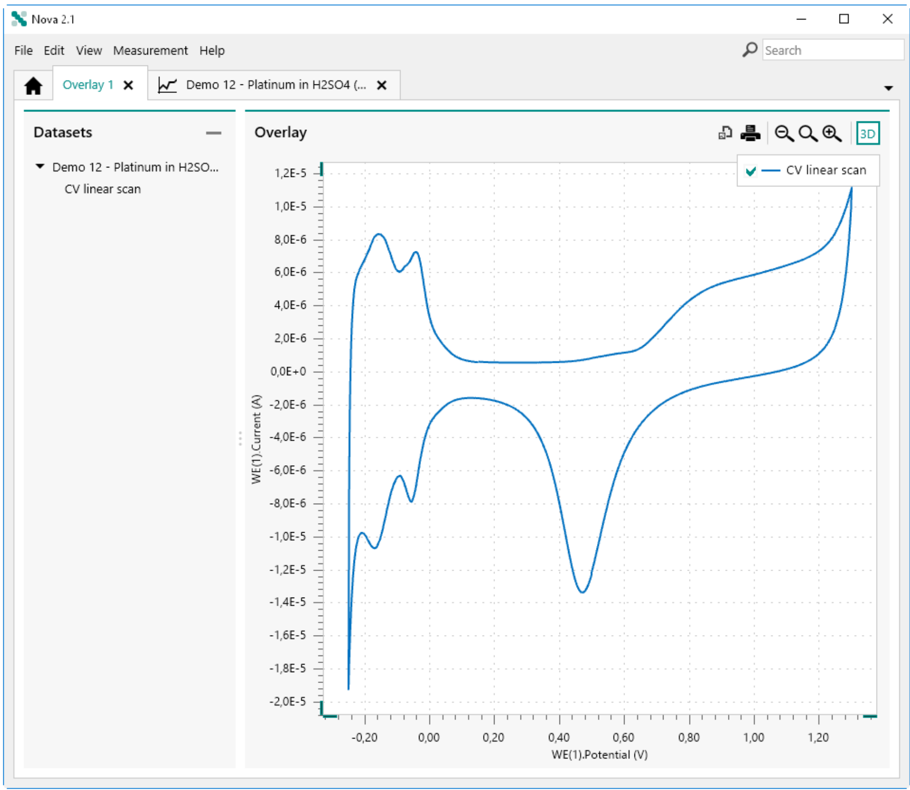


Figure 1002 The data is added to the overlay

> **NOTE**
>
> By default, the Overlay tab starts at number 1 and overlays will be incremented until NOVA is closed.

The information provided in the Overlay tab is distributed in two different panels:

- **Datasets panel:** this panel lists all of the datasets added to the overlay.
- **Overlay panel:** this panel provides a plot of the data from the datasets added to the overlay.

14.2 Adding data to an overlay

To add a new dataset to an existing overlay, right-click a plot from the new source dataset and select the *Add to overlay X* option, where X is the number of the target overlay *(see Figure 1003, page 817)*.

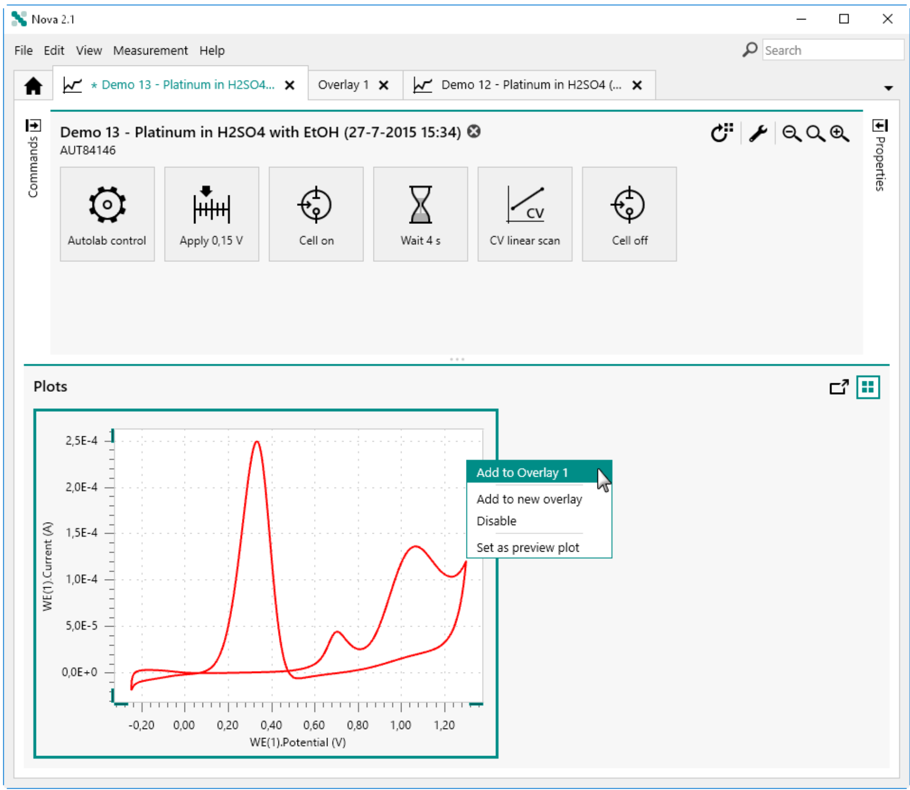


Figure 1003 Adding a dataset to an existing overlay

The new dataset will be added to the target overlay. The information in the **Datasets** panel will be updated, indicating that a new dataset is available in the Overlay tab *(see Figure 1004, page 818)*.

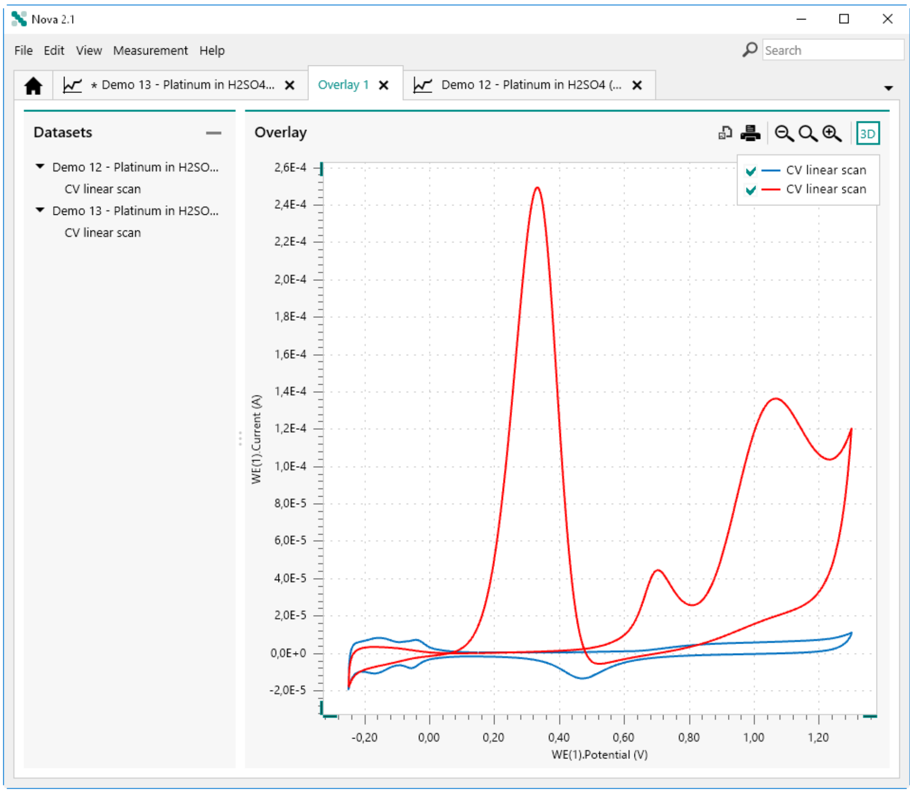


Figure 1004 The new dataset is added to the Overlay

The data from the new dataset will be added to the Overlay panel on the right-hand side *(see Figure 1004, page 818)*.

14.3 Changing overlay plot settings

It is possible to adjust the way the data is plotted in the **Overlay** panel and to change the signal used on the X, Y and Z axis at any time. To change the axes settings, right-click on one of the axes and select a new signal from the popout menu *(see Figure 1005, page 819)*.

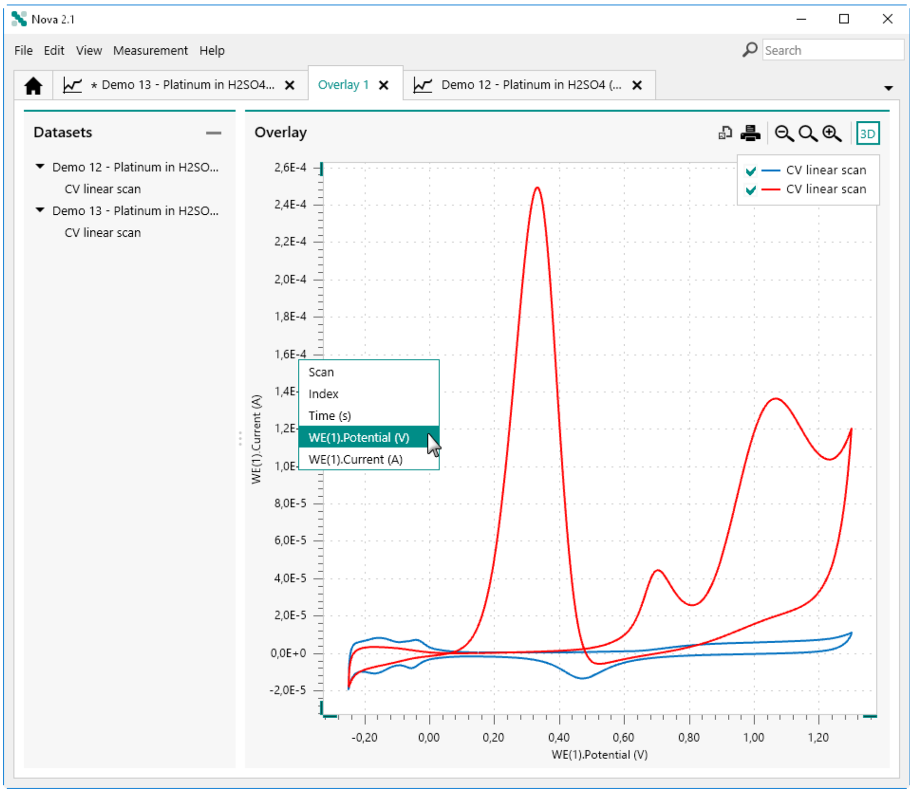


Figure 1005 Changing the Y axis signal for the overlay

NOTE

The popout menu shows the available common signals provided by all of the datasets in the overlay.

When a new signal is selected, all of the datasets in the Overlay panel will be replotted, using the new signal *(see Figure 1006, page 820)*.

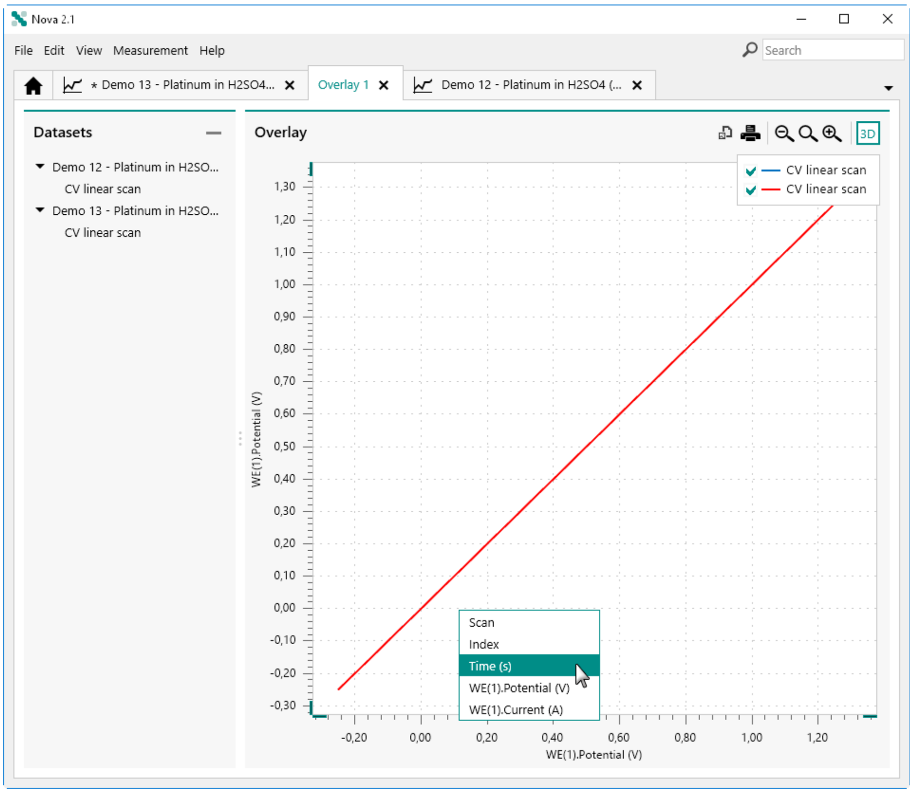


Figure 1006 Changing the X axis signal for the overlay

It is possible to repeat this for each axis, as shown in *Figure 1006*. Each time a signal is modified, all the data shown in the Overlay panel will be updated *(see Figure 1007, page 821)*.

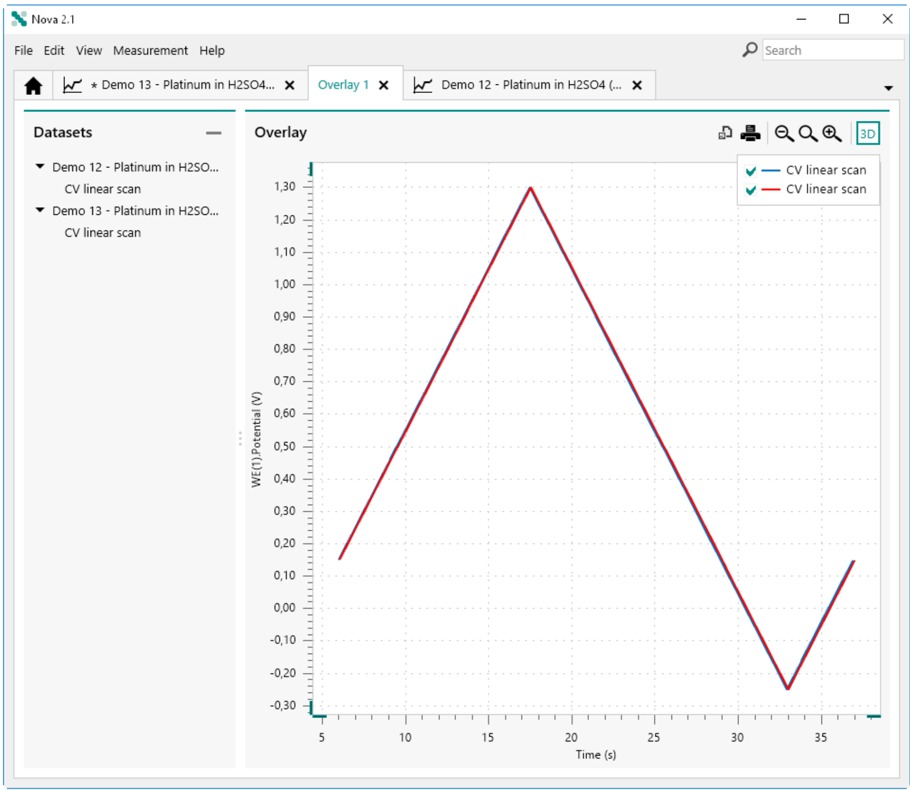


Figure 1007 The data is replotted after the new signal is selected

14.4 Hiding and showing plots

The Legend box, shown in the top right corner of the plot of the **Overlay** panel can be used to show or hide plots. For each plot, a checkbox is available *(see Figure 1008, page 822)*.

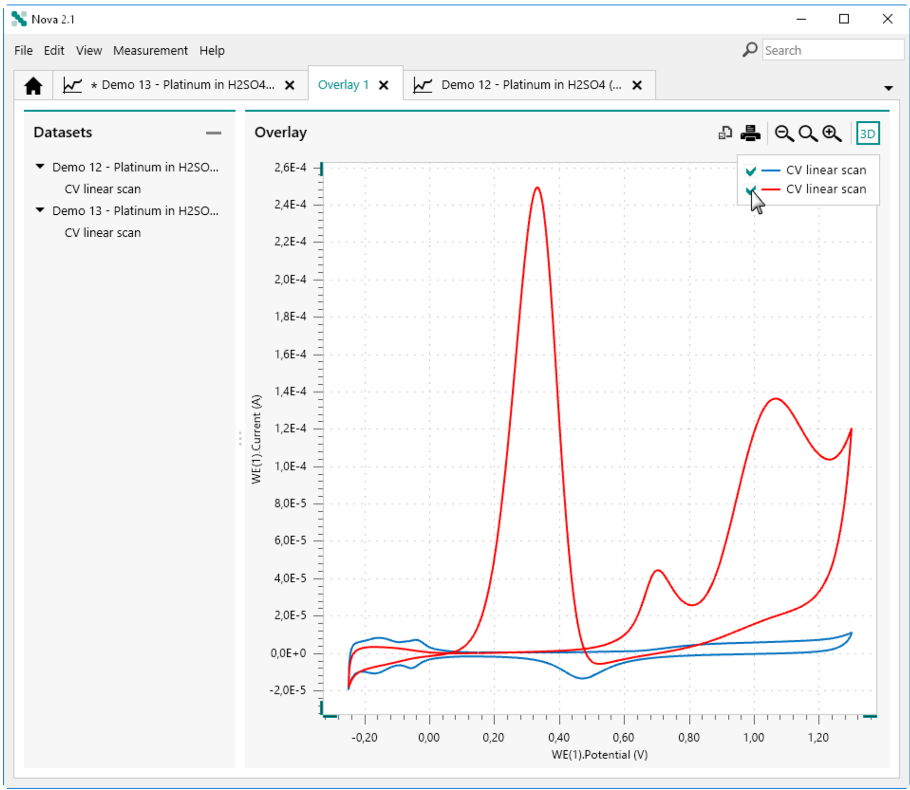


Figure 1008 Checkboxes are provided in the Legend box of the Overlay panel

Using this control, it is possible to show or hide any of the available plots *(see Figure 1009, page 823)*.

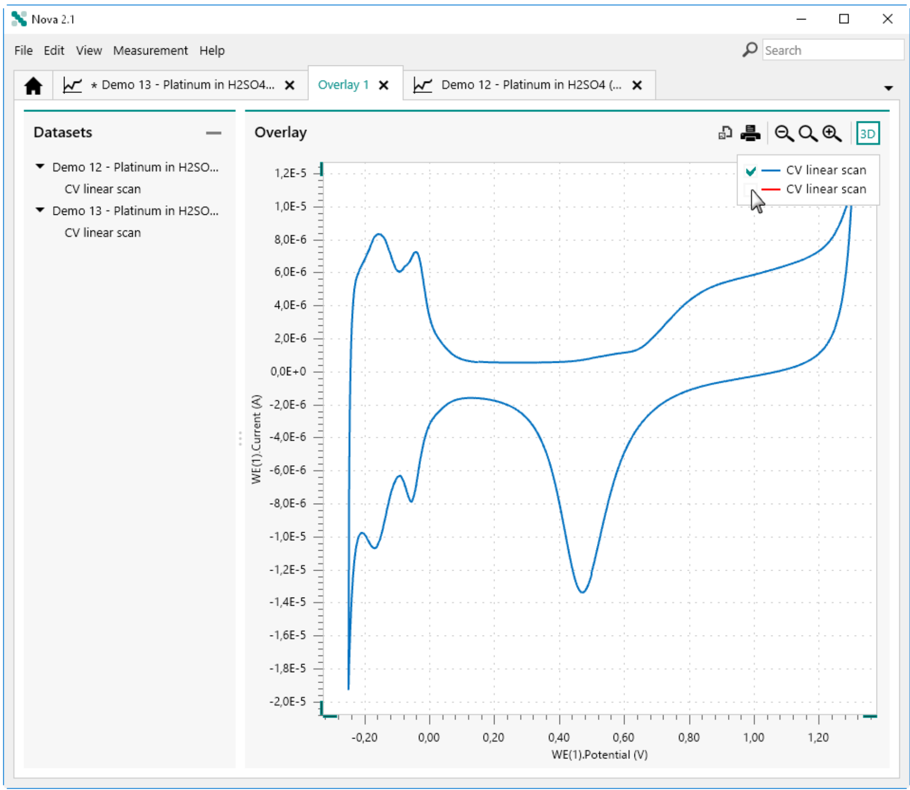


Figure 1009 The checkboxes can be used to hide or show plots in the overlay

At any time, it is possible to use this control to show hidden plots or hide shown plots *(see Figure 1010, page 824)*.

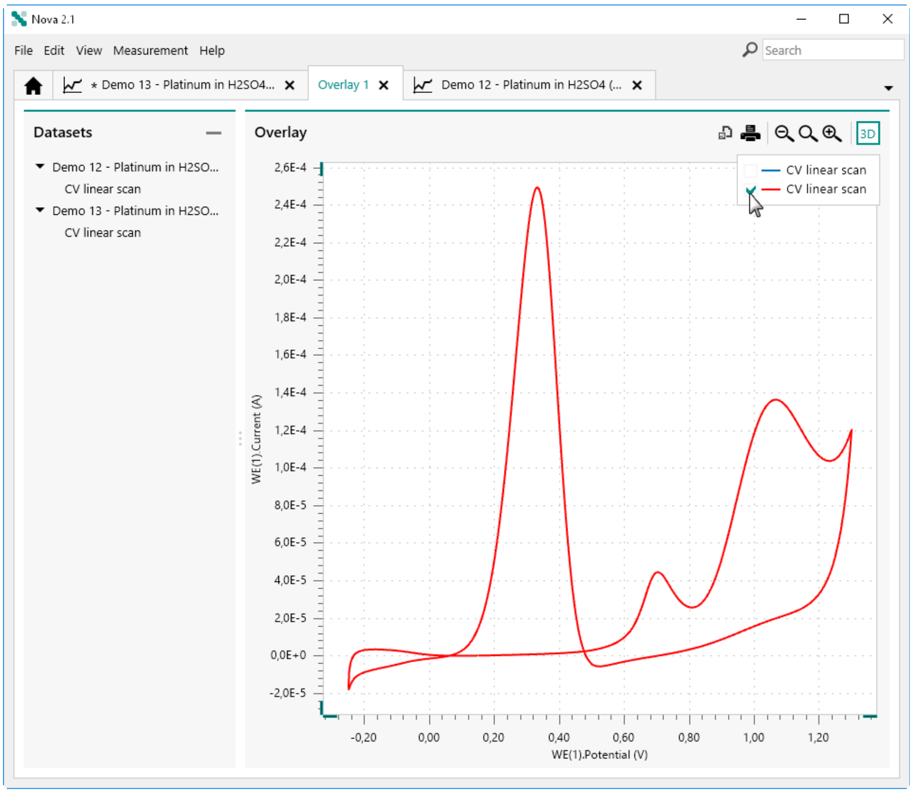


Figure 1010 The hidden and shown plots can be adjusted at any time

14.5 Remove data from overlay

It is possible to remove a dataset from an overlay by selecting the dataset in the **Datasets** panel and clicking the ▬ button in the top right corner of the panel *(see Figure 1011, page 825)*.

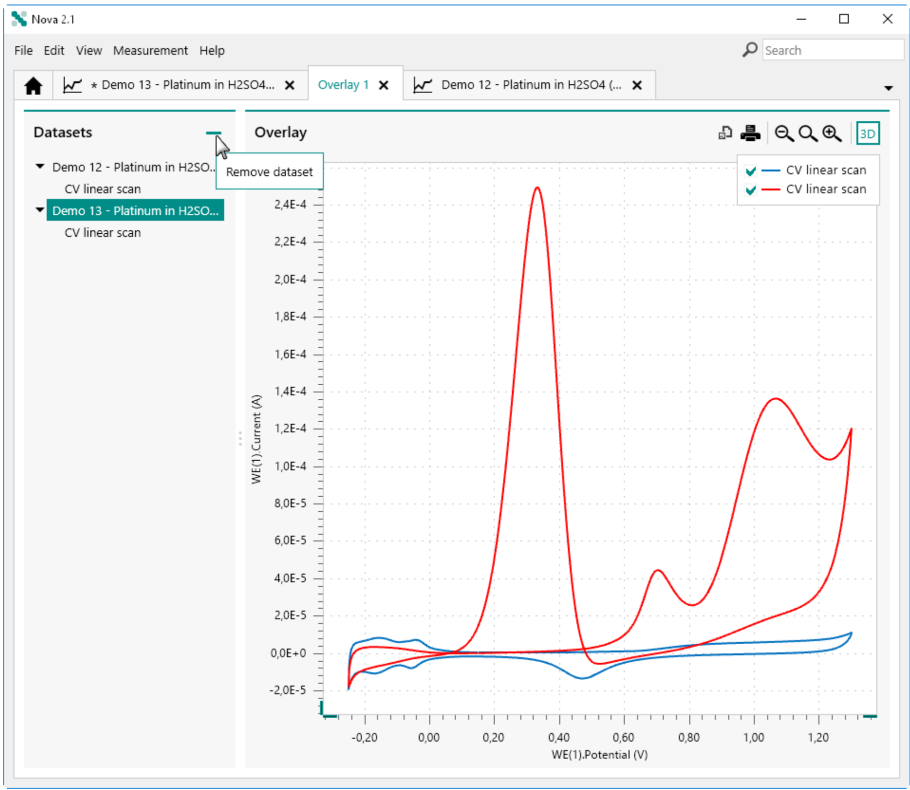


Figure 1011 Select the dataset to remove

The dataset will be remove from the overlay and the plot displayed in the **Overlay** panel will be updated *(see Figure 1012, page 826)*.

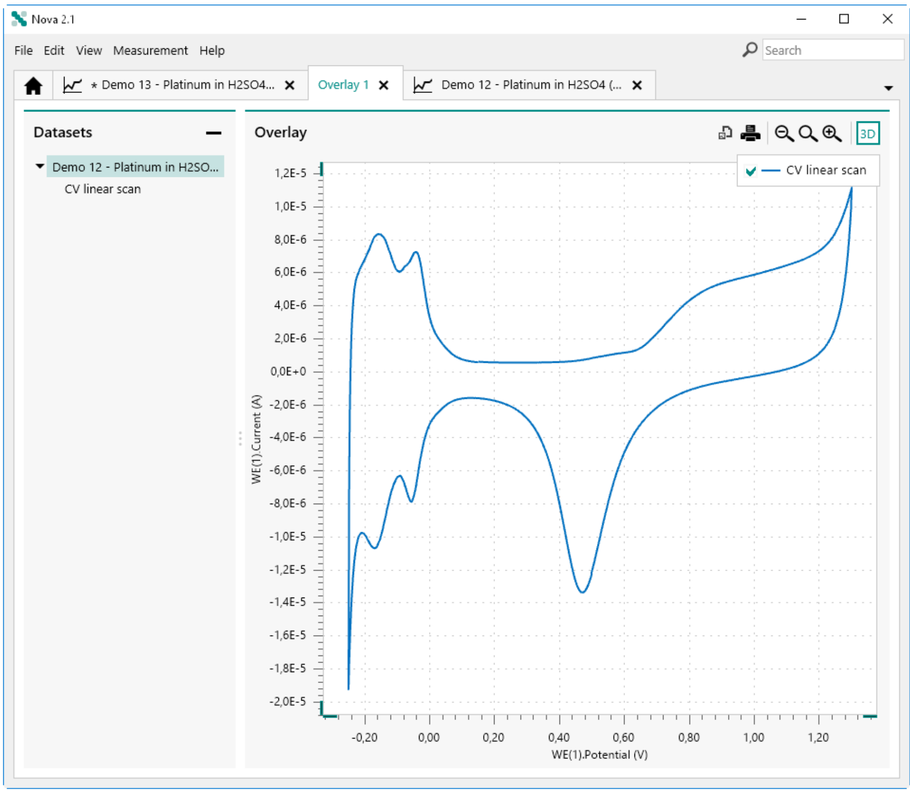


Figure 1012 The selected dataset is removed from the overlay

14.6 Additional Overlay controls

Additional controls are available in the in **Overlay** panel, through the dedicated buttons in the top right corner *(see Figure 1013, page 827)*.

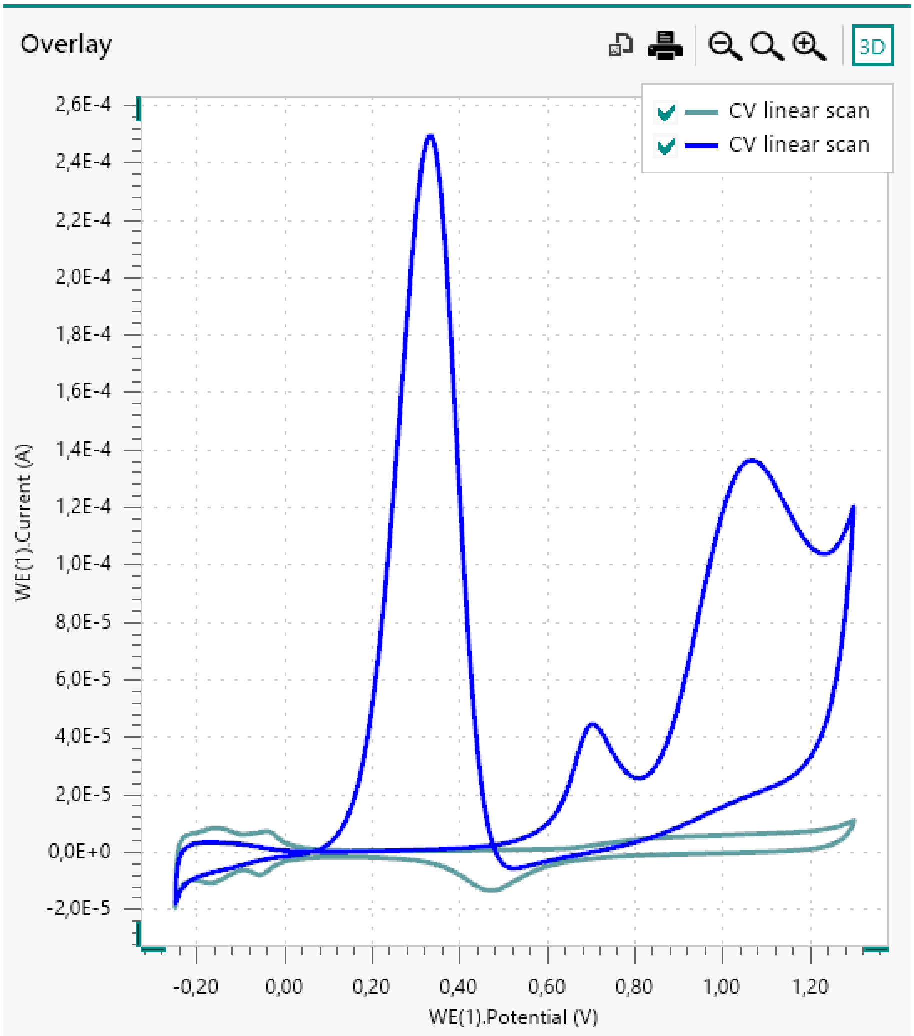


Figure 1013 Additional controls are available in the top right corner of the Overlay frame

NOTE

These controls are the same as the controls available normal plots.

The following controls are available:

- **3D view (button):** toggles the 3D plot on or off *(see Chapter 11.8.2, page 704)*.
- **Zoom in (button):** zooms in on the plot *(see Chapter 11.8.5, page 707)*.
- **Fit view (button):** fits all the data on the plot *(see Chapter 11.8.5, page 707)*.
- **Zoom out (button):** zooms out on the plot *(see Chapter 11.8.5, page 707)*.

- **Print plot (🖶):** prints the plot *(see Chapter 11.8.6, page 708)*.
- **Export image (button):** export the data to an image file *(see Chapter 11.8.7, page 710)*.

15 Procedure scheduler

The procedure scheduler is an advanced feature of NOVA. The procedure scheduler can be used to specify a series of procedures to run in sequence on one or more instruments connected to the computer. Each instrument involved used in the scheduler will run the specified procedures sequentially without user intervention.

To create a new procedure schedule, click the New schedule button in the **Actions** panel in the dashboard *(see Figure 1014, page 829)*.

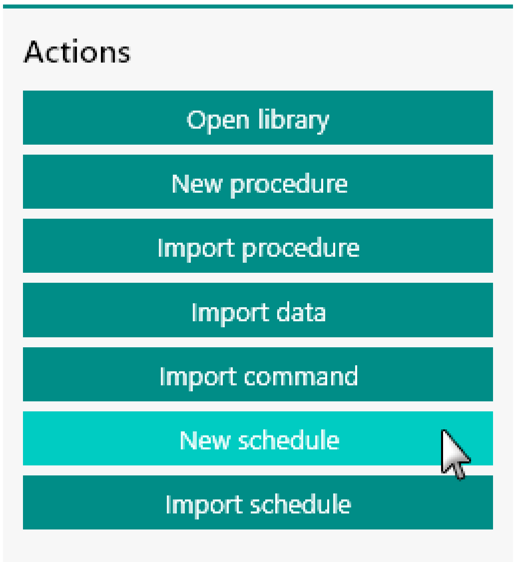


Figure 1014 Starting a new schedule

NOTE

It is also possible to import an existing schedule into the **Library** by clicking the Import schedule button.

A new tab will be created and the controls for the procedure scheduler will be displayed *(see Figure 1015, page 830)*.

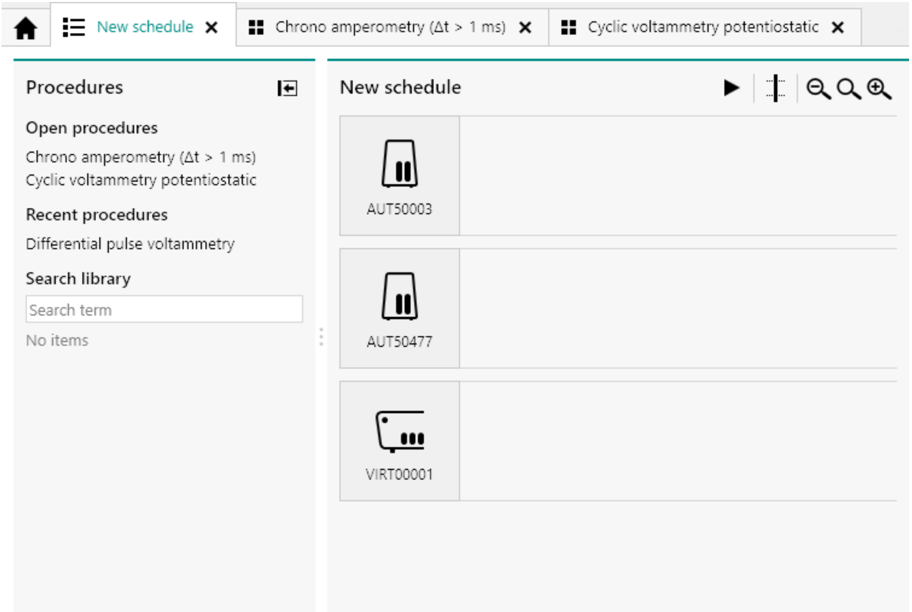


Figure 1015 The procedure scheduler

The procedure scheduler provides two panels:

- **Procedures panel:** a panel that displays all open procedures, all recent procedures and a search box which can be used to search any location defined in the **Library** for a given procedure.
- **New schedule panel:** this panel list all available instrument connected to the computer and the procedure schedule for each instrument.

NOTE

When a new procedure scheduler is started, all connected instrument are automatically listed in the **New schedule** panel. Instruments that are busy are also listed in the new schedule panel. These instruments will not be able to start a measurement until the current measurement is finished.

Using the controls provided in the procedure scheduler tab, it is possible to carry out the following tasks:

- Remove instruments from the procedure scheduler.
- Add a procedure to an instrument schedule.
- Create a synchronization point.
- Run the procedure scheduler.
- Inspect data from a running procedure.

15.1 Remove instrument from schedule

To remove instruments from the procedure scheduler, select the instrument to remove in the New schedule panel and press the **[Delete]** key *(see Figure 1016, page 831)*.

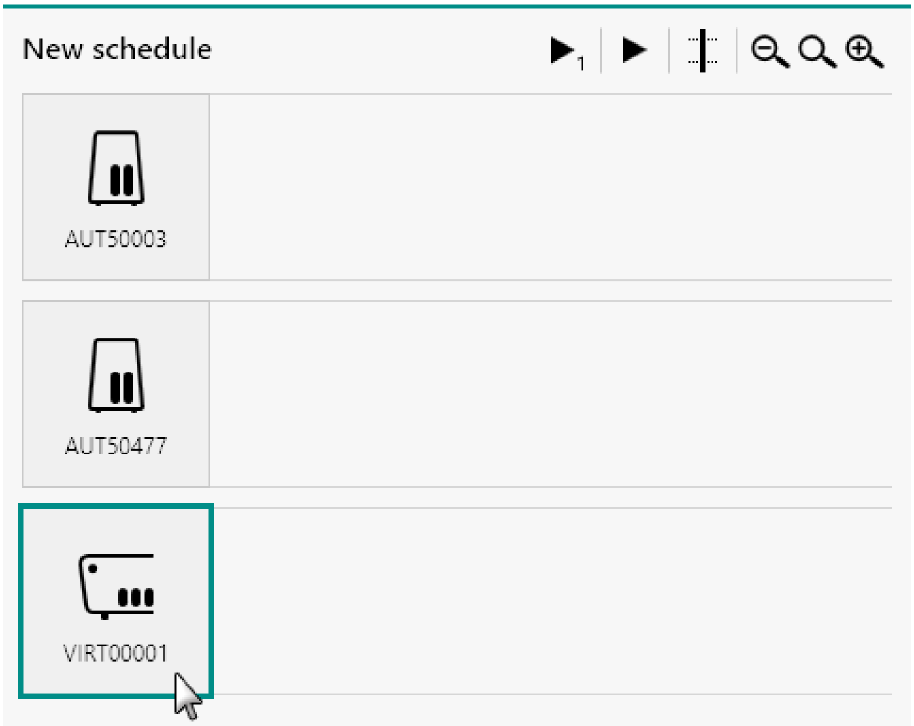


Figure 1016 Removing an instrument from the procedure scheduler

The selected instrument will be removed from the procedure scheduler *(see Figure 1017, page 831)*.

Figure 1017 The instrument is removed from the procedure scheduler

15.2 Creating a procedure schedule

To create a procedure schedule for one or more instruments in the procedure scheduler, it is necessary to add the required procedures from the **Procedures** panel on the left hand side to the **New schedule** panel on the right hand side. Adding procedures is performed using the *drag and drop* method *(see Figure 1018, page 832)*.

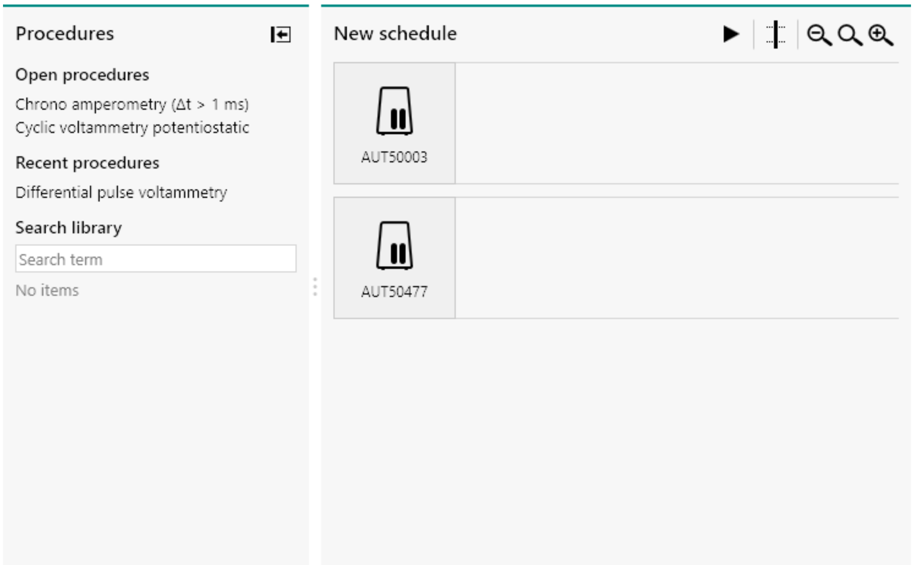


Figure 1018 Creating a procedure schedule

It is possible to add procedures from three different sources to a procedure schedule:

- **Open procedures:** these are all the procedures currently open in NOVA.
- **Recent procedures:** these are the five last saved procedures.
- **Search Library:** this search option can be used to search for any procedure in the **Library**.

15.2.1 Open procedures

The procedures listed under *Open procedures* in the **Procedures** panel are all procedures that are currently open for editing in NOVA. Any of these open procedures can be dragged over to the **New schedule** panel in order to add it a procedure schedule for one of the available instrument *(see Figure 1019, page 833)*.

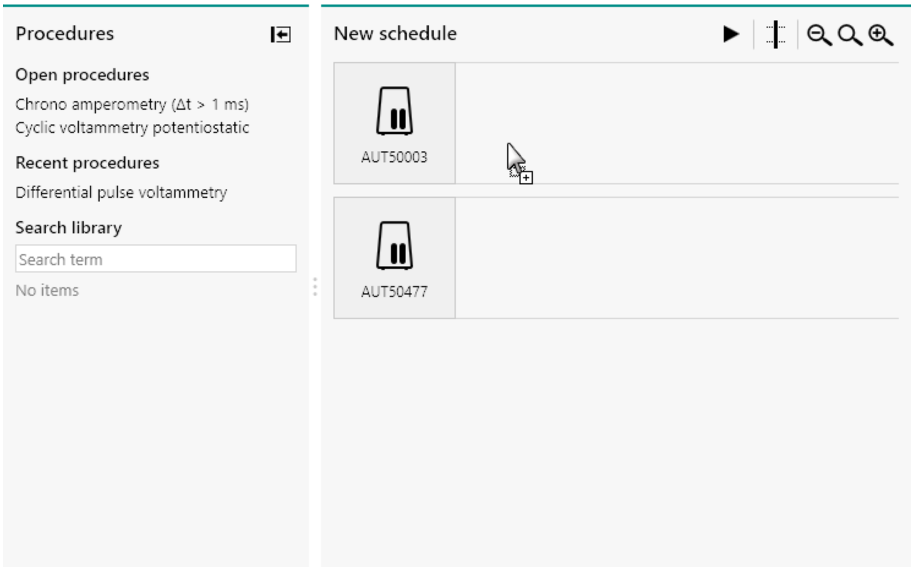


Figure 1019 Dragging the open procedure to the schedule

Using the drag and drop method, select an open procedure and drag it to an instrument schedule. Release the mouse button to add the procedure to the schedule. The procedure will be added to the instrument schedule, identified by a white box next to the instrument tile *(see Figure 1020, page 833)*.

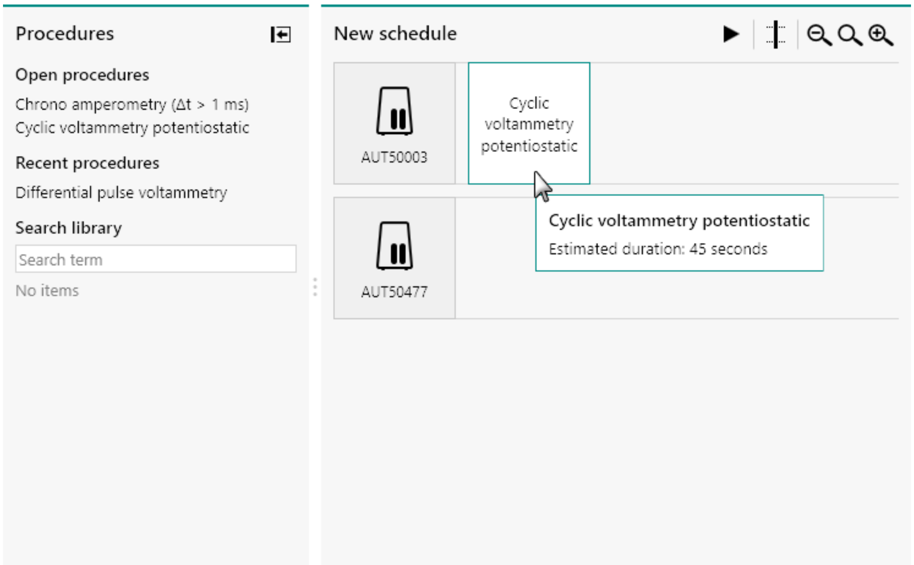


Figure 1020 The procedure is added to the schedule

NOTE

A tooltip indicates the expected duration of the procedure in the schedule.

15.2.2 Recent procedures

The procedures listed under *Recent procedures* in the **Procedures** panel are the five last saved procedure. Any of these procedures can be dragged over to the **New schedule** panel in order to add it a procedure schedule for one of the available instrument *(see Figure 1021, page 834)*.

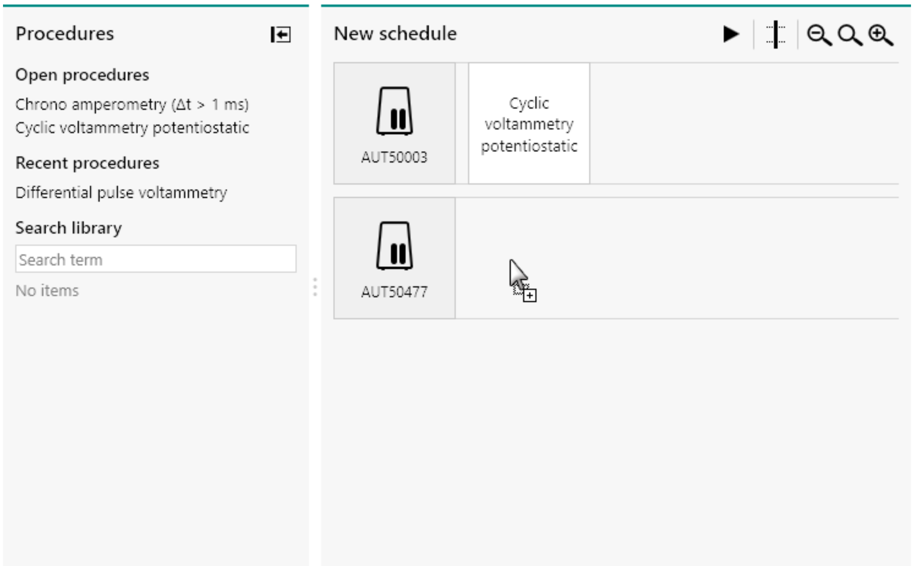


Figure 1021 Dragging the recent procedure to the schedule

Using the drag and drop method, select a recent procedure and drag it to an instrument schedule. Release the mouse button to add the procedure to the schedule. The procedure will be added to the instrument schedule, identified by a white box next to the instrument tile *(see Figure 1022, page 834)*.

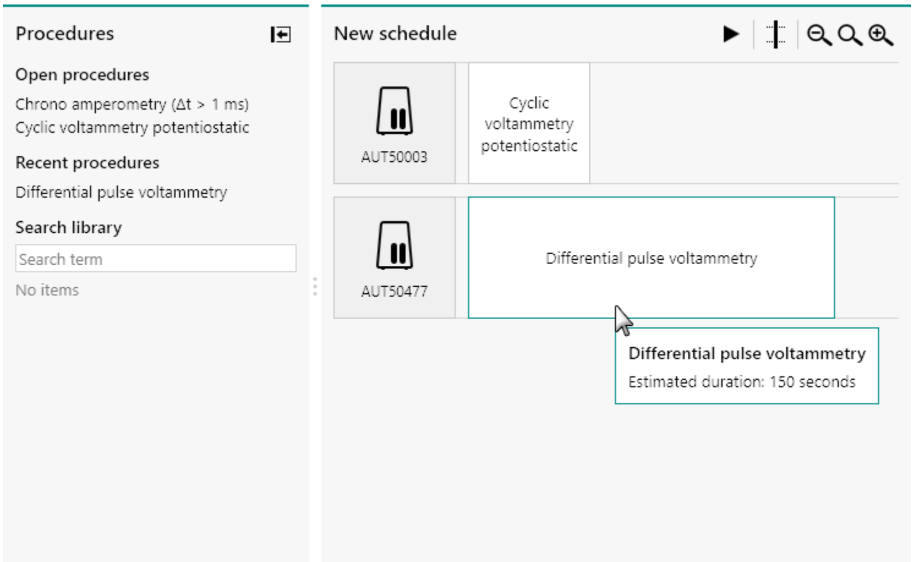


Figure 1022 The procedure is added to the schedule

NOTE

The procedure boxes in the scheduler are scaled with respect to one another to indicate the relative difference in expected duration.

15.2.3 Search Library

The input field located under *Search Library* can be used to search any procedure in any of the locations specified in the **Library** that contains the terms specified in the field. Typing anything in this field will run a search in the background and the list of procedures will updated while typing *(see Figure 1023, page 835)*.

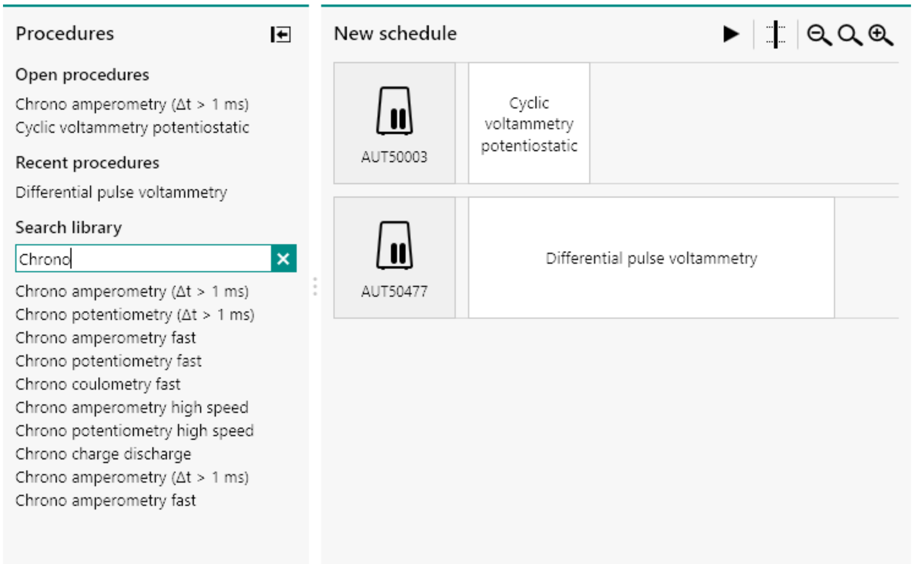


Figure 1023 Searching from procedures in the Library

NOTE

NOVA searches for procedures that contain the specified search criterium in the **Name** of the procedure or in the **Remarks** field.

NOTE

More information on the **Library** is available *Chapter 6*.

The procedures listed under *Search Library* in the **Procedures** panel are all the procedures that match the specified search criterium specified in the input field. Any of these procedure can be dragged over to the **New**

schedule panel in order to add it a procedure schedule for one of the available instrument.

Using the drag and drop method, select an searched procedure and drag it to an instrument schedule. Release the mouse button to add the procedure to the schedule. The procedure will be added to the instrument schedule, identified by a white box next to the instrument tile *(see Figure 1024, page 836)*.

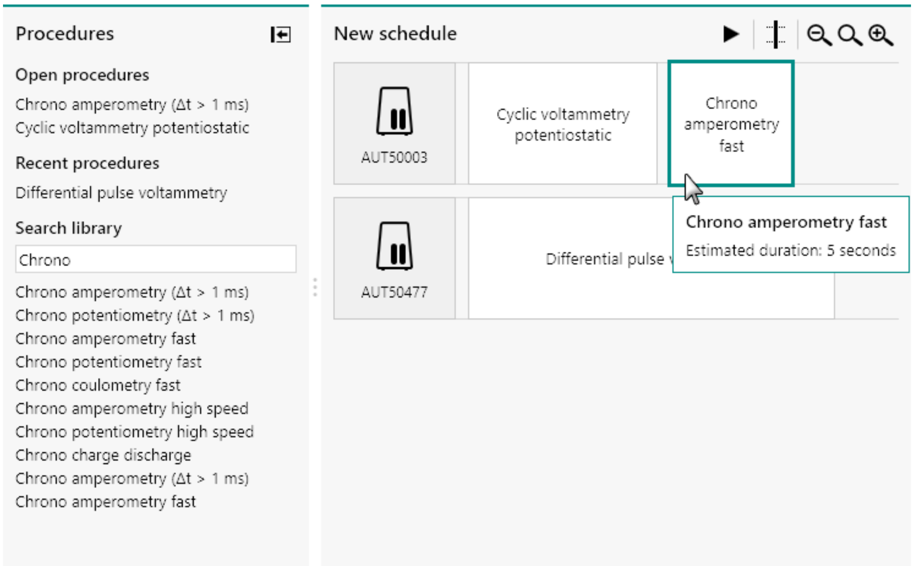


Figure 1024 The procedure is added to the schedule

NOTE

Any time a procedure is added to an instrument schedule, the size of the boxes representing these procedures will be adjusted in order to indicate their respective duration.

15.2.4 Remove procedure

To remove a procedure from a schedule, select the procedure to remove and press the **[Delete]** key *(see Figure 1025, page 837)*.

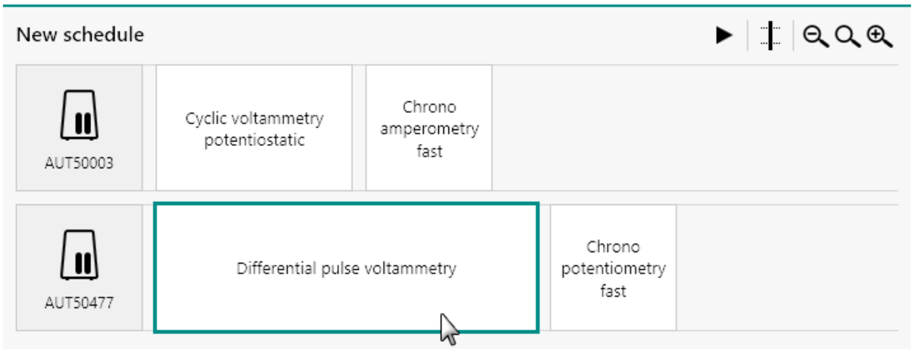


Figure 1025 Select the procedure to remove

The procedure will be removed and the schedule will be rearranged. If needed, the size of the procedure boxes will be readjusted *(see Figure 1026, page 837)*.

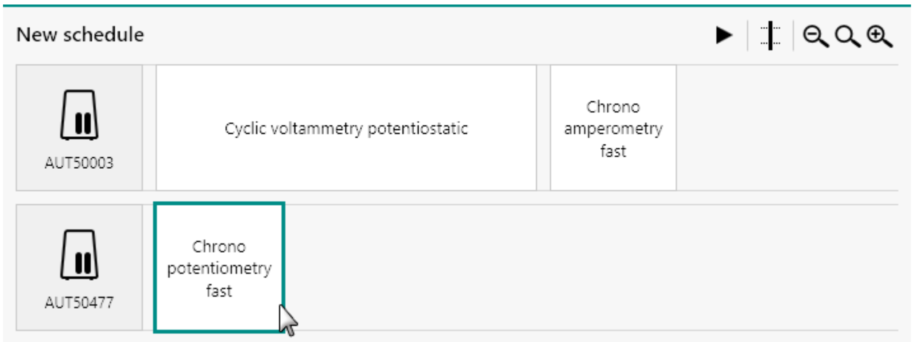


Figure 1026 The procedure is removed from the schedule

15.3 Using synchronization points

It is possible to force instruments involved in a procedure schedule to synchronize their measurements. This can be done by adding a synchronization point. To add a synchronization point to the procedure schedule, click the button, located in the top right corner of the **New schedule** panel *(see Figure 1027, page 837)*.

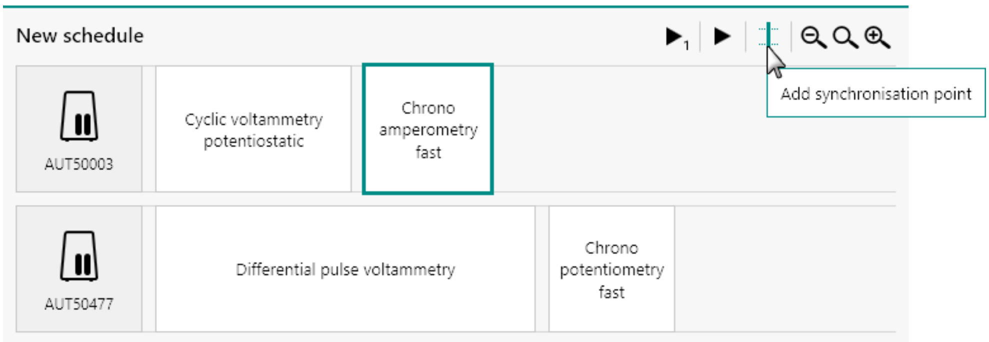


Figure 1027 Adding a synchronization line

A vertical synchronization line will be added to the procedure schedule, after the last procedure in the schedule *(see Figure 1028, page 838)*.

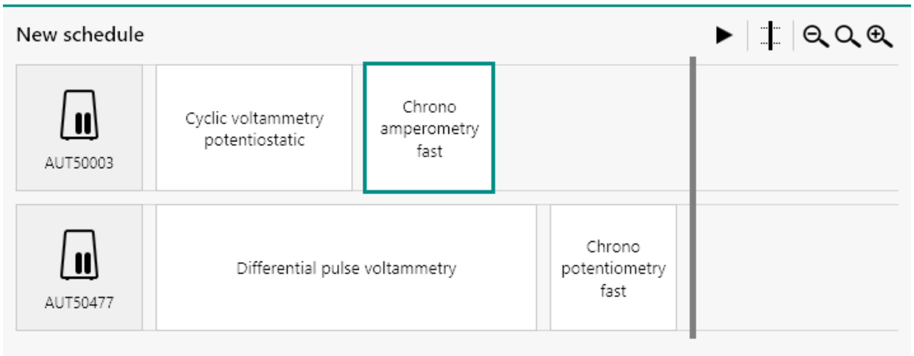


Figure 1028 The synchronization line is added to the schedule

Using the *drag and drop* method, it is possible to relocate the procedures in the schedule on either side of the synchronization line *(see Figure 1029, page 838)*.

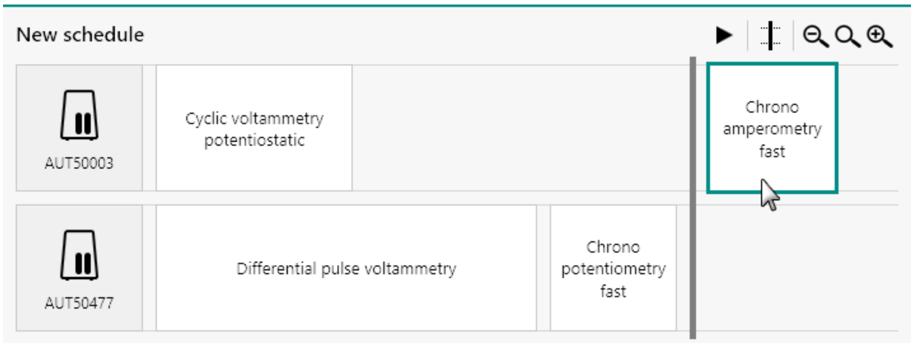


Figure 1029 Relocating procedures with respect to the synchronization line

NOTE

It is also possible to add new procedures the schedule, on either side of the synchronization line.

All procedures located after a synchronization will be synchronized. This means that the all the procedures located on the left hand side of the synchronization line must be finished before starting the next procedure in the schedule. In the example shown in *Figure 1030*, the Chrono amperometry procedure fast and the Chrono potentiometry fast procedure will be synchronized.

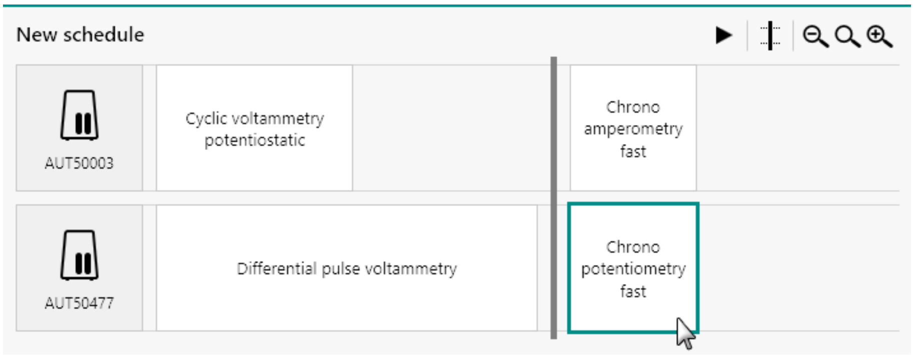


Figure 1030 Creating a synchronized schedule

It is possible to relocate the synchronization line by clicking the line. The synchronization will be highlighted *(see Figure 1031, page 839)*.

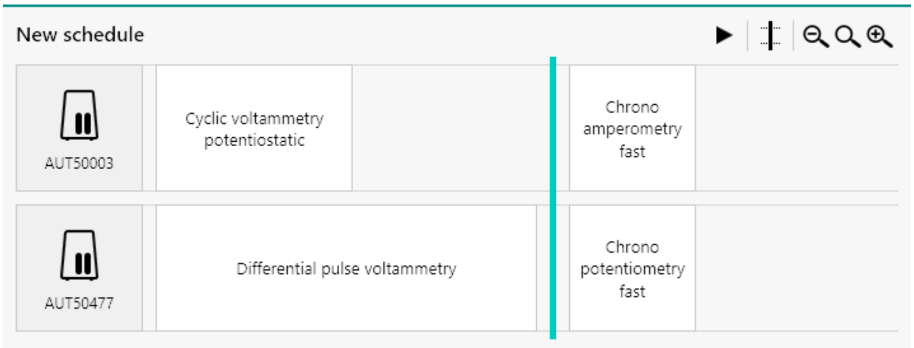


Figure 1031 Selecting the synchronization line

Using the mouse, it is possible to drag the line to the left or the right to adjust its position in the schedule *(see Figure 1032, page 839)*.

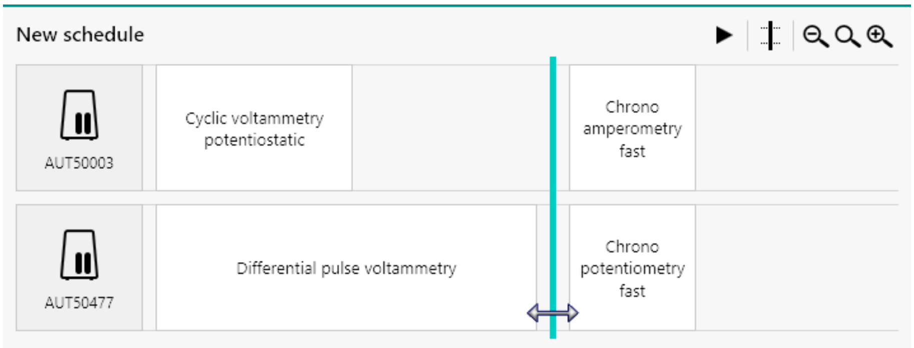


Figure 1032 It is possible to relocate the synchronization line

Clicking the **[Delete]** key, when the synchronization line is selected, will delete the line *(see Figure 1033, page 840)*.

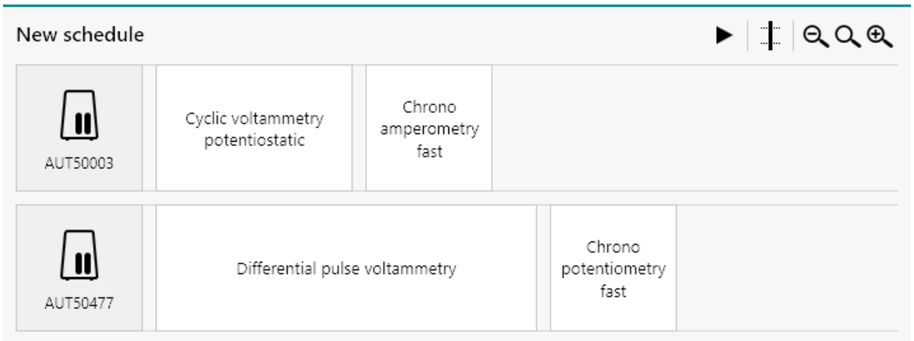


Figure 1033 Pressing the Delete key will delete the synchronization line

NOTE

To synchronize measurements it is also possible to use the **Synchronization** command *(see Chapter 7.2.10, page 240)*.

15.4 Naming and saving the schedule

For bookkeeping purposes, it is possible to provide a name to the schedule and save the schedule. To rename the schedule, click the **New schedule** name in the top left corner of the panel *(see Figure 1034, page 840)*.

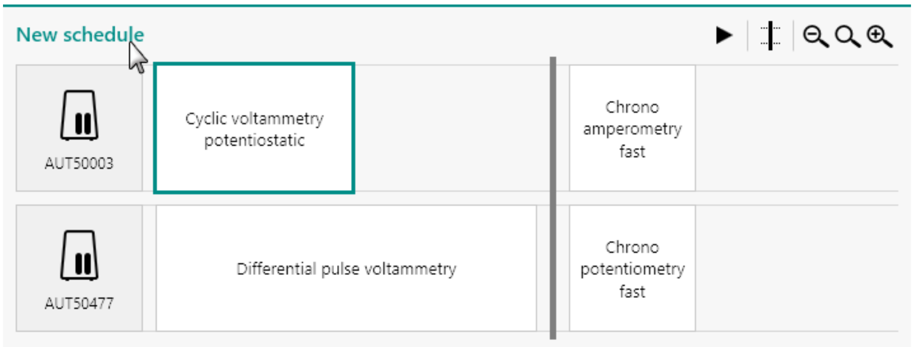


Figure 1034 Renaming the schedule

An input field will be displayed *(see Figure 1035, page 841)*.

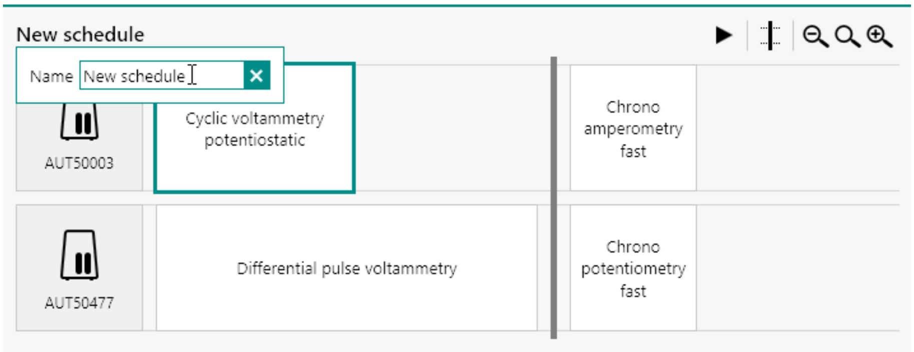


Figure 1035 A new name can be specified

A name of the procedure schedule can be specified *(see Figure 1036, page 841)*.

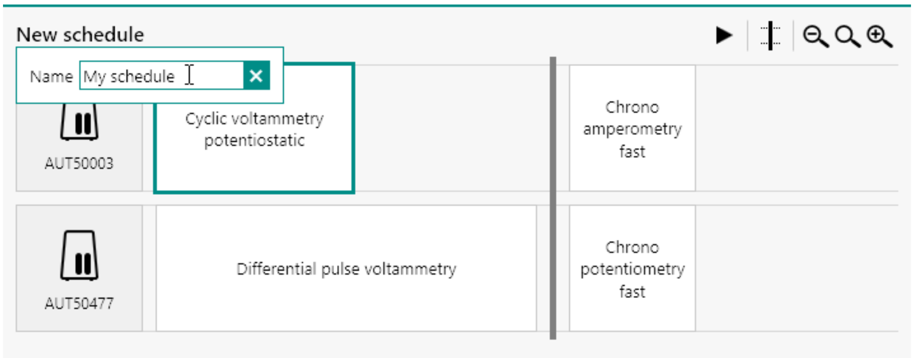


Figure 1036 Specifying the new name of the schedule

Press the **[Enter]** key or click away from the input field to validate the new name of the procedure schedule. The name will be updated in the top left corner of the panel *(see Figure 1037, page 841)*.

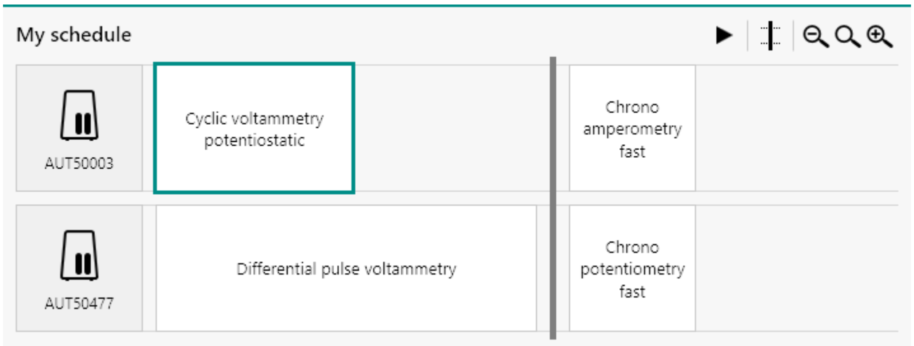


Figure 1037 The schedule name is updated

Once a name has been provided, it is possible to save the schedule by selecting the *Save My schedule* option from the **File** menu, or by using the **[CTRL]** + **[S]** keyboard shortcut *(see Figure 1038, page 842)*.

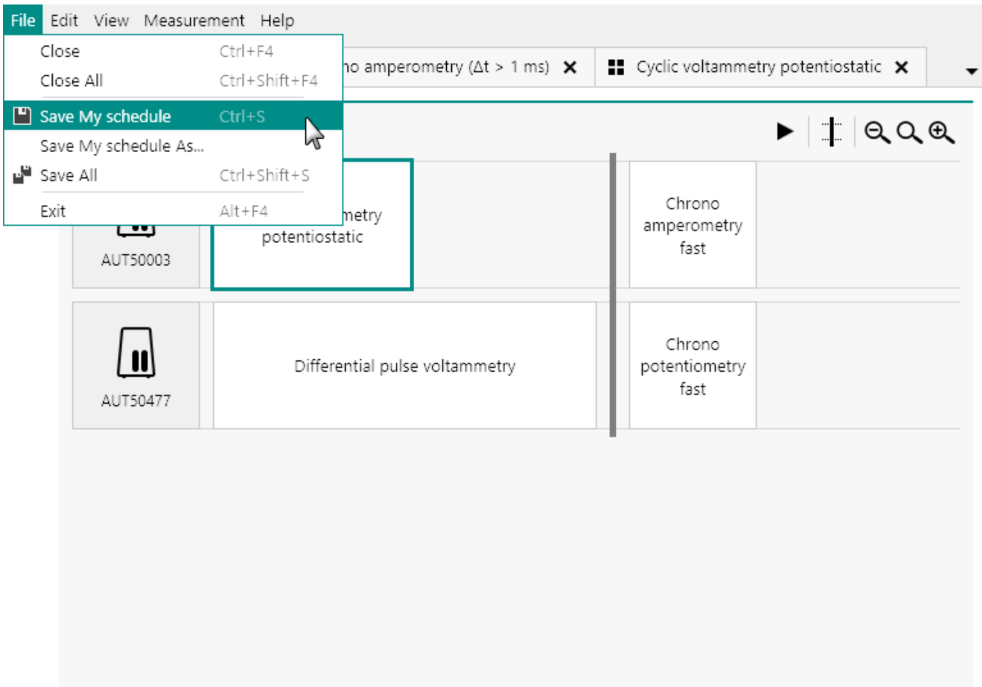


Figure 1038 Saving the schedule

The schedule will be saved in the default Schedules *location* defined in the **Library**. By default, this location is mapped to the **My Documents \NOVA 2.X** folder on the computer. It is also possible to specify the save location of the schedule by using the *Save My schedule As...* option from the **File** menu *(see Figure 1039, page 842)*.

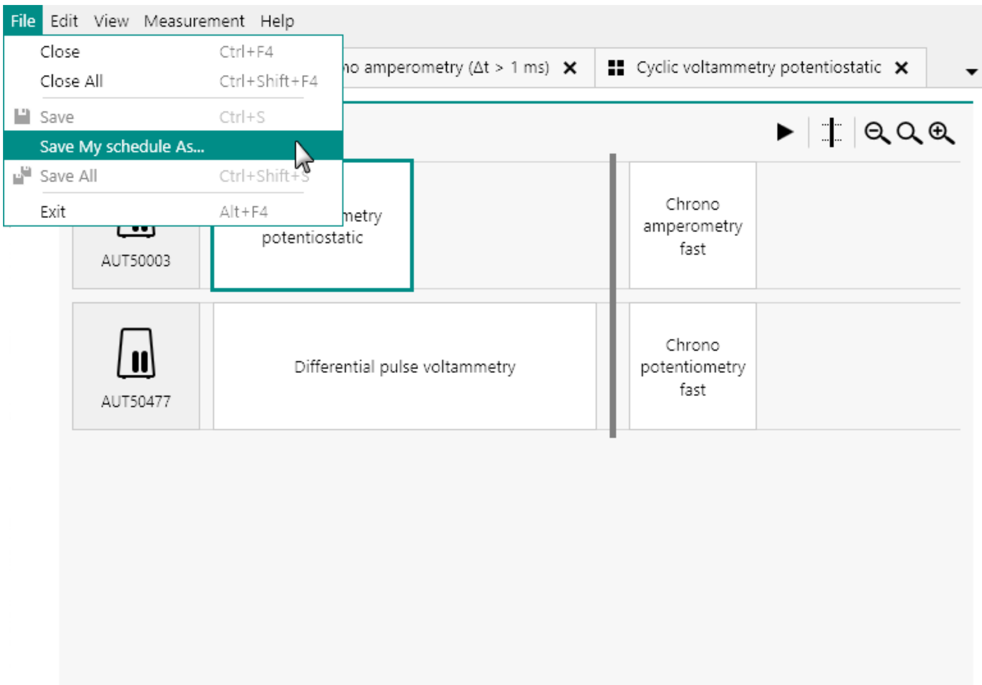


Figure 1039 Saving the schedule in a specific location

A save file dialog will be displayed, providing the means to specify the name and location of the file *(see Figure 1040, page 843)*.

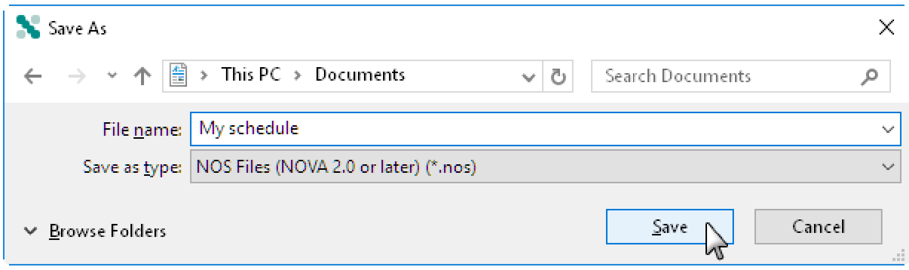


Figure 1040 Specifying the savename and location

A saved schedule can be opened through the **Library** or can be imported into the **Library** by clicking the Import schedule button in the **Actions** panel of the **Dashboard** *(see Figure 1041, page 843)*.

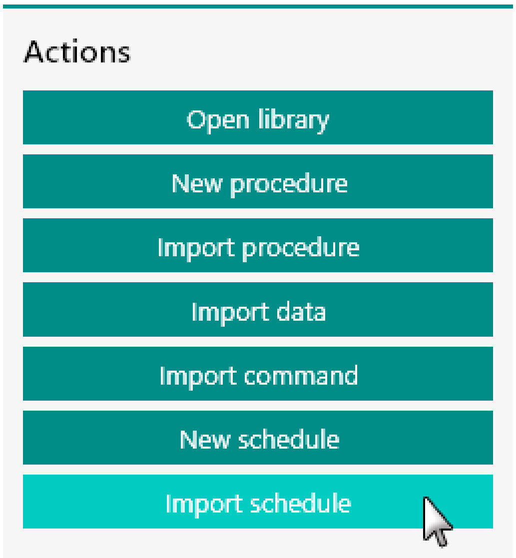


Figure 1041 Opening a new schedule

15.5 Running the schedule

When the procedure schedule is ready, it is possible to start it. It is possible to start the schedule in two different ways:

- Start the complete schedule at once.
- Start the sequence for each instrument sequentially.

Regardless of the method used, the procedure validation will always first check if all the procedures used in the schedule can be executed on the selected instruments. If a warning or error is detected, this will be displayed *(see Figure 1042, page 844)*.

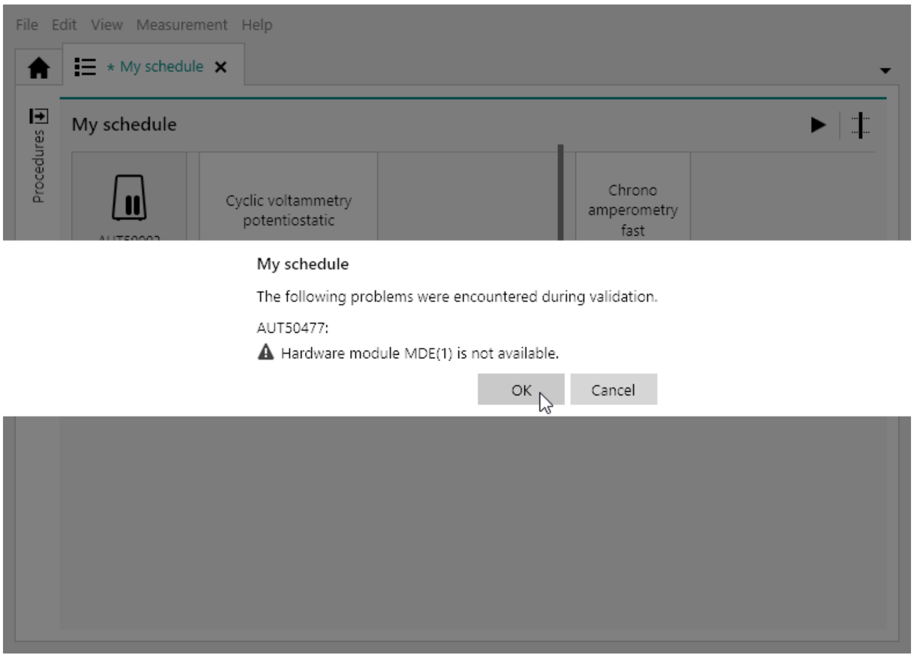


Figure 1042 All the procedures are validated when the procedure schedule is started

If only warnings are displayed, it is possible to proceed with the schedule. If errors are displayed, it is not possible to proceed with the schedule. The errors will first need to be corrected.

15.5.1 Starting the complete procedure schedule

To start the complete procedure schedule, click the ▶ button in the top right corner of the panel *(see Figure 1043, page 844)*.

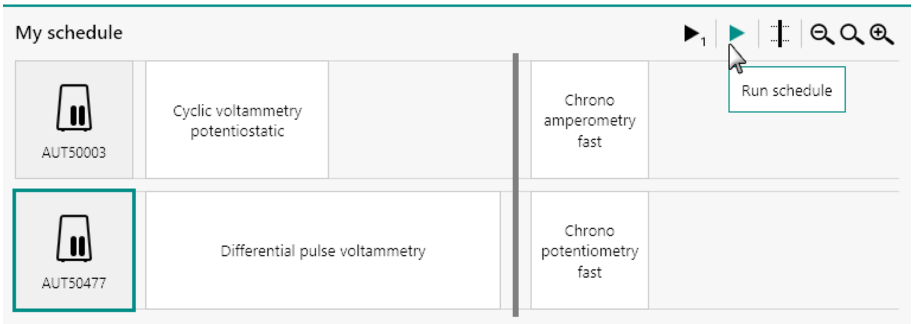


Figure 1043 Starting the complete schedule

After validating the procedures, the schedule will start on the all the instruments specified in the schedule *(see Figure 1044, page 845)*.

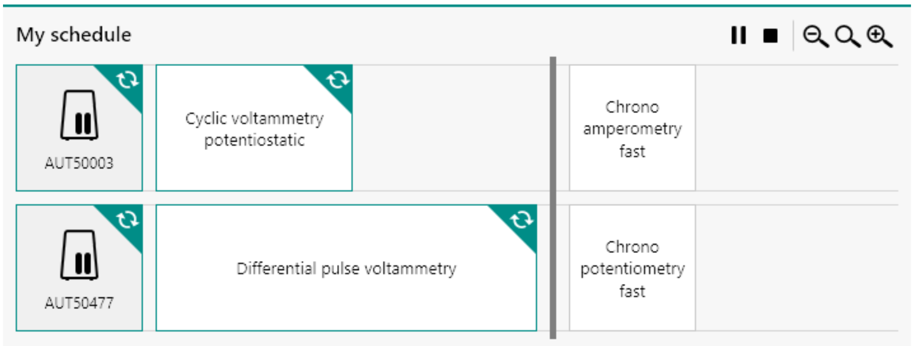


Figure 1044 The procedure schedule is started on all instruments

The procedure schedule will be executed as specified on all instruments.

15.5.2 Starting the schedule sequentially

To start the schedule sequentially, select one of the instruments in the schedule and click the ▶1 button in the top right corner of the panel *(see Figure 1045, page 845)*.

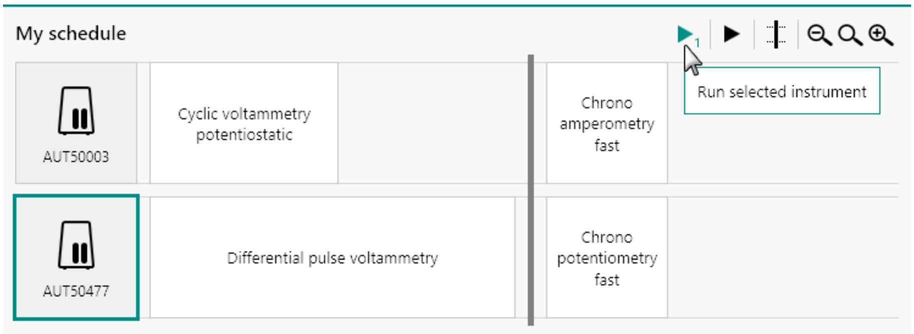


Figure 1045 Starting the procedure schedule for one instrument

NOTE

The ▶1 button is only visible if a single instrument is selected.

The schedule of the selected instrument will start, as shown in *(see Figure 1046, page 846)*. It is possible to repeat this for the other instruments in the procedure schedule *(see Figure 1046, page 846)*.

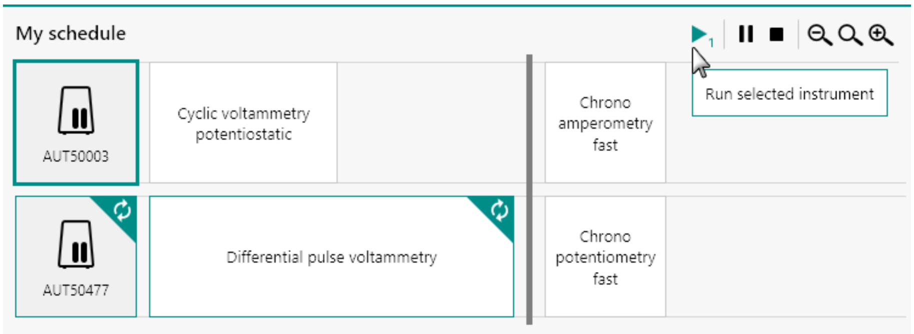


Figure 1046 Starting the procedure schedule of the other instrument

The procedure schedule will start for the other instrument *(see Figure 1047, page 846)*.

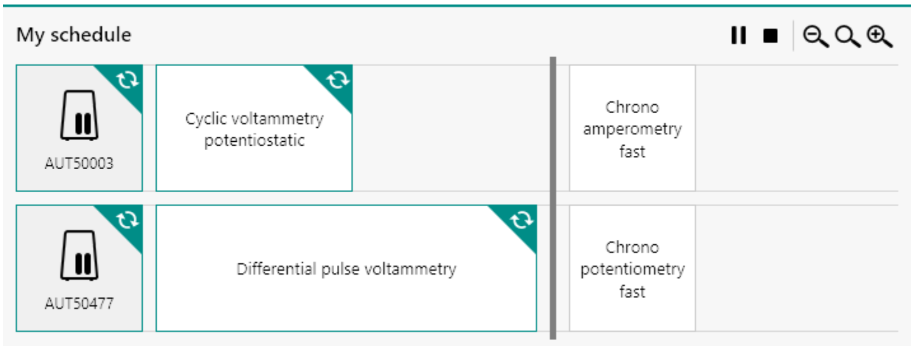


Figure 1047 Both instrument are running

The procedure schedule will be executed as specified on all measuring instruments.

15.5.3 Procedure schedule control

At any point, it is possible to control the procedure schedule. The following actions are possible:

- **Pause one of the instrument:** select one of the measuring instruments and click the $\mathbf{II}_1$ button to pause that instrument. The schedule will be paused for that instrument *(see Figure 1048, page 846)*.

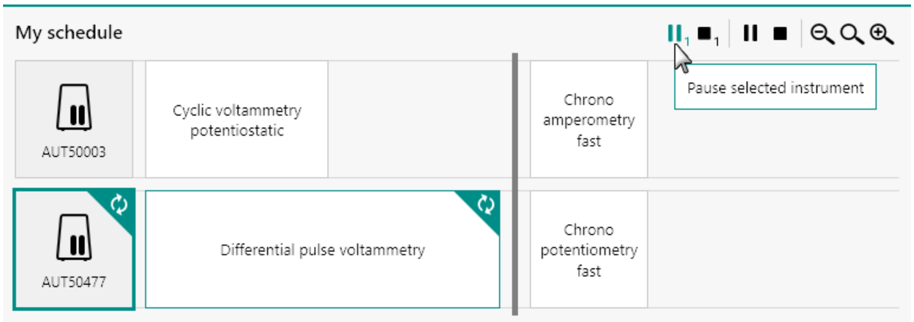


Figure 1048 Pausing one of the instruments

- **Stop one of the instruments:** select one of the measuring instruments and click the ■₁ button to stop that instrument. The schedule will be stopped for that instrument *(see Figure 1049, page 847)*.

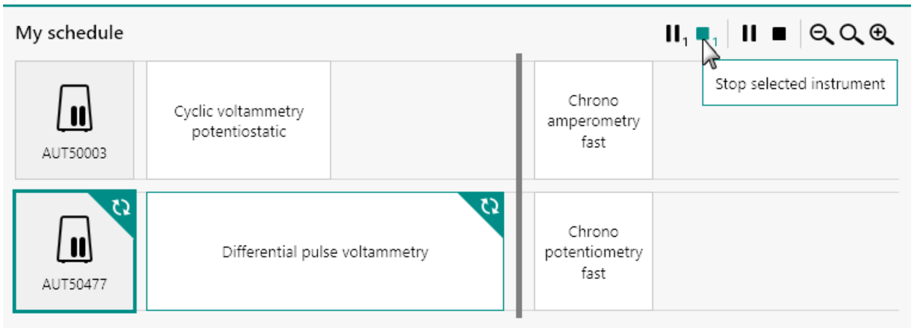


Figure 1049 Stopping one of the instruments

- **Pause all the instruments:** click the ❚❚ button to pause all instruments *(see Figure 1050, page 847)*.

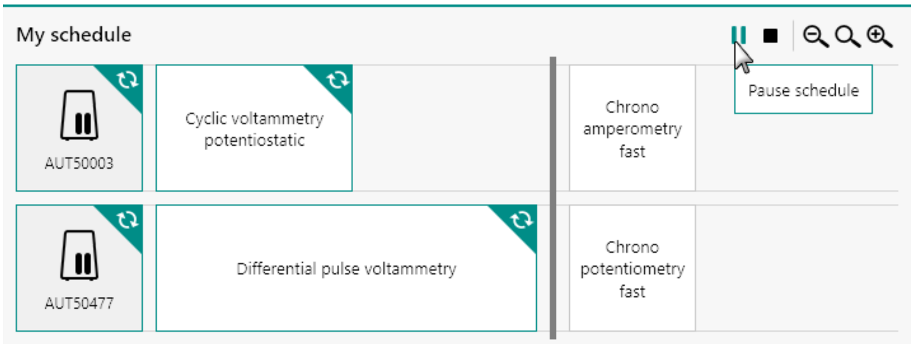


Figure 1050 Pausing all of the instruments

- **Stop all the instruments:** click the ■ button to stop all instruments *(see Figure 1051, page 847)*.

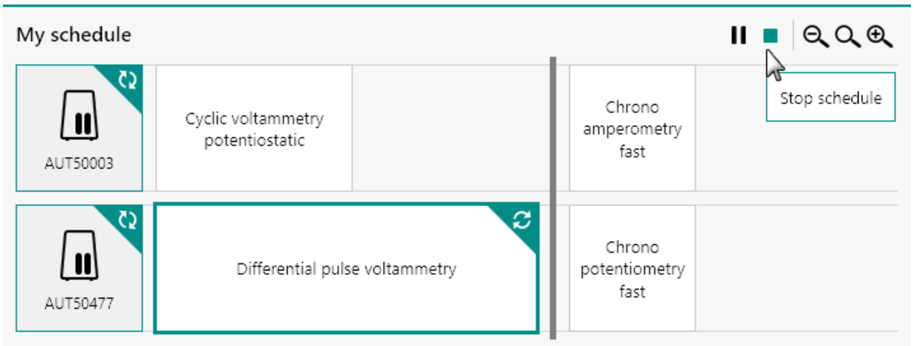


Figure 1051 Stopping all of the instruments

NOTE

Pause measurements can be resumed by clicking the ▶ or ▶₁ buttons in the top right corner of the panel.

15.6 Inspecting procedures or data

At any time, it is possible to inspect and edit a procedure used in a procedure schedule or to inspect data measured by a procedure used in a procedure schedule.

To inspect or edit a procedure, double click on the white box for this procedure in the procedure scheduler *(see Figure 1052, page 848)*.

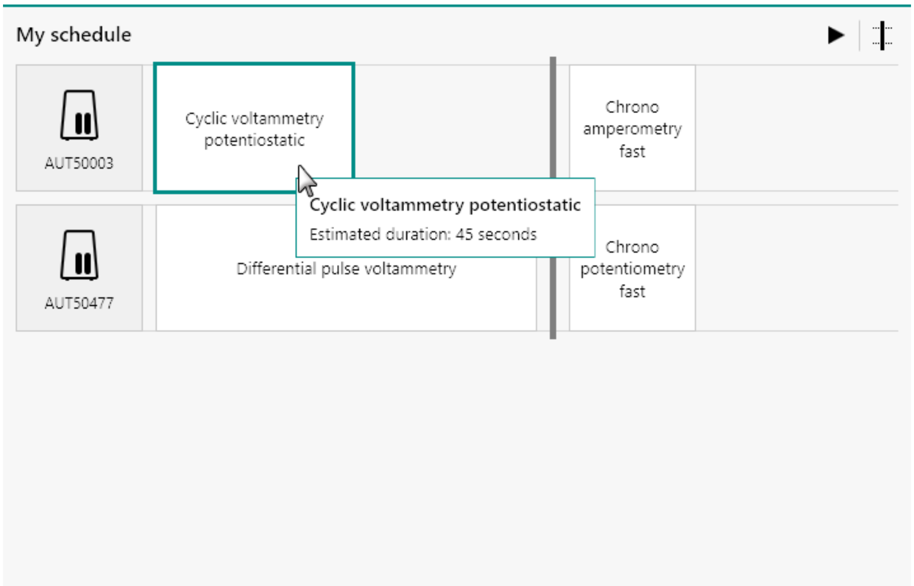


Figure 1052 Double click the procedure to open or edit the procedure

The procedure will be opened in a new tab *(see Figure 1053, page 849)*.

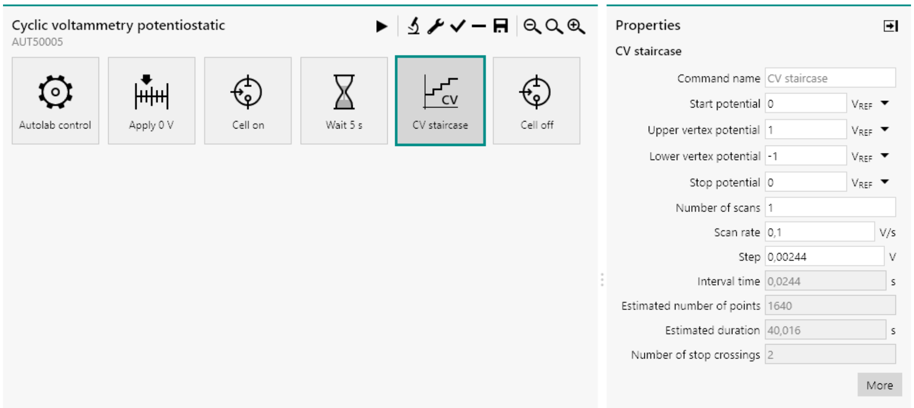


Figure 1053 The procedure is opened in a new tab

The procedure can be edited if required. Modifications that are saved will be automatically carried over to the procedure scheduler.

To inspect data recorded during the procedure scheduler, double click on the white box for a running or finished procedure in the procedure scheduler *(see Figure 1054, page 849)*.

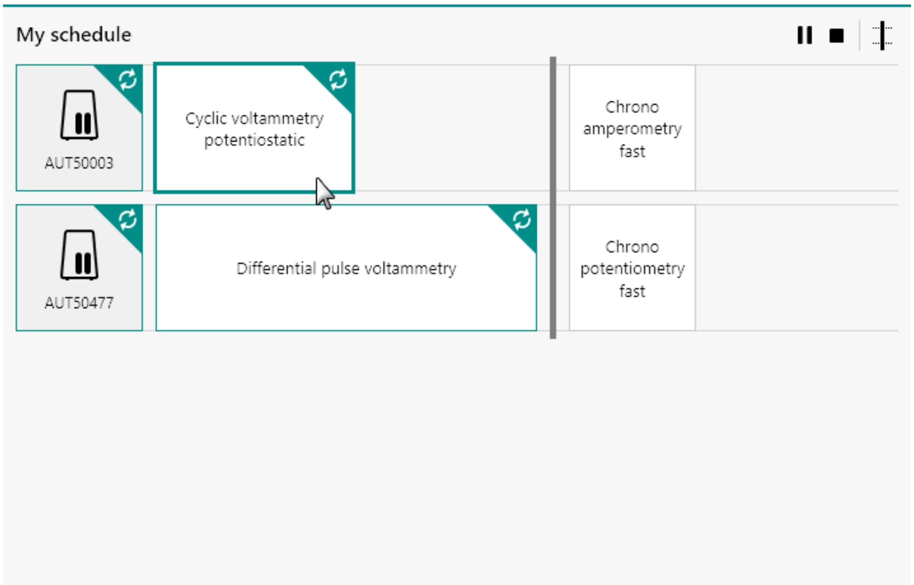


Figure 1054 Click a running or finished procedure to inspect the measured data

The running procedure or the measured data will be opened in a new tab *(see Figure 1055, page 850)*.

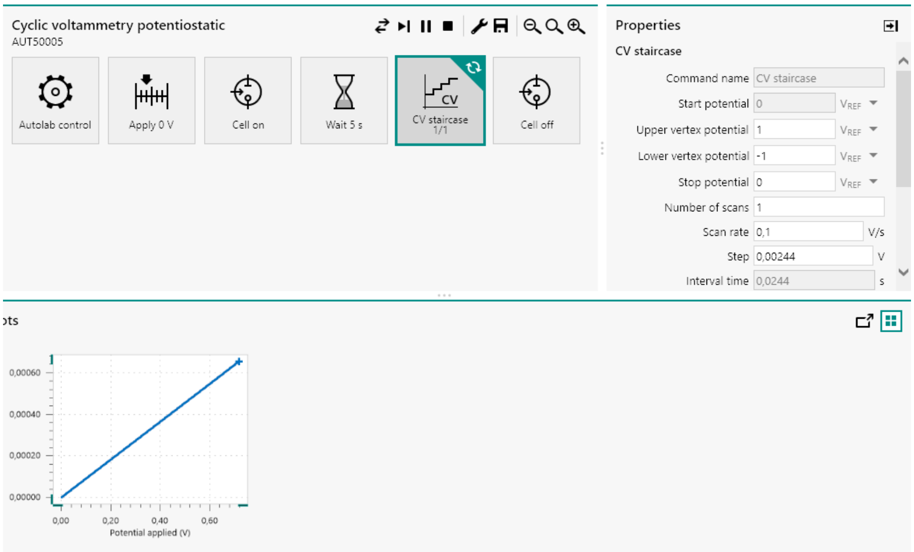


Figure 1055 The data is opened in a new tab

15.7 Schedule zooming

The schedule editor frame has a limited width. If needed, the size of the items in the schedule editor frame can be adjusted with the controls located in the top right corner of the frame *(see Figure 1056, page 850)*.

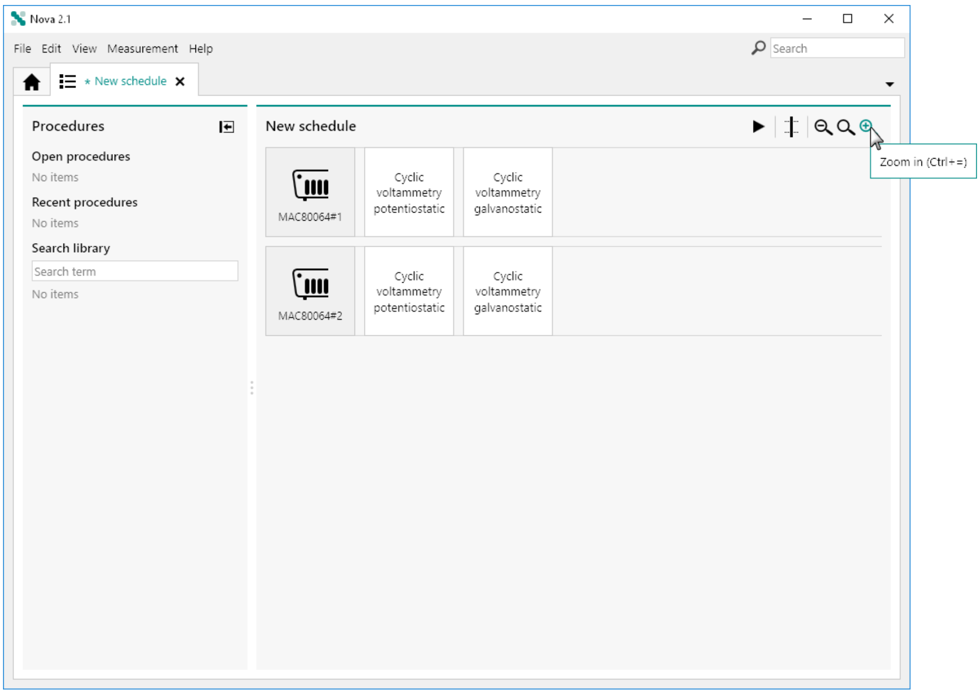


Figure 1056 Zoom controls are provided in the schedule editor

Using this function will either scale the size of the items and the text up or down (between 200 % and 50 % of the original size), as shown in *Figure 1057*.

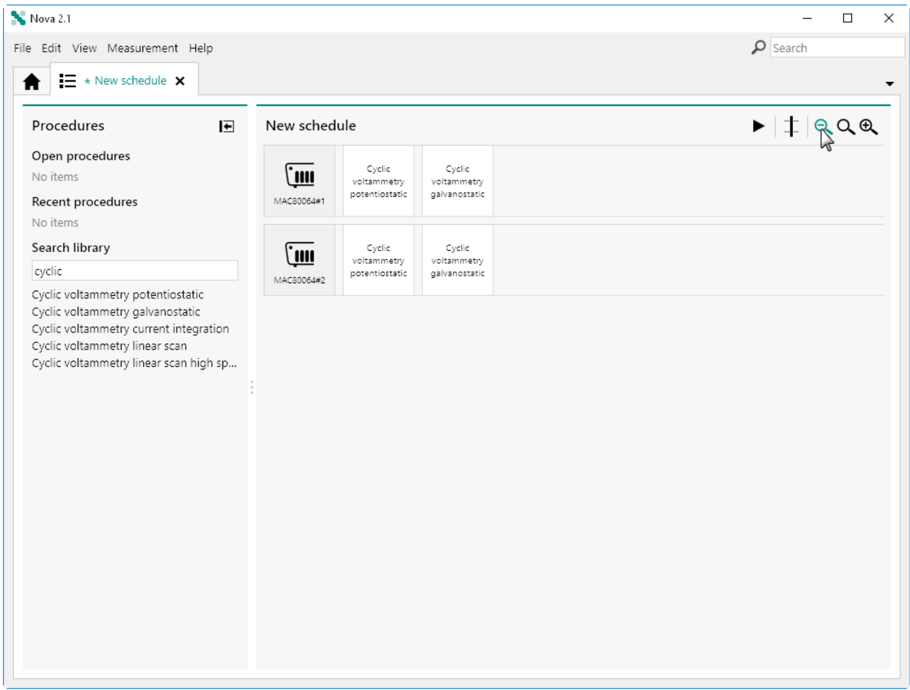


Figure 1057 Zooming the schedule editor out

The following zooming controls are available:

- **Zoom out:** decreases the scaling of the items and text shown on screen. The button or **[CTRL]** + **[-]** keyboard shortcut can be used to do this.
- **Zoom to 100%:** resets the scaling of the items and text shown on screen to the default size. The button or **[F4]** keyboard shortcut can be used to do this.
- **Zoom in:** increases the scaling of the items and text shown on screen. The button or **[CTRL]** + **[=]** keyboard shortcut can be used to do this.

16 Hardware description

This chapter provides extended information on the Autolab hardware, extension modules and accessories.

Table 14 provides a list of the currently available instruments and modules, supported in NOVA.

Table 14 Overview of the available and supported instruments and modules

Instruments	Modules
PGSTAT100N	ADC10M
PGSTAT128N	BA
PGSTAT302N	Booster10A
PGSTAT302F	Booster20A
PGSTAT101	ECD
PGSTAT204	ECI10M
Multi Autolab M101	ECN
Multi Autolab M204	EQCM
PGSTAT302N MBA	FI20
PGSTAT128N MBA	IME303
	IME663
	MUX
	pX1000
	SCAN250

Table 15 provides a list of the phased out instruments and modules, supported in NOVA.

Table 15 Overview of the phased out and supported instruments and modules

Instruments	Modules
PGSTAT10	ADC750 (replaced by ADC10M)
PGSTAT12	ARRAY (replaced by BA)
PGSTAT20	BIPOT (replaced by BA)
PGSTAT30	FRA2 (replaced by FRA32M)

Instruments	Modules
PGSTAT100	pX (replaced by pX1000)
PGSTAT100 floating option	SCANGEN (replaced by SCAN250)
PGSTAT302	
µAutolab II	
µAutolab III	

Table 16 provides a list of the legacy instruments and modules, unsupported in NOVA.

Table 16 Overview of the legacy and unsupported instruments and modules

Instruments	Modules
PSTAT10	ADC124
µAutolab	DAC124
Multi Autolab PSTAT10	DAC168
	FRA

16.1 General considerations on the use of the Autolab potentiostat/galvanostat systems

This chapter provides general information on the use of the Autolab potentiostat/galvanostat. The information provided in this chapter applies to all instrument, unless otherwise specified. It is highly recommended to review this information before using the Autolab potentiostat/galvanostat.

16.1.1 Electrode connections

The Autolab instruments are supplied with cell cables providing connections for **three** or **four** electrodes, depending on the type of instrument. The electrode connections are provided through 4 mm male banana connectors.

These electrode connections are labeled as follows:

- **Working (or indicator electrode):** WE (red)
- **Sense electrode:** S (red)
- **Reference electrode:** RE (blue)
- **Counter electrode:** CE (black)

An additional green ground connector is provided for connections to a Faraday cage.

> **NOTE**
>
> The µAutolab type II, µAutolab type III, PGSTAT10 and PGSTAT10 are not fitted with the Sense electrode.

16.1.1.1 Three electrode connections

Instruments with three electrode connectors can be connected to the electrochemical cell in two different ways:

- **Two electrode mode:** in this mode, the counter electrode (CE) and reference electrode (RE) are connected together to one electrode while the working electrode (WE) is connected to the other electrode. The current is measured between the CE and the WE and the potential difference is measured between the RE and the WE. This mode is commonly used for the characterization of energy storage and conversion devices like batteries, fuel cells, solar cells and supercapacitors.

> **NOTE**
>
> For high current applications it is highly recommended to separately connect the RE and CE to the same electrode. Furthermore, it is recommended to place the RE as close as possible to the electrodes in the cell. This will reduce ohmic losses coming from the connections.

- **Three electrode mode:** in this mode, the counter electrode (CE) and reference electrode (RE) are connected to a counter and reference electrode, respectively. The working electrode (WE) is connected to the working electrode. The current is measured between the CE and the WE and the potential difference is measured between the RE and the WE. This mode is commonly used for the characterization most electrochemical cells in which a separate reference electrode is used.

NOTE

It is common practice to place the reference electrode as close as possible to the working electrode to reduce the uncompensated resistance and reduce the ohmic losses arising from this resistance. This can be achieved by physically placing the reference electrode close to the working electrode or by using a *Luggin-Haber* capillary.

16.1.1.2 Four electrode connections

Instruments with four electrode connectors can be connected to the electrochemical cell in three different ways:

- **Two electrode mode:** in this mode, the counter electrode (CE) and reference electrode (RE) are connected together to one electrode while the working electrode (WE) and sense electrode (S) are connected to the other electrode. The current is measured between the CE and the WE and the potential difference is measured between the RE and the S. This mode is commonly used for the characterization of energy storage and conversion devices like batteries, fuel cells, solar cells and supercapacitors.

NOTE

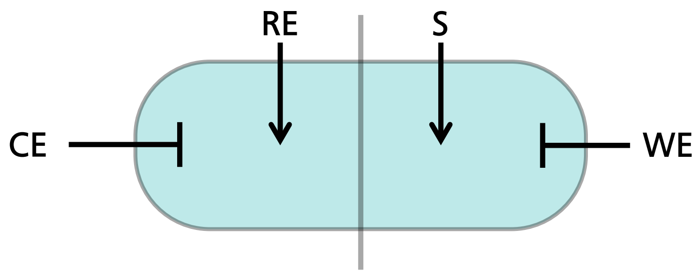
For high current applications it is highly recommended to separately connect the RE and CE to the same electrode and to do the same with the WE and S on the other electrode. Furthermore, it is recommended to place the RE and S as close as possible to the electrodes in the cell. This will reduce ohmic losses coming from the connections.

- **Three electrode mode:** in this mode, the counter electrode (CE) and reference electrode (RE) are connected to a counter and reference electrode, respectively. The working electrode (WE) and sense electrode (S) are connected to the working electrode. The current is measured between the CE and the WE and the potential difference is measured between the RE and the S. This mode is commonly used for the characterization most electrochemical cells in which a separate reference electrode is used.

NOTE

It is common practice to place the reference electrode as close as possible to the working electrode to reduce the uncompensated resistance and reduce the ohmic losses arising from this resistance. This can be achieved by physically placing the reference electrode close to the working electrode or by using a *Luggin-Haber* capillary.

- **Four electrode mode:** in this mode, the counter electrode (CE) and reference electrode (RE) are connected to a counter and reference electrode, respectively on side of the electrochemical cell. The working electrode (WE) and sense electrode (S) are connected to a second set of working electrode and reference electrode on the other side of the electrochemical cell. Both sides of the cell are separated by a membrane or by using non miscible solvent. The current is measured between the CE and the WE and the potential difference is measured between the RE and the S. This mode is commonly used for the characterization of the liquid-liquid interface.



16.1.2 Operating principles of the Autolab PGSTAT

The Autolab instrument combined with the software is a computer-controlled electrochemical measurement system. It consists of a data-acquisition system and a potentiostat/galvanostat. The basic working principle is schematically represented in *Figure 1058*.

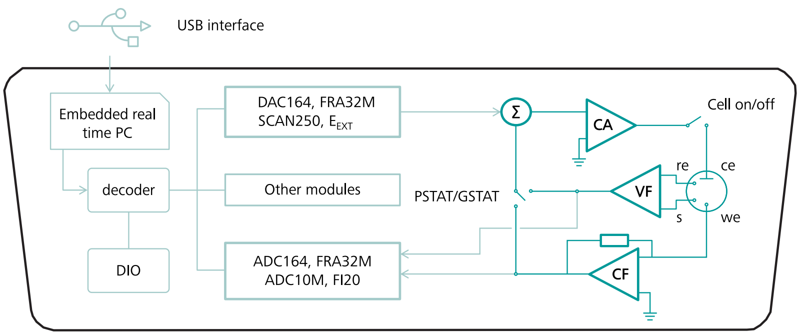


Figure 1058 Schematic representation of the Autolab potentiostat/ galvanostat

The Autolab system is fitted with the following common **digital** control components:

- **USB interface:** the interface between the Autolab and the host computer.
- **Embedded PC with real-time operating system:** a dedicated controller embedded into the instrument, which is responsible for timing and interfacing the host application and the instrument controls.
- **Decoder and DIO:** a data decoder and digital input/output interface.

The digital components are interfaced through the Autolab modules to the **analog** potentiostat/galvanostat circuit. The latter consists of the following components:

- **Summation point (Σ):** a circuit used to add the control signals required to generate the waveform used in electrochemical measurements.
- **Control amplifier (CA):** a circuit used to amplify the output of the summation point.
- **Voltage follower (VF):** a circuit used to measure the potential.
- **Current follower (CF):** a circuit used to measure the current.

The arrangement of these analog circuits with respect to the electrochemical cell are represented in *Figure 1059* for a four electrode Autolab system and in *Figure 1060* for a three electrode Autolab system.

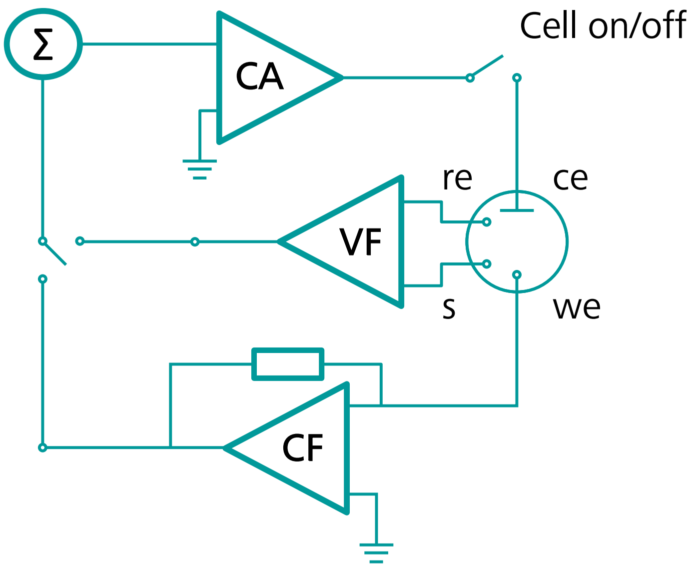


Figure 1059 Schematic representation of the analog circuits of the Autolab in a four electrode system

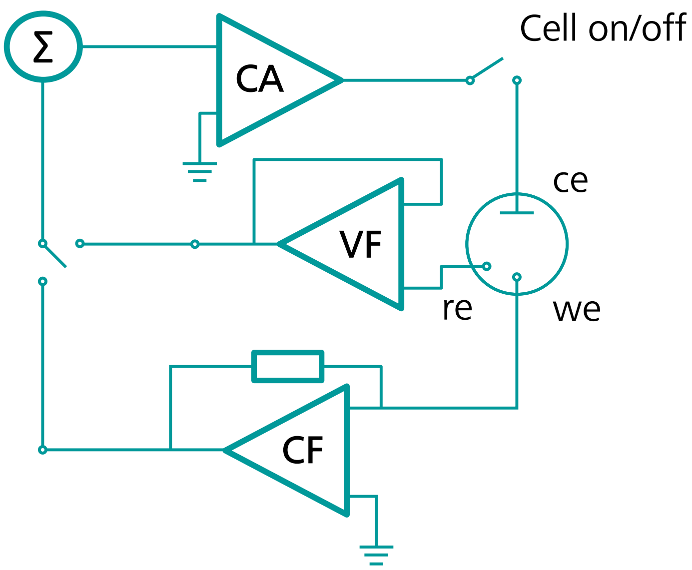


Figure 1060 Schematic representation of the analog circuits of the Autolab in a three electrode system

The summation point (Σ) is an **adder** circuit that feeds the input of the control amplifier (**CA**). Each of the inputs of the summation point is divided by a hardware-defined value *(see Figure 1061, page 859)*.

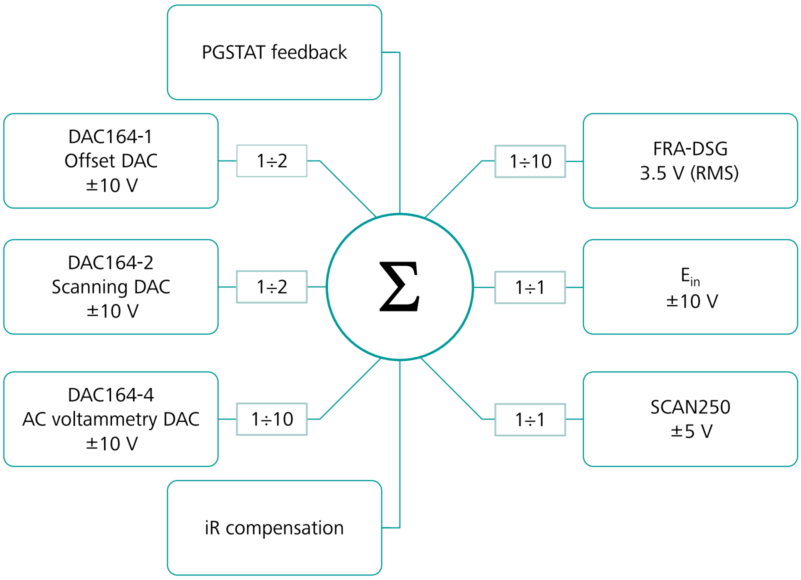


Figure 1061 Schematic representation of the summation point of the Autolab

It is connected to the output of the several key modules of the Autolab:

- **DAC164 (or on-board DAC):** the digital-to-analog converters of the Autolab. Depending on the type of instrument, the following DAC inputs are available:
  - ***Offset DAC:*** used to generate an offset. This signal is divided by **2**.
  - ***Scanning DAC:*** used to generate steps and scans. This signal is divided by **2**.
  - ***AC voltammetry DAC:*** used for AC voltammetry only. This signal is divided by **10**.
- **FRA32M DSG or FRA2 DSG:** the digital waveform generator of the *optional* **FRA32M** or **FRA2** module *(see Chapter 16.3.2.13, page 1091)*. This signal is divided by **10**.
- **SCAN250 or SCANGEN output:** the analog scan output of the *optional* linear **SCAN250** or **SCANGEN** module *(see Chapter 16.3.2.19, page 1148)*. This signal is divided by **1**.
- **E_{in}:** the external input provided through the monitor cable. This signal is divided by **1**.
- **PGSTAT feedback:** the feedback from the voltage follower (**VF**), in potentiostatic mode or the feedback from the current follower (**CF**), in galvanostatic mode.
- **iR compensation feedback:** the feedback from the iR compensation circuit, when in use in potentiostatic mode.

CAUTION

Some of the summation point inputs are not available on all Autolab instruments.

The control amplifier provides the output voltage on the counter electrode (CE) with respect to the working electrode (WE) required to keep the potential difference between the reference electrode (RE) and the sense (S) or the potential difference between the reference electrode (RE) and the working electrode (WE) at the user defined value, in potentiostatic mode, or the user required current between the counter electrode (CE) and the working electrode (WE) in galvanostatic mode.

The output of the control amplifier can be manually or remotely disconnected from the electrochemical cell through a cell ON/OFF switch. The voltage follower (**VF**) is used to measure the potential difference between the reference electrode and the sense and the current follower (**CF**). The current follower has several current ranges providing different current-to-voltage conversion factors.

The output of the current follower (**CF**) and the voltage follower (**VF**) are fed back to the analog-to-digital converter modules of the Autolab:

- **ADC164 (or on-board ADC):** general purpose analog-to-digital converter of the Autolab instrument.
- **FRA32M ADCs or FRA2 ADCs:** two synchronized analog-to-digital converters located on the *optional* **FRA32M** or **FRA2** module used for impedance spectroscopy measurements *(see Chapter 16.3.2.13, page 1091)*.
- **ADC10M or ADC750:** two synchronized analog-to-digital converters located on the *optional* **ADC10M** or **ADC750** module for ultra-fast sampling *(see Chapter 16.3.2.1, page 977)*.
- **FI20 (or on-board integrator):** an *optional* filter and integrator module (**FI20**) or on-board integrator which can be used to convert the current to charge and filter the current signal *(see Chapter 16.3.2.11, page 1061)*.

Furthermore, the output of the voltage follower (VF) or the current follower (**CF**) is fed back to the summation point to close the feedback loop in potentiostatic or galvanostatic mode, respectively.

The **ADC164 (or on-board ADC)** provides the possibility of measuring analog signals. The input sensitivity is software-controlled, with ranges of ±10 V (gain 1), ±1 V (gain 10) and ±0.1 V (gain 100). The resolution of the measurement is 1 in 65536 (16 bits). Analog signals can be measured with a rate of up to 60 kHz. The ADC164 is used to measure the output of the voltage follower (**VF**) and current follower (**CF**) of the potentiostat/galvanostat.

The **DAC164 (or on-board)** generates analog output signals. The output is software-controlled within a range of ±10 V. The resolution of the DAC164 is 1 in 65536 (300 µV). In the Autolab PGSTAT two channels of the DAC (*Scanning DAC* and *Offset DAC*) are used to control the analog input signal of the potentiostat/galvanostat. The µAutolab type II and µAutolab type III only uses a *Scanning DAC* to control the analog input. The values of the DACs are added up in the potentiostat and divided by 2. This results in an output of ±10 V with a resolution of 150 µV.

In practice this means that the potential range available with the Autolab PGSTAT during an electrochemical experiment is ±5 V with respect to the offset potential generated by the *Offset DAC*. The available potential range is therefore -10 V to 10 V with the Autolab PGSTAT and -5 V to 5 V with the µAutolab type II and µAutolab type III.

The AC voltammetry DAC, if present, is hardwired to the summation point and it is divided by 10. This input is used for measurements involving a small amplitude modulation (like AC voltammetry).

16.1.2.1 Event timing

The embedded controller of the Autolab is equipped with a 1 MHz timer that is used by the software to control the timing of events during measurements. The shortest interval time on the embedded controller is 1 µs. When a procedure is started in NOVA, the procedure is first uploaded from the host PC to the embedded PC, through the USB connection. The measurement can then be started.

Depending on the type of command that NOVA encounters during the measurements two timing protocols are used:

1. **Real-time control:** all measurement commands in NOVA are timed using the embedded processor timer. Whenever NOVA encounters a measurement command, it will be executed using the timing provided by the embedded computer of the Autolab. If several measurement commands are located in sequence, the sequence is executed without interruption. This ensures that the measurement commands in the sequence are executed with the smallest possible time gap. The actual time difference between two consecutive commands depends on the hardware changes required during the transition between the two commands. Switching current ranges or using the cell switch are time consuming steps since they involve mechanical relays which require a fixed settling time. Taking into account these hardware defined interval times, the effective time gap between two consecutive measurement commands will always be ≤ 10 ms.

2. **Host computer control:** all the other commands in NOVA are timed using the timer of the host computer. Since the host computer is also involved in other Windows activity, accurate timing of events cannot be guaranteed and the effective interval time between two consecutive host commands will depend entirely on the amount of activity on the host computer. Depending on the command sequence, the time gap can be as short as ~ 1 s (transition between host command to measurement command) or several seconds (transition between measurement command and host command). Transfer of large amounts of measured data points is particularly time consuming.

NOTE

To reduce the time gap between commands timed by the host computer it is recommended to close all unnecessary Windows applications while using NOVA.

16.1.2.2 Consequences of the digital base of the Autolab

The digital nature of the instrument control has consequences for the measurements. The consequences for the different techniques are the following:

- The minimum potential step or pulse in all techniques is 150 µV (16 Bit DAC).
- All potential steps are rounded up or down to the nearest possible multiple of 150 µV.
- In staircase potential (or current) scans, the interval time, Δt, or time between two consecutive steps is given by:

$$\Delta t = \frac{E_{step}}{\vec{v}}$$

Where E_{step} is the potential step (or current step) and $\vec{v}$ is the scan rate.

The response of the electrochemical cell is recorded digitally. Therefore the resolution of the measurements is also limited. The actual resolution depends on the technique and on the amplitude of the signal. Since the analog-to-digital converter is equipped with a software programmable amplifier, the absolute resolution depends on the gain of the amplifier. The gains used are 1, 10 and 100 times the input signal.

NOVA automatically selects the best possible gain during a measurement. Gain 10 and 100 are used when the signal is small enough.

Depending on the gain, the resolution for potential, current and external analog signal are listed in *Table 17*.

Table 17 The resolution of the measurable signals

Signal	Potential	Current ([CR] active current range)	External
Gain 1	$\frac{20\,V}{2^{16}\cdot 1}$	$\frac{20[CR]}{2^{16}\cdot 1}$	$\frac{20\,V}{2^{16}\cdot 1}$
Gain 10	$\frac{20\,V}{2^{16}\cdot 10}$	$\frac{20[CR]}{2^{16}\cdot 10}$	$\frac{20\,V}{2^{16}\cdot 10}$
Gain 100	$\frac{20\,V}{2^{16}\cdot 100}$	$\frac{20[CR]}{2^{16}\cdot 100}$	$\frac{20\,V}{2^{16}\cdot 100}$

The effect of the limited resolution can be seen, for instance when low currents are measured at a high current range. In such cases a lower current range has to be applied, if possible. When automatic current ranging is used, the most suitable current range is selected automatically.

Care must be taken when using this option in the following situations:

- Square wave voltammetry measurements at high frequency.
- Cyclic and linear sweep voltammetry measurements at high scan rates.

Switching of the current range takes about 0.5 ms to 2 ms. Therefore an erroneous point can be measured when the current range is switched. Most of the time, this error can be corrected by smoothing the plot afterwards.

16.1.2.3 Bandwidth settings

The control amplifier of the Autolab is equipped with three different bandwidth settings:

- High stability
- High speed
- Ultra-high speed

NOTE

The Ultra-high speed mode is not available for the PGSTAT302F, the PGSTAT10, the PGSTAT20, the µAutolab type II and µAutolab type III.

The bandwidth setting can be specified using the **Autolab control** command *(see Chapter 7.2.1, page 221)*.

This property defines the bandwidth of the instrument control amplifier. The three settings provided by the Autolab control command can be used to reach the required bandwidth while maintaining stability of the potentiostatic or galvanostatic control loop. The normal mode of operation is **High stability** *(see Figure 1062, page 864)*.

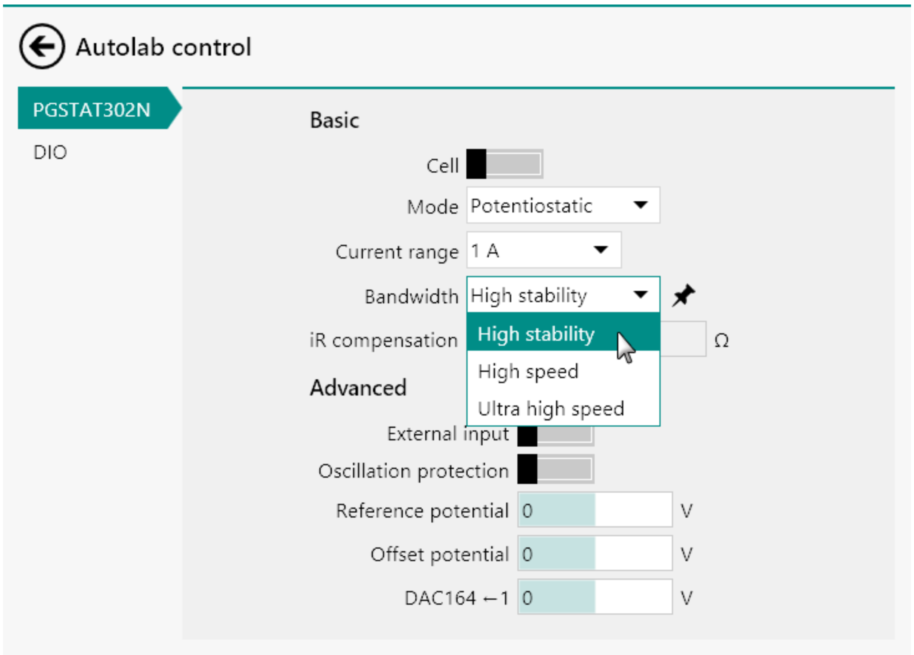


Figure 1062 The instrument bandwidth is defined in the Autolab control command

It is also possible to define the bandwidth using the **Autolab display** panel *(see Figure 1063, page 865)*.

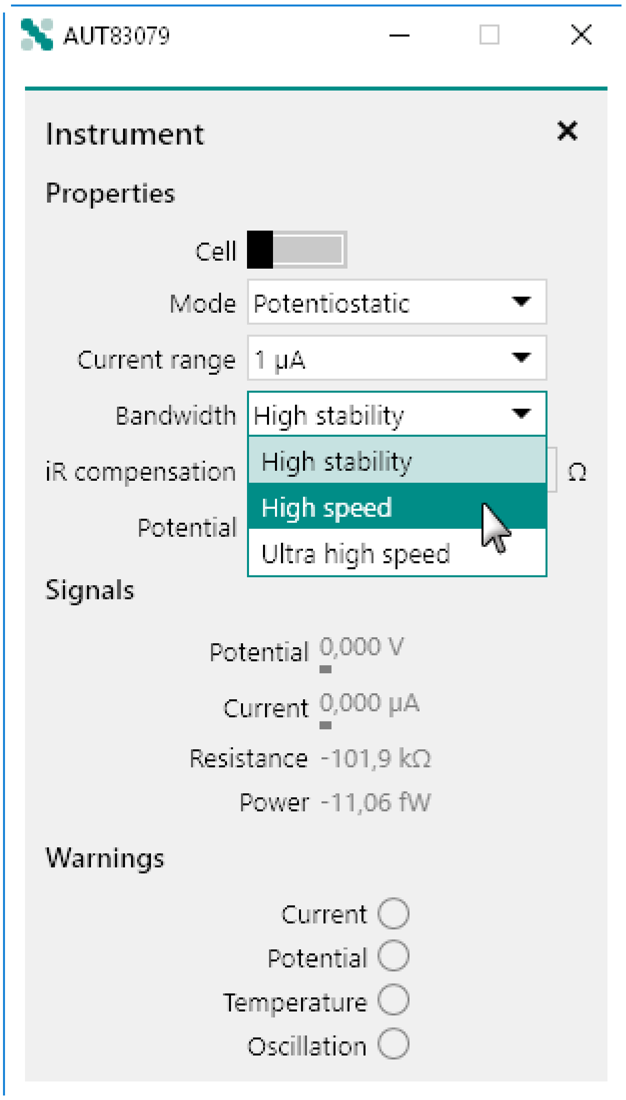


Figure 1063 The instrument bandwidth can be specified directly from the Autolab display panel

NOTE

The **High stability** mode is the power-up default of the instrument

The **High stability** mode is suitable for electrochemical measurements carried out at low frequencies or low scan rates. In this mode of operation, the instrument control amplifier will use the slowest possible feedback. This usually ensures a stable control loop and low noise levels on the measured potential and current. In this mode, the bandwidth of the control amplifier is limited to 10 kHz.

Faster measurements may require a higher bandwidth setting. In **High speed** mode, the control amplifier bandwidth is extended to 125 kHz while in **Ultra-high speed** mode, the control amplifier bandwidth is extended to 1.25 MHz (for the PGSTAT302N) and to 500 kHz (for the PGSTAT128N and PGSTAT100N).

With these settings, the risk of oscillations is higher than in **High stability** mode. This is especially the case with electrochemical cells exhibiting a high capacitance. There is a significant oscillation risk in these modes of operation. Additionally, the noise in the measured potential and current signals will be higher than in **High stability** mode.

NOTE

The specified or active bandwidth setting is not changed by any measurement command **except** the FRA measurement command which automatically selects the most suitable bandwidth setting in function of the applied frequency.

CAUTION

The higher the bandwidth, the higher the noise and probability of oscillation. When working with a high bandwidth setting (**High speed** or **Ultra high speed**), it is necessary to pay attention to adequate shielding of the cell and electrode connectors. The use of a Faraday cage is recommended in these cases.

16.1.2.3.1 Input impedance and stability

The voltage follower (**VF**) input contains a small capacitive load. If the capacitive part of the impedance between CE and RE is comparatively large, phase shifts will occur which can lead to instability problems when working in potentiostatic mode. If the impedance between the CE and the RE cannot be changed and oscillations are observed, it is recommended to select the **High stability** mode to increase the system stability.

In general, the use of **High stability** leads to a more stable control loop, compared to **High speed** or **Ultra-high speed** and a significantly lower bandwidth.

To make use of the full potentiostat bandwidth (**Ultra-high speed** mode), the impedance between CE and RE has to be lower than 35 kΩ. This value is derived by testing. In galvanostat mode, this large impedance between CE and RE, will usually not lead to stability problems, because of the current feedback regulation.

16.1.2.3.2 Galvanostat and iR compensation bandwidth limitations

For galvanostatic measurements on low current ranges, the bandwidth limiting factor becomes the current follower (**CF**) rather than the control amplifier. The same applies to potentiostatic measurements with the iR compensation circuit on. In both cases, the bandwidth of this circuit directly determines the maximum bandwidth of the control loop.

When the iR compensation circuit is not used in potentiostatic mode, the bandwidth limitation of the current follower (**CF**) does not directly influence the control loop bandwidth but it influences the measurement of the current signal.

For stability reasons, the following guidelines are provided when working in galvanostatic mode or in potentiostatic mode with the iR compensation circuit on:

- The use of the **High speed** mode is only recommended for current range of 10 µA and higher.
- The use of the **Ultra-high speed** mode is only recommended for current ranges of 1 mA and higher.

A general indication of the maximum available bandwidth for galvanostatic measurements and measurements with the iR compensation circuit on can be found in:

- *Table 18* for the N Series Autolab instruments.
- *Table 19* for the 7 Series Autolab instruments.
- *Table 20* for the PGSTAT101, PGSTAT204, M101 and M204.
- *Table 21* for the µAutolab type II and µAutolab type III, PGSTAT10 and PGSTAT20.

Table 18 Bandwidth overview of the N Series Autolab instruments

Instrument	**PGSTAT128N, PGSTAT100N**		**PGSTAT302N**	
Mode	**GSTAT**	**iR compensation**	**GSTAT**	**iR compensation**
1 A - 1 mA	> 500 kHz	> 500 kHz	> 1.25 MHz	> 1.25 MHz
100 µA	125 kHz	500 kHz	125 kHz	1 MHz
10 µA	100 kHz	100 kHz	100 kHz	100 kHz
1 µA	10 kHz	10 kHz	10 kHz	10 kHz
100 nA	1 kHz	1 kHz	1 kHz	1 kHz
10 nA	100 Hz	100 Hz	100 Hz	100 Hz

Table 19 Bandwidth overview of the 7 Series Autolab instruments

Instrument	PGSTAT12, PGSTAT100		PGSTAT30, PGSTAT302	
Mode	**GSTAT**	**iR compensation**	**GSTAT**	**iR compensation**
1 A - 1 mA	> 500 kHz	> 500 kHz	> 1.25 MHz	> 1.25 MHz
100 µA	125 kHz	500 kHz	125 kHz	1 MHz
10 µA	100 kHz	100 kHz	100 kHz	100 kHz
1 µA	10 kHz	10 kHz	10 kHz	10 kHz
100 nA	1 kHz	1 kHz	1 kHz	1 kHz
10 nA	100 Hz	100 Hz	100 Hz	100 Hz

Table 20 Bandwidth overview of the Autolab PGSTAT101, M101, PGSTAT204 and M204

Instrument	PGSTAT101, M101		PGSTAT204, M204	
Mode	**GSTAT**	**iR compensation**	**GSTAT**	**iR compensation**
100 mA			> 1 MHz	> 1 MHz
10 mA - 1 mA	> 1 MHz	> 1 MHz	> 1 MHz	> 1 MHz
100 µA	1 MHz	1 MHz	> 1 MHz	> 1 MHz
10 µA	10 kHz	75 kHz	10 kHz	50 kHz
1 µA	10 kHz	20 kHz	10 kHz	50 kHz
100 nA	400 Hz	4 kHz	500 Hz	500 Hz
10 nA	400 Hz	400 Hz	500 Hz	500 Hz

Table 21 Bandwidth overview of the Autolab PGSTAT10, PGSTAT20, µAutolab type II and µAutolab type III

Instrument	PGSTAT10, µAutolab type II, µAutolab type III		PGSTAT20	
Mode	**GSTAT**	**iR compensation**	**GSTAT**	**iR compensation**
1 A - 10 mA			> 1 MHz	> 1 MHz
10 mA - 1 mA	> 1 MHz		> 1 MHz	> 1 MHz

Instrument	PGSTAT10, µAutolab type II, µAutolab type III		PGSTAT20	
Mode	GSTAT	iR compensation	GSTAT	iR compensation
100 µA	500 kHz		500 kHz	500 kHz
10 µA	50 kHz		50 kHz	50 kHz
1 µA	5 kHz		5 kHz	5 kHz
100 nA	400 Hz		400 Hz	400 Hz
10 nA	20 Hz			

When a bandwidth conflict is detected in NOVA, a warning is provided in order to provide information on this conflict *(see Figure 1064, page 869)*.

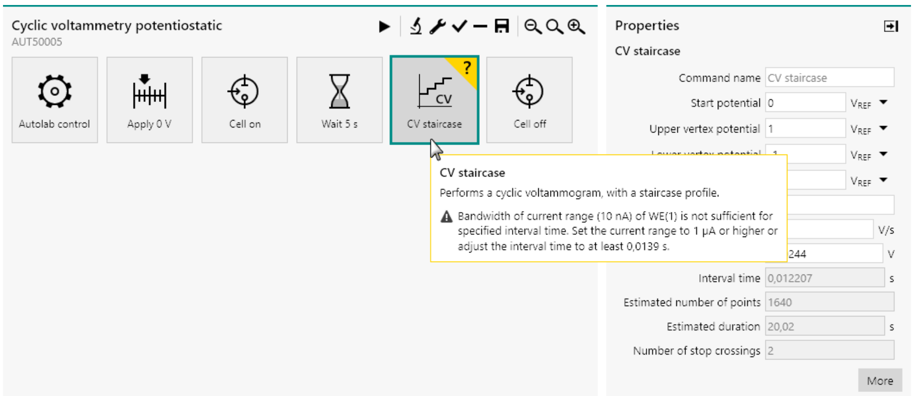


Figure 1064 Warnings are provided when bandwidth limitations are encountered

NOTE

It is possible to ignore this warning and to proceed with the measurement. This can however lead to instabilities or invalid measurements. It is therefore not recommended to adjust the procedure properties.

16.1.2.3.3 Oscillation detection and protection

The N Series Autolab instruments and the 7 Series Autolab instruments are fitted with a detector for large-amplitude oscillation. The detector will spot any signal swing that causes the control amplifier to produce both a positive and a negative voltage overload within ~ 200 µs. Thus, large oscillations at frequencies > 2.5 kHz will be detected.

Upon oscillation, the **OSC** indicator on the PGSTAT front panel will be activated *(see Figure 1070, page 882)*. The **Vovl** warning will also be shown in the Autolab display.

When an oscillation is detected, the cell will be automatically disconnected for safety reasons and the **OSC** indicator will blink on the Autolab front panel. The **Autolab display** panel will display that the cell is set to *Manually off* and the **Oscillation** warning indicator will be lit in the **Warnings** sub-panel *(see Figure 1065, page 870)*.

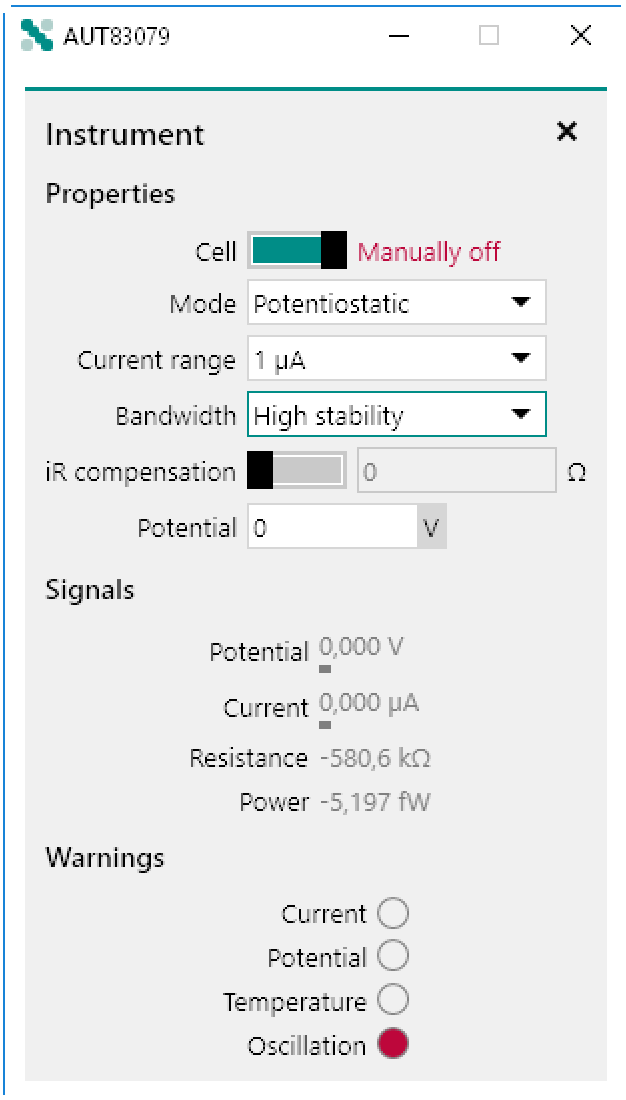


Figure 1065 The oscillation status is reported in the Autolab display panel

The **Cell ON** button (item 1 in *Figure 1070*) or the **CELL ENABLE** button (item 1 in *Figure 1097*), located on the right-hand side of the instrument front panel will blink.

The cell may be switched on again by pressing the **Cell ON/CELL ENABLE** button. If oscillation resumes, the cell will be switched off as soon as the button is released. Holding the button pressed in, provides an opportunity to observe the system during oscillation. Some cells that cause ringing when switching the cell on or changing the current range can falsely trigger the oscillation detector. If this happens, the oscillation protection may be switched off in the software in order to prevent an accidental disconnection of the cell.

The oscillation protection feature can be enabled or disabled in the software, using the **Autolab control** command *(see Figure 1066, page 871)*.

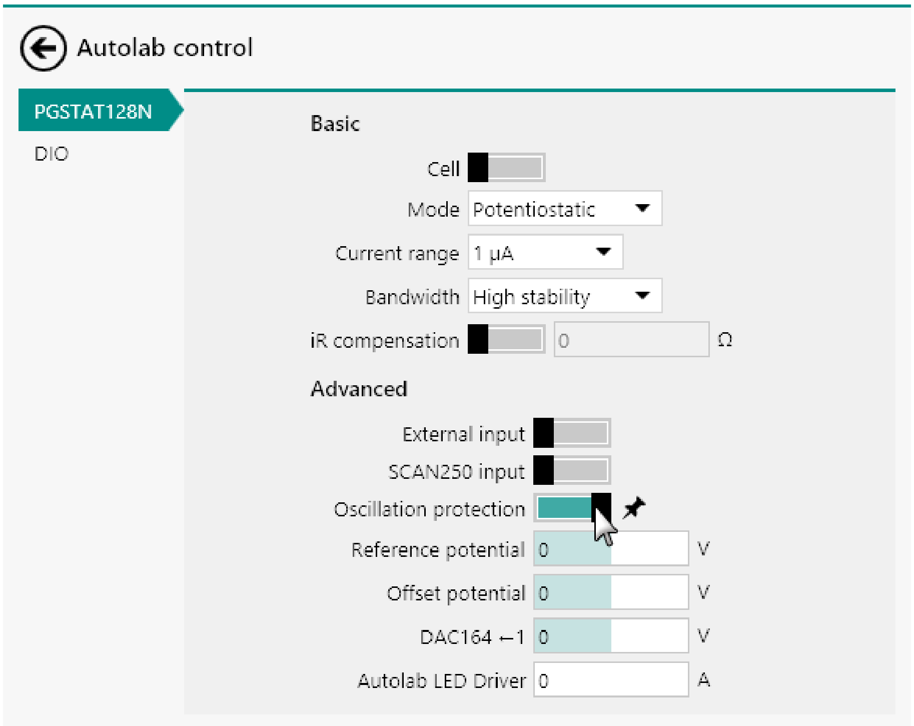


Figure 1066 The oscillation protection can be enabled or disabled in the Autolab control command

CAUTION

It is **not** recommended to switch off the oscillation protection circuit.

16.1.2.4 Current range linearity

Each current range on the instrument is characterized by a specific linearity limit and this specification determines the maximum current that can be applied in galvanostatic mode. This limit also determines the maximum current that can be measured in potentiostatic mode in a given current range.

The procedure validation provides an **error** message when the specified current exceeds the linearity limit in a galvanostatic experiment *(see Figure 1067, page 872)*.

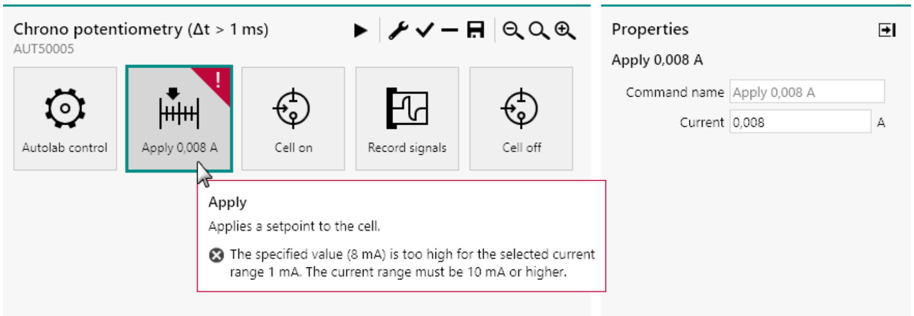


Figure 1067 An error is displayed when the applied current exceeds the linearity limit of the active current range

NOTE

It is **not** possible to start the measurement when an error is shown.

Whenever this limit is exceeded during a potentiostat measurement, a current overload **warning** message is shown after the measurement finishes *(see Figure 1068, page 872)*.

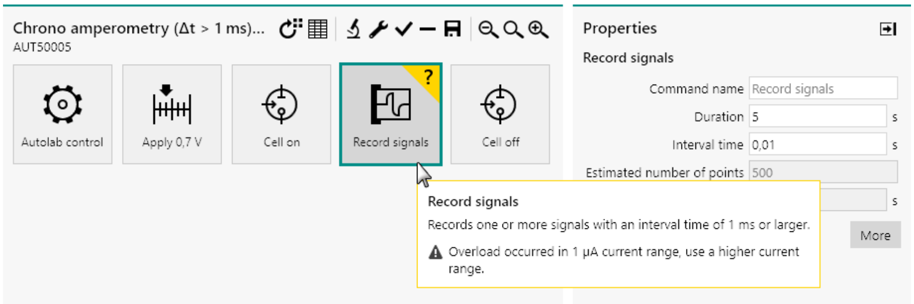


Figure 1068 A warning is displayed if the linearity limit is exceeded during a measurement

An overview of the current range linearity can be found in:

- *Table 22* for the N Series Autolab instrument.
- *Table 23* for the 7 Series Autolab instruments.
- *Table 24* for the PGSTAT101, PGSTAT204, M101 and M204.
- *Table 25* for the µAutolab type II and µAutolab type III, PGSTAT10 and PGSTAT20.

Table 22 Linearity limit for the N Series Autolab instruments

Current range	PGSTAT128N	PGSTAT302N	PGSTAT100N
1 A	0.8	2	

Current range	PGSTAT128N	PGSTAT302N	PGSTAT100N
100 mA	3	3	2.5
10 mA - 1 mA	3	3	3
100 µA - 1 µA	3	3	3
100 nA - 10 nA	3	3	3

Table 23 Linearity limit for the 7 Series Autolab instruments

Current range	PGSTAT12	PGSTAT30/302	PGSTAT100
1 A		1/2	
100 mA	2.5	3	2.5
100 mA - 1 mA	3	3	3
100 µA - 1 µA	3	3	3
100 nA - 10 nA	3	3	3

Table 24 Linearity limit for the Autolab PGSTAT101, M101, PGSTAT204 and M204

Current range	PGSTAT101, M101	PGSTAT204, M204
100 mA		4
10 mA	10	7
1 mA	7	7
100 µA - 1 µA	7	7
100 nA - 10 nA	7	7

Table 25 Linearity limit for the Autolab PGSTAT10, PGSTAT20, µAutolab type II and µAutolab type III

Current range	PGSTAT10	PGSTAT20	µAutolab type II, µAutolab type III
1 A		1	
100 mA		4	
10 mA	5	4	5
1 mA	4	4	4
100 µA - 1 µA	4	4	4

Current range	PGSTAT10	PGSTAT20	µAutolab type II, µAutolab type III
100 nA	4	4	4
10 nA			4

The values reported in the tables indicate how many times the current range value can be applied or measured for each current range. For example, for in the 1 mA current range, the linearity limit is ± 3 mA.

16.1.2.5 Maximum input voltage

The differential electrometer input contains an input protection circuitry that becomes active after crossing the ± 10 V limit. This is implemented to avoid electrometer damage. Please note that the **Vovl** indicator, on the front panel of the instrument, will not light up for this type of voltage overload.

The measured voltage will be cutoff at an absolute value of ± 10.00 V.

Depending on the cell properties, galvanostatic control of the cell could lead to a potential difference between the reference electrode (RE) and the sense electrode (S) larger than 10 V. This situation will trigger the cutoff of the measured voltage to prevent overloading the differential amplifier.

In this case it is possible to connect the Autolab **Voltage Multiplier** to extend the measurable range of the differential amplifier to ± 100 V.

16.1.2.6 Active cells

Energy storage and conversion devices like batteries and fuel cells are capable of delivering power to the Autolab potentiostat/galvanostat. This is allowed only to a maximum active cell power, $\mathbf{P_{MAX}}$. The values for $\mathbf{P_{MAX}}$ depend on the instrument type and are reported in *Table 26*.

Table 26 Maximum power rating for the different Autolab instruments

Instrument	Maximum power, P_{MAX} (W)
PGSTAT128N	8
PGSTAT302N	20
PGSTAT100N	2.5
PGSTAT12	2.5
PGSTAT302	20
PGSTAT30	10

Instrument	Maximum power, P_{MAX} (W)
PGSTAT100	2.5
PGSTAT101/M101	1
PGSTAT204/M204	4
PGSTAT10	1
PGSTAT20	10
µAutolab type II, µAutolab III	0.5
Booster10A	100
Booster20A	200

This means that cells showing an absolute voltage, $|V_{Cell}|$, of less than 10 V between the working electrode (WE) and counter electrode (CE) are intrinsically safe. They may drive the PGSTAT control amplifier into current limit but will not overload the amplifier. On the other hand, cells that have an absolute voltage higher than 10 V between WE and CE may only deliver a maximum current, i_{MAX} given by:

$$i_{MAX} = \frac{P_{MAX}}{|V_{MAX}|}$$

NOTE

Instruments that can be connected to the optional **Booster10A** or **Booster20A** can work with active cell power values of 100 W and 200 W, respectively. More information on the Booster10A and Booster 20A can be found in *Chapter 16.3.2.5* and *Chapter 16.3.2.6*.

16.1.2.7 Grounded cells

The measurement circuitry of the Autolab is internally connected to protective earth (P.E.). This can be an obstacle when measurement is desired of a cell that is itself in contact with P.E.. In such a case, undefined currents will flow through the loop that is formed when the electrode connections from the PGSTAT are linked to the cell and measurements will not be possible.

Please note that not only a short circuit or a resistance can make a connection to earth, but also a capacitance is capable of providing a conductive path (for AC signals). The earth connection between the cell and P.E. should always be broken.

If there is no possibility of doing this, please contact Metrohm Autolab for a custom solution, if available.

16.1.3 Environmental conditions

The PGSTAT may be used at temperatures of 0 to 40 degrees Celsius. The instrument is calibrated at 25 degrees Celsius and will show minimum errors at that temperature. The ventilation holes on the bottom plate and on the rear panel may never be obstructed, nor should the instrument be placed in direct sunlight or near other sources of heat.

16.1.3.1 Temperature overload

As a safety precaution, the PGSTAT is equipped with a circuit that monitors the temperature of the internal power electronics. A temperature overload will be displayed as a blinking indicator in the manual cell switch, with the cell automatically turned off. You will not be able to turn the cell back on until the temperature inside the instrument has fallen to an acceptable level. It can then be switched on again by pressing the manual cell switch button on the front panel.

During normal operation the temperature should never become extremely high and no temperature overload will occur. If this does happen, the origin of the temperature overload should be identified:

1. Is the room temperature unusually high?
2. Was the PGSTAT oscillating?
3. Is the voltage selector for mains power set to the right value?
4. Is the fan turning and are all the ventilation holes unobstructed?
5. Was the cell delivering a considerable amount of power to the PGSTAT?
6. Are the WE and CE cables shorted in PSTAT mode?

NOTE

If a temperature overload takes place repeatedly, for no obvious reason, Metrohm Autolab recommends having the instrument checked by their service department.

16.1.4 Noise considerations

When measuring low level currents, some precautions should be taken in order to minimize noise. The personal computer must be placed as far away as possible from the electrochemical cell and the cell cables. The cell cables should not cross other electrical cables. Other equipment with power supplies can also cause noise. For instance, the interface for mercury electrodes IME should also be placed with some care. If possible place the computer between the PGSTAT and other equipments. Avoid using unshielded extension cables to the electrodes. The use of a Faraday cage is also advised.

If the cell system has a ground connector, it can be connected to the analog ground connector at the front of the PGSTAT. If a Faraday cage is used, it should be connected to this ground connector. Some experiments concerning optimization of the signal-to-noise ratio can readily indicate whether or not a configuration is satisfactory.

When investigating the sources of noise, it is recommended to consider following items:

1. Problems with the reference electrode
2. Problems with unshielded cables
3. Faraday cage
4. Grounding of the instrument
5. Magnetic stirrer
6. Position of the cell with respect to the instrument and accessories
7. Measurements in a glove box

NOTE

The **Check cell** tool, provided in the **Instrument control panel**, can be used to evaluate the noise levels. More information can be found in *Chapter 5.2.2.4*.

16.1.4.1 Problems with reference electrodes

If the reference electrode is not filled properly with electrolyte solution or when it has, for other reasons, a very high impedance, it may introduce noise in electrochemical measurement. In most cases the applied potential is not the same as the measured potential. Refer to the user manual provided by the reference electrode supplier for more information on the proper care of your reference electrode.

16.1.4.2 Problems with unshielded cables

It is not advisable to use unshielded electrode cables. Make the connections to the electrodes as close as possible to the electrode itself. Avoid the use of unshielded extension cables to the electrodes.

16.1.4.3 Faraday cage

The use of a Faraday cage is always recommended. It protects the cell from external noise interference. Connect the cage to the green ground connector embedded in the cell of the Autolab.

16.1.4.4 Grounding of the instrument

Not properly grounding of the Autolab and computer will decrease the signal-to-noise ratio. Always use a grounded power outlet and grounded power cables. Be sure to connect the Autolab and computer to the same power ground. This means they should be connected to the same power outlet.

16.1.4.5 Magnetic stirrer

In some cases a magnetic stirrer can cause noise problems. Try the measurements with the stirrer on and off and monitor the current. If the stirrer causes a lot of noise please try to find another way of stirring.

16.1.4.6 Optimizing the position of the instrument

The signal-to-noise ratio can often be improved by changing the positions of the cell, computer and ancillary equipment relative to the Autolab. In general, the electrochemical cell should be placed as far as possible from the computer and other devices, without extending the cell cables with unshielded cables. If the noise level remains too high, a Faraday cage may be necessary.

16.1.4.7 Measurements in a glove box

When the cell needs to be placed into a glove box, it is highly recommended to use **isolated** feedthrough that allows the Autolab cell cables to be connected to the cell inside the glove box. If necessary, the cell cables of the Autolab can be fitted with male BNC connectors rather than 4 mm banana connectors. This allows using BNC feedthroughs. Contact your Autolab distributor for more information about this modification.

CAUTION

The shielding of the reference electrode (RE) and sense electrode (S) cable on the Autolab is driven (or guarded). Use **isolated** cable feedthroughs for these cables in order to extend the driven shield inside the glove box. The shield of these cables must not be connected to the ground of the glove box.

16.1.5 Cleaning and inspection

It is recommended to clean the Autolab instrument and the accessories on a regular basis. This can be done with a damp cloth, optionally using a mild detergent. Never use an excessive amount of water; it may never enter into the instrument. As a precaution, disconnect Autolab from the mains when cleaning it. Also perform an inspection of the instrument and all of the connecting cables. If you find any cables with damaged insulation or other irregularities, stop using the instrument until it has been repaired.

CAUTION

Damaged equipment or damaged cables may be hazardous!

16.2 Instrument description

This chapter describes the Autolab instruments supported in NOVA.

NOTE

Some of the instruments described in this chapter are no longer available but are still supported. Whenever applicable, the successor instrument is specified.

CAUTION

Information on the 9 Series Autolab instruments is not provided in this manual. The reader is kindly invited to refer to the original documentation provided with the instrument.

16.2.1 Autolab N Series (AUT8) instruments

The Autolab N Series is the latest version of the modular potentiostat/galvanostat produced by Metrohm Autolab. These instruments, identified by a serial number starting with **AUT8**, are based on a modular concept that allows the instrument to be complemented by internal or external extension modules.

The following instruments belong to the Autolab N Series:

- **Autolab PGSTAT302N:** modular PGSTAT with 30 V compliance and 2 A maximum current.
- **Autolab PGSTAT128N:** modular PGSTAT with 12 V compliance and 800 mA maximum current.
- **Autolab PGSTAT100N:** modular PGSTAT with 100 V compliance and 250 mA maximum current.

16.2.1.1 Scope of delivery

The N Series Autolab systems are supplied with the following items:

- Autolab potentiostat/galvanostat
- ADC164 (installed)
- DAC164 (installed)
- Cell cable (WE/CE/GND)
- Differential amplifier (RE/S)
- Monitor cable
- Power cable
- BNC cable (50 cm)
- USB cable
- Set of four alligator clips
- Autolab dummy cell

16.2.1.2 Instrument power-up state

The power-up state of the instrument is hardware defined. The following settings are automatically selected whenever the instrument is powered on or whenever the connection to the instrument is reset by the software.

- Cell: off
- Mode: potentiostatic
- Control bandwidth: high stability
- iR compensation: off
- Current range: 10 mA
- Optional modules: off
- DIO ports: write mode, low state
- Summation point inputs: off
- Oscillation protection: on

16.2.1.3 N Series Autolab front panel

The front panel of the N Series Autolab provides a number of connections, controls and indicators *(see Figure 1069, page 881)*.

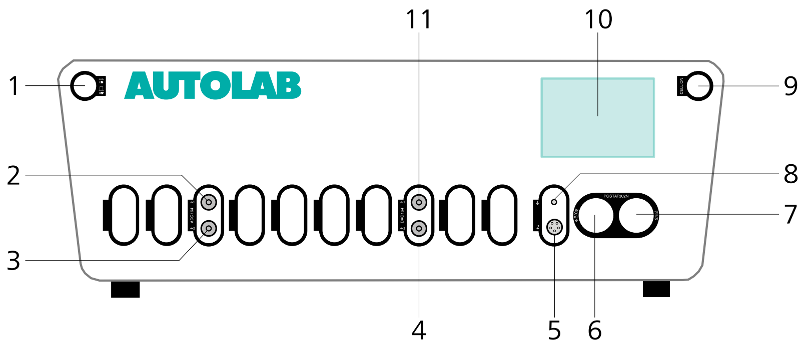


Figure 1069 Overview of the front panel of the N Series Autolab

1 On/Off button For switching the Autolab on or off.	**2 ADC164 →1** Analog input for recording external signals (ADC164 →1).
3 ADC164 →2 Analog input for recording external signals (ADC164 →2).	**4 DAC164 ←2** Analog output for controlling external signals (DAC164 ←2).
5 Monitor cable connector ⇄ For connecting the monitor cable.	**6 CE/WE connector** For connecting the Autolab cell cable, providing connections to the counter electrode (CE), working electrode (WE) and ground.
7 RE/S connector For connecting the Autolab differential amplifier, providing connections to the reference electrode (RE) and sense electrode (S).	**8 Ground connector** Additional ground connector for connecting external devices to the Autolab ground.
9 Cell ON button For enabling and disabling the cell.	**10 Display** Display indicating real-time information on the measured current and potential and instrumental settings.
11 DAC164 ←1 Analog output for controlling external signals (DAC164 ←1).	

The display (item 10 in *Figure 1069*) is used to provide information about the Autolab to the user. *Figure 1070* shows a detail of this display.

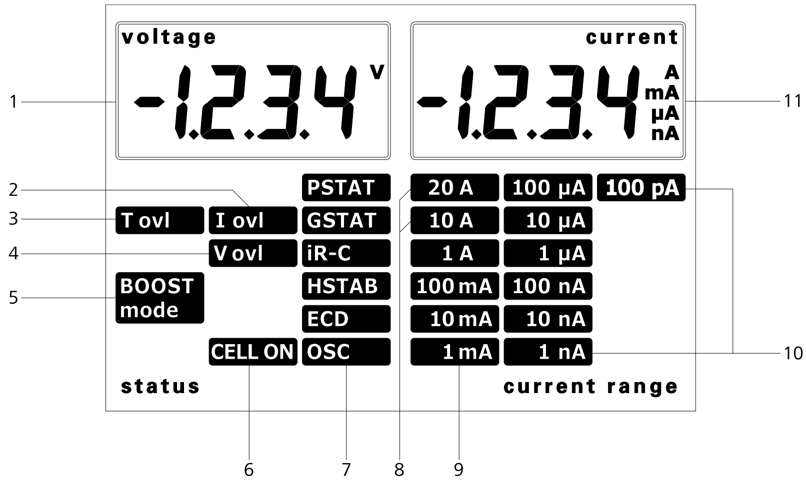


Figure 1070 Overview of the display of the Autolab

1 Voltage indicator
Displays the measured voltage.

2 I ovl indicator
Indicates that a current overload is detected when lit.

3 T ovl
Indicates that a temperature overload is detected when lit.

4 V ovl
Indicates that a voltage overload is detected when lit.

5 BOOST mode
Indicates that a connected Booster is active when lit.

6 CELL ON
Indicates that the cell is on when lit.

7 Operation mode indicators
Indicate the operation settings of the Autolab. From top to bottom:

PSTAT indicates that the Autolab is operating in potentiostatic mode when lit.

GSTAT indicates that the Autolab is operating in galvanostatic mode when lit.

iR-C indicates that the ohmic drop compensation is on when lit

HSTAB indicates that the Autolab is operating in high stability mode when lit.

ECD indicated that the ECD module is on when lit.

OSC indicates that oscillations are detected when lit.

8 Booster current range
Indicate that a current range provided by a Booster is active when lit.

9 Autolab current ranges
The current range indicator which is lit corresponds to the active current of the Autolab.

10 ECD current ranges
Additional current ranges provided by the ECD module. These current ranges extend the ranges of the Autolab.

11 Current indicator
Displays the measured current.

NOTE

The Voltage and Current values shown in the display are provided with an accuracy of 0.5 %. These values are provided for information only.

16.2.1.4 Autolab N Series back plane

The back plane of the Autolab N Series provides a number of connections, shown in *Figure 1071*.

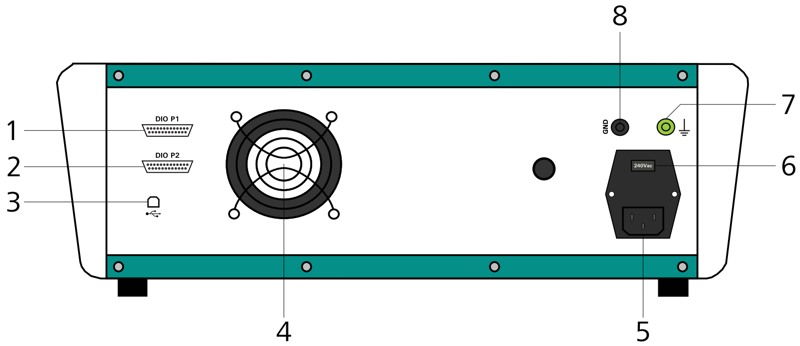


Figure 1071 Overview of the back plane of the Autolab N Series

1 DIO P1 connector
Digital input/output connector **P1** for sending and receiving external TTL triggers.

2 DIO P2 connector
Digital input/output connector **P2** for sending and receiving external TTL triggers.

3 USB connector
Type B USB plug for connecting the USB cable to the host computer.

4 Fan
Required for cooling the Autolab during operation.

5 Mains connection socket
For connecting the Autolab to the mains supply.

6 Mains voltage indicator
Indicates the mains voltage settings of the Autolab.

7 Earth plug
For connections to the protective earth

8 GND plug
For connections to the Autolab ground.

CAUTION

Make sure that the mains voltage indicator is set properly before switching the Autolab on.

16.2.1.5 Connections for analog signals

The N Series Autolab instruments provide connections for analog signals through two different types of connectors:

- BNC connectors directly located on the front panel of the instrument *(see Chapter 16.2.1.5.1, page 884)*.
- BNC connectors located on the monitor cable *(see Chapter 16.2.1.5.2, page 884)*.

CAUTION

Avoid creating ground loops when connecting the Autolab to external signals as this will degrade the performance of the instrument.

16.2.1.5.1 Front panel connections for analog signals

The **ADC164** module and the **DAC164** module, installed in all the N Series Autolab instruments, are fitted with two analog inputs and two analog outputs, respectively *(see Figure 1069, page 881)*.

- **ADC164:** the ADC164 inputs, labeled →1 and →2 on the front panel, can be used to record any analog signal with a ± 10 V value range. The input impedance of the two analog inputs is ≥ 1 GΩ. More information on the ADC164 is provided in *Chapter 16.3.1.1*.
- **DAC164** the DAC164 outputs, labeled ←1 and ←2 on the front panel, can be used to generate any analog signal with a ± 10 V value range. The output impedance of these two inputs is 50 Ω. Corrections should be made with loads smaller than 100 kΩ. Because of dissipation, the minimum load impedance should be 200 Ω. More information on the DAC164 is provided in *Chapter 16.3.1.2*.

16.2.1.5.2 Monitor cable connections for analog signals

The **monitor cable**, supplied with the instrument, provides additional connections for analog signals, through BNC connectors. All the connections are with respect to the Autolab ground directly and indirectly with respect to the protective earth *(see Figure 1072, page 885)*.

Figure 1072 The monitor cable provided with the N Series Autolab instruments

To use the monitor cable, connect the cable to the matching connector located on the front panel of the Autolab. This connector, labeled ⇄, is located below the front panel display (item 5 in *Figure 1069*).

The following connections are provided through the monitor cable:

- **Eout:** this output corresponds to the differential potential of the reference electrode (RE) with respect to the sense electrode (S). The output voltage will vary between ± 10 V. The output impedance is 50 Ω, so a correction should be made if a load smaller than 100 kΩ is connected to this output. The minimum load is 200 Ω.
- **iout:** this output corresponds to the output of the current-to-voltage converter circuit of the Autolab. The output corresponds to the measured current divided by the current range. The output voltage will vary between ± 10 V. The output impedance is 50 Ω, so a correction should be made if a load smaller than 100 kΩ is connected to this output. The minimum load is 200 Ω.
- **Ein:** this input corresponds to an analog voltage input, directly connected to the summation point of the Autolab. This input is disabled by default. When it is enabled, it can be used to control the Autolab through an external waveform generator. In potentiostatic mode, 1 V provided on this input will add 1 V to the applied potential. In galvanostat mode, 1 V provided on this input will add an extra current equal to 1 multiplied by the current range. In both cases, the converted signal is added to the value already applied by the potentiostat or the galvanostat circuit. The input range is ± 10 V and the input impedance is 1 kΩ when the connection is enabled, so a correction should be made when the source impedance is larger than 1 Ω.

The **Ein** input is enabled and disabled using the **Autolab control** command *(see Figure 1073, page 886)*.

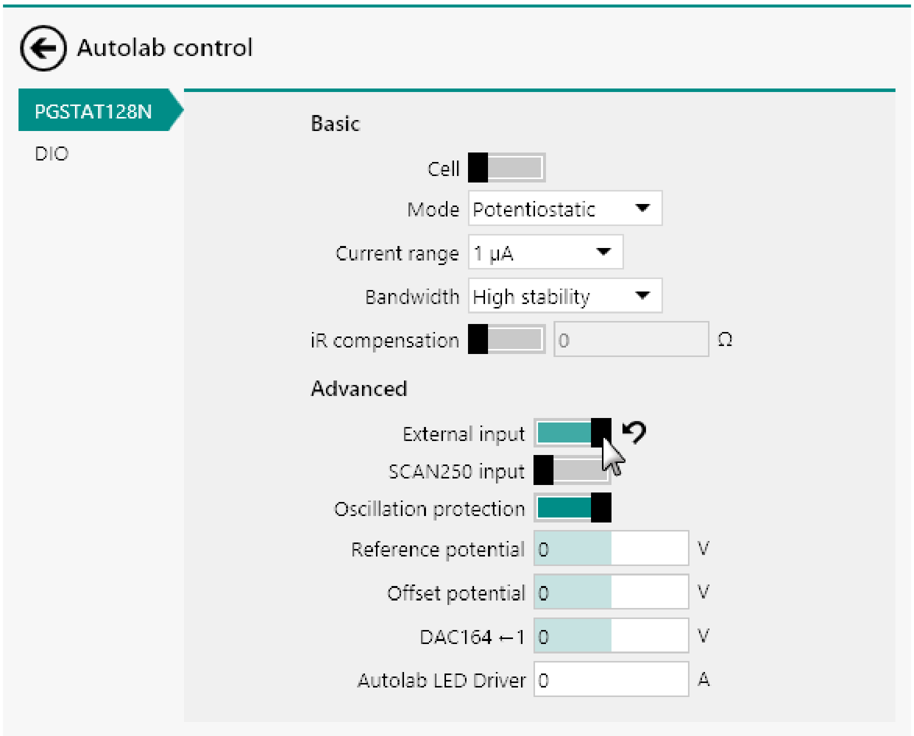


Figure 1073 The Ein connection is enable or disabled in the Autolab control command

CAUTION

Do not leave the **Ein** connection enabled unnecessarily to prevent noise pickup by the Autolab.

16.2.1.6 N Series Autolab restrictions

Restrictions apply when using the N Series Autolab potentiostat/galvanostat:

- **Intended use:** the Autolab potentiostat/galvanostat is intended to be used for electrochemical research only.
- **Service:** there are **no** serviceable parts inside. Servicing of the instrument can only be carried out by qualified personnel.

CAUTION

All attempts to service the instrument will lead to the immediate voiding of any warranty.

WARNING

The PGSTAT100N is fitted with a control amplifier capable of generating up to 100 V potential difference between the counter electrode (CE) and the working electrode (WE). Take all necessary precautions when working with this instrument and use the supplied warning laminated sheet to warn others.

16.2.1.7 N Series Autolab testing

NOVA is shipped with a procedure which can be used, alongside the **Diagnostics** application, to verify that the instrument is working as expected.

NOTE

For more information on the **Diagnostics** application, please refer to *Chapter 17*.

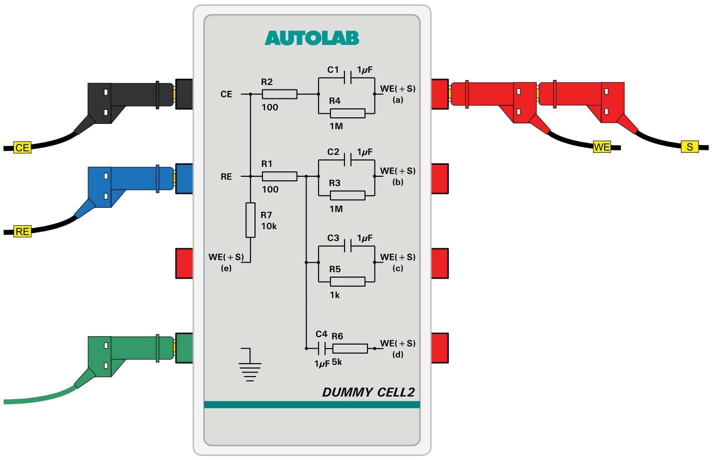
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestCV** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test \TestCV.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (a).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1074*.

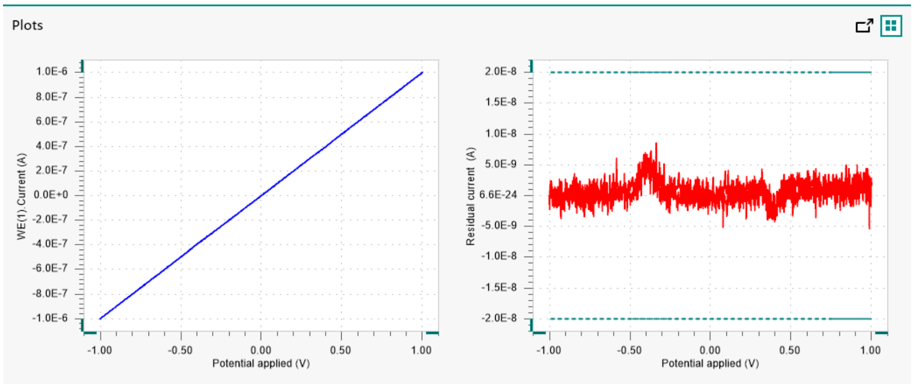
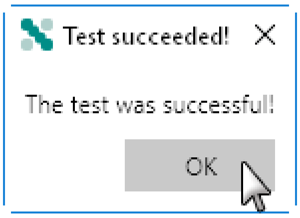


Figure 1074 The data measured by the TestCV procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestCV automatic evaluation of the data requires the following tests to succeed:

1. The residual current, determined by subtracting the expected current from the measured current, must be smaller or equal than ± 20 nA.
2. The inverted slope of the measured current versus the applied potential must be equal to 1000100 Ω± 5 %.
3. The intercept of the measured current versus the applied potential must be equal to ± 4 mV divided by 1000100 Ω± 5 %.

All three conditions must be valid for the test to succeed.

16.2.1.8 Autolab N Series specifications

The specifications of the Autolab N Series are provided in *Table 27*.

Table 27 Specifications of the Autolab 7 Series instruments

Instrument	PGSTAT128N	PGSTAT302N	PGSTAT100N
Maximum current	± 800 mA	± 2 A	± 250 mA
Compliance voltage	± 12 V	± 30 V	± 100 V
Potential range	± 10 V		
Applied potential accuracy	± 0.2 % ± 2 mV		
Applied potential resolution	150 µV		
Measured potential resolution	300 nV (gain 1000)		
Current ranges	10 nA to 1 A, 9 decades		10 nA to 100 mA, 8 decades
Current accuracy	± 0.2 % of current range		
Applied current resolution	0.015 % of current range		
Measured current resolution	0.00003 % of current range (gain 1000)		
Potentiostat bandwidth	500 kHz	1 MHz	400 kHz
Potentiostat rise/fall time	< 250 ns		< 500 ns
Input impedance of electrometer	> 1 TΩ, 8 pF		> 100 GΩ, 8 pF
Input bias current	< 1 pA		
Electrometer bandwidth	> 4 MHz		

Instrument	PGSTAT128N	PGSTAT302N	PGSTAT100N
iR compensation	2 Ω - 200 MΩ		200 mΩ - 200 MΩ
iR compensation resolution	0.025 %		
Analog output	Potential and current		
Analog voltage input	Yes		
External inputs	2		
External outputs	2		
Digital input/output	48		
Interface	USB (internal or external)		
Warm-up time	30 minutes		
Pollution degree	2		
Installation category	II		
External dimensions (without cables and accessories)	52x42x16 cm^3		
Weight	16 kg	18 kg	21 kg
Power requirements	180 W	300 W	247 W
Power supply	100 - 240 V ± 10% in four ranges 100 V: [90 - 121 V] 120 V: [104 - 139 V] 230 V: [198 - 242 V] 240 V: [207 - 264 V]		
Power line frequency	47-63 Hz		

<table>
<tr><th>Instrument</th><th>PGSTAT128N</th><th>PGSTAT302N</th><th>PGSTAT100N</th></tr>
<tr><td>Fuse</td><td colspan="2">100 V, 120 V: 3.15 A (slow-slow)
230 V, 240 V: 1.6 A (slow-slow)</td><td>100 V, 120 V: 3.15 A (slow-slow)
230 V, 240 V: 1.25 A (slow-slow)</td></tr>
<tr><td>Operating environment</td><td colspan="3">0 °C to 40 °C, 80 % relative humidity without derating</td></tr>
<tr><td>Storage environment</td><td colspan="3">-10 °C to 60 °C</td></tr>
</table>

16.2.2 Autolab F Series (AUT8) instrument

The Autolab F Series is a special version of the Autolab N Series. A single instrument is available in this series:

- **Autolab PGSTAT302F:** modular PGSTAT with 30 V compliance and 2 A maximum with floating option.

The Autolab **PGSTAT302F** is a special version of the modular **PGSTAT302N** potentiostat/galvanostat produced by Metrohm Autolab. This instrument, identified by a serial number starting with **AUT8**, is based on a modular concept that allows the instrument to be complemented by internal or external extension modules.

NOTE

The Autolab **PGSTAT302F** derives from the **PGSTAT302N**. This chapter will only provide details on instrumental properties that deviate from the properties of the **PGSTAT302N**. Please refer to *Chapter 16.2.1* for additional information on the common properties of the instruments.

The **PGSTAT302F** is designed to be operated in two different modes:

- **Normal mode (grounded):** in this mode, the **PGSTAT302F** operates like a normal **PGSTAT302N**. In this mode, the electrochemical and working electrodes are floating with respect to the grounded instrument.
- **Floating mode:** in this mode, the **PGSTAT302F** can be used to control the potential of grounded working electrodes or can work with electrochemical cells connected to ground. In this configuration, the Autolab is floating with respect to the working electrode sample or with respect to the cell.

NOTE

Special precautions must be taken with the cell connections when the **PGSTAT302F** is used in floating mode. Only the working electrode can be connected to ground, all other electrodes must be isolated from ground. External equipments connected to the **PGSTAT302F** must be isolated when the instrument is used in floating mode. Keep in mind that grounding of external equipment can occur through connections to a computer, if applicable (for example through a USB or RS232 cable).

CAUTION

The floating mode of the **PGSTAT302F** must only be used on grounded working electrodes or grounded cells. The working electrode or the cell can be grounded using the green ground connector embedded in the CE/WE cable of the **PGSTAT302F**.

CAUTION

Instrument performance can be substantially degraded when the **PGSTAT302F** is operated in floating mode. The instrument specifications provided by Metrohm Autolab can only be achieved when the **PGSTAT302F** is used in normal mode.

Unlike the **PGSTAT302N** from which it is derived, the **PGSTAT302F** only accommodate the following module:

- **FRA32M:** impedance spectroscopy module *(see Chapter 16.3.2.13, page 1091)*.

16.2.2.1 Scope of delivery

The F Series Autolab systems are supplied with the following items:

- Autolab potentiostat/galvanostat
- ADC164 (installed)
- DAC164 (installed)
- Cell cable (WE/CE/GND), only suitable for PGSTAT302F
- Differential amplifier (RE/S), only suitable for PGSTAT302F
- Monitor cable
- Power cable
- BNC cable (50 cm)

- USB cable
- Set of four alligator clips
- Autolab dummy cell

CAUTION

The cables supplied with the **PGSTAT302F** can only be used in combination with this type of instrument.

16.2.2.2 Instrument power-up state

The power-up state of the instrument is hardware defined. The following settings are automatically selected whenever the instrument is powered on or whenever the connection to the instrument is reset by the software.

- Cell: off
- Mode: potentiostatic
- Control bandwidth: high stability
- iR compensation: off
- Current range: 10 mA
- Optional modules: off
- DIO ports: write mode, low state
- Summation point inputs: off
- Oscillation protection: on

CAUTION

In floating mode, the **Current overload** warning (**iOVL**) may be lit when the cell is off. This warning can be ignored.

16.2.2.3 Connections for analog signals

The Autolab PGSTAT302F provides connections for analog signals through two different types of connectors:

- BNC connectors directly located on the front panel of the instrument *(see Chapter 16.2.2.3.1, page 894)*.
- BNC connectors located on the monitor cable *(see Chapter 16.2.2.3.2, page 894)*.

CAUTION

Avoid creating ground loops when connecting the Autolab to external signals as this will degrade the performance of the instrument.

16.2.2.3.1 Front panel connections for analog signals

The **ADC164** module and the **DAC164** module, installed in all the **PGSTAT302F**, are fitted with two analog inputs and two analog outputs, respectively *(see Figure 1069, page 881)*.

- **ADC164:** the ADC164 inputs, labeled →1 and →2 on the front panel, can be used to record any analog signal with a ± 10 V value range. The input impedance of the two analog inputs is ≥ 1GΩ. More information on the ADC164 is provided in *Chapter 16.3.1.1*.
- **DAC164** the DAC164 outputs, labeled ←1 and ←2 on the front panel, can be used to generate any analog signal with a ± 10 V value range. The output impedance of these two inputs is 50 Ω. Corrections should be made with loads smaller than 100 kΩ. Because of dissipation, the minimum load impedance should be 200 Ω. More information on the DAC164 is provided in *Chapter 16.3.1.2*.

CAUTION

All the signals are with respect to Autolab ground and indirectly to protective earth when the **PGSTAT302F** is operated in *normal* mode. These connectors are **floating** when the **PGSTAT302F** is operated in *floating* mode. Connected equipment may not be connected to ground and the shield of the BNC cables may not be connected to safety ground.

16.2.2.3.2 Monitor cable connections for analog signals

The **monitor cable**, supplied with the **PGSTAT302F**, provides additional connections for analog signals, through BNC connectors *(see Figure 1075, page 894)*.

Figure 1075 The monitor cable supplied with the PGSTAT302F

To use the monitor cable, connect the cable to the matching connector located on the front panel of the Autolab. This connector, labeled ⇄, is located below the front panel display (item 5 in *Figure 1069)*.

The following connections are provided through the monitor cable:

- **Eout:** this output corresponds to the inverted differential potential of the reference electrode (RE) with respect to the sense electrode (S). The output voltage will vary between ± 10 V. The output impedance is 50 Ω, so a correction should be made if a load smaller than 100 kΩ is connected to this output. The minimum load is 200 Ω.
- **iout:** this output corresponds to the inverted output of the current-to-voltage converter circuit of the Autolab. The output corresponds to the measured current divided by the current range. The output voltage will vary between ± 10 V. The output impedance is 50 Ω, so a correction should be made if a load smaller than 100 kΩ is connected to this output. The minimum load is 200 Ω.
- **Ein:** this input corresponds to an analog voltage input, directly connected to the summation point of the Autolab. This input is disabled by default. When it is enabled, it can be used to control the Autolab through an external waveform generator. In potentiostatic mode, 1 V provided on this input will add -1 V to the applied potential. In galvanostat mode, 1 V provided on this input will add an extra current equal to -1 multiplied by the current range. In both cases, the converted signal is added to the value already applied by the potentiostat or the galvanostat circuit. The input range is ± 10 V and the input impedance is 1 kΩ when the connection is enabled, so a correction should be made when the source impedance is larger than 1 Ω.

CAUTION

All the signals are with respect to Autolab ground and indirectly to protective earth when the **PGSTAT302F** is operated in *normal* mode. These connectors are **floating** when the **PGSTAT302F** is operated in *floating* mode. Connected equipment may not be connected to ground and the shield of the BNC cables may not be connected to safety ground.

The **Ein** input is enabled and disabled using the **Autolab control** command *(see Figure 1073, page 886)*.

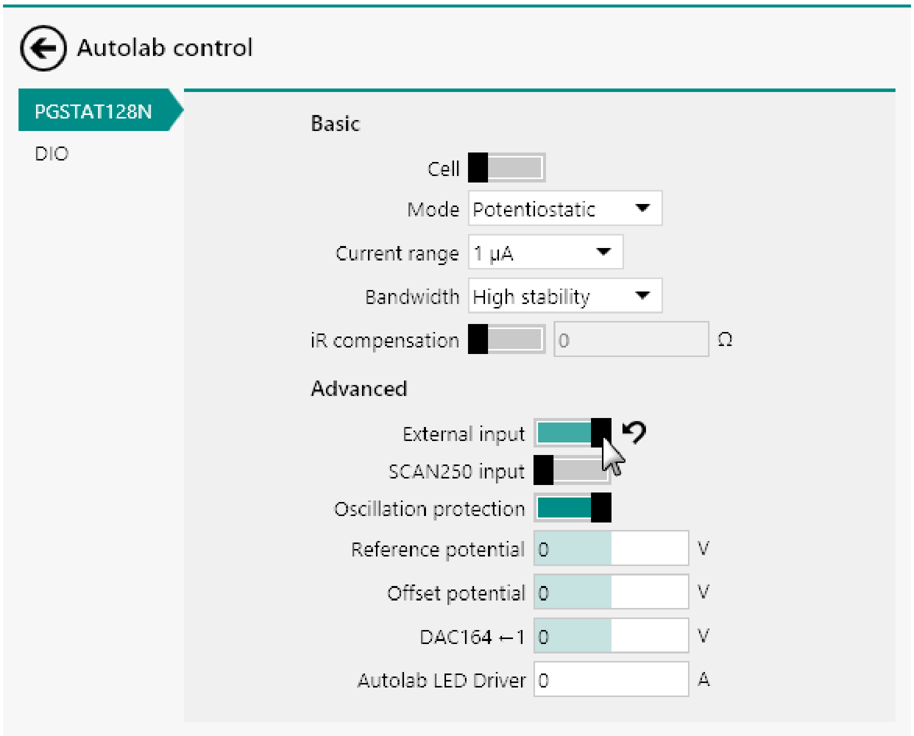


Figure 1076 The Ein connection is enable or disabled in the Autolab control command

CAUTION

Do not leave the **Ein** connection enabled unnecessarily to prevent noise pickup by the Autolab.

16.2.2.4 Grounded cells and grounded electrodes

The **PGSTAT302F** can be operated in two different modes:

- **Normal mode:** this mode corresponds to the operating mode using in all the PGSTAT instruments. This mode is suitable for working on electrochemical cells or electrodes that are floating with respect to the instrument.
- **Floating mode:** this mode is only available on the **PGSTAT302F**. In this mode, measurement circuitry of the Autolab is internally discon-nected to protective earth (P.E.). This allows the instrument to be used in combination with a grounded working electrode or a grounded cell.

The **PGSTAT302F** can be set to either normal mode or floating mode using a dedicated short-circuit plug on the back plane of the instrument *(see Figure 1077, page 897)*.

Figure 1077 The PGSTAT302F can be set to normal mode (left) or to floating mode (right) using the provided short-circuit plug

When the short-circuit plug is connected as shown above, the instrument operates in *normal* mode. When the short-circuit plug is disconnected from the back panel, the instrument operates in *floating* mode.

16.2.2.5 Autolab PGSTAT302F restrictions

Restrictions apply when using the Autolab **PGSTAT302F** potentiostat/galvanostat:

- **Intended use:** the Autolab potentiostat/galvanostat is intended to be used for electrochemical research only.
- **Service:** there are **no** serviceable parts inside. Servicing of the instrument can only be carried out by qualified personnel.

> **CAUTION**
>
> All attempts to service the instrument will lead to the immediate voiding of any warranty.

- **Compliance voltage limitation:** the control amplifier of the PGSTAT302F has an output range of ± 30 V. In combination with the default cell cables, supplied with the instrument, the output range of the instrument is reduced to ± 10 V. An optional set of cell cables can be used to increase the output range to ± 30 V. These optional cables cannot be used in *floating* mode.

16.2.2.6 Autolab PGSTAT302F testing

The Autolab PGSTAT302F can be tested using the following procedures:

1. Using the TestCV procedure for the Autolab PGSTAT302F in normal (grounded) mode). Please refer to *Chapter 16.2.2.6.1* for more information.
2. Using the TestCV PGSTAT302F procedure for the Autolab PGSTAT302F in floating mode. Please refer to *Chapter 16.2.2.6.2* for more information.

16.2.2.6.1 Autolab PGSTAT302F testing in Normal mode

NOVA is shipped with a procedure which can be used, alongside the **Diagnostics** application, to verify that the instrument is working as expected.

NOTE

For more information on the **Diagnostics** application, please refer to *Chapter 17*.

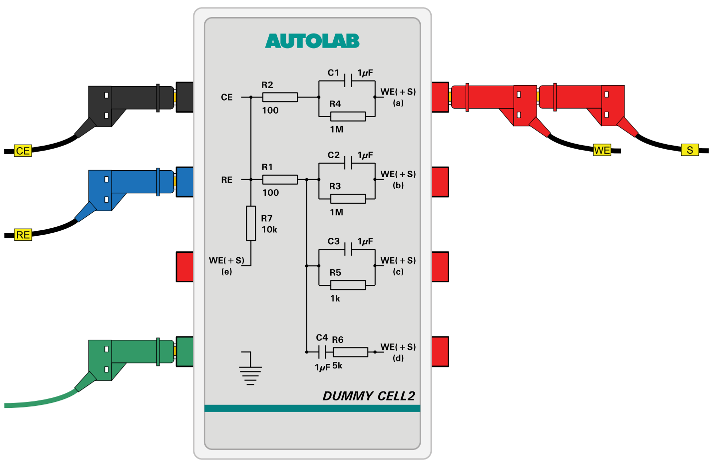
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestCV** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test \TestCV.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (a).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1078*.

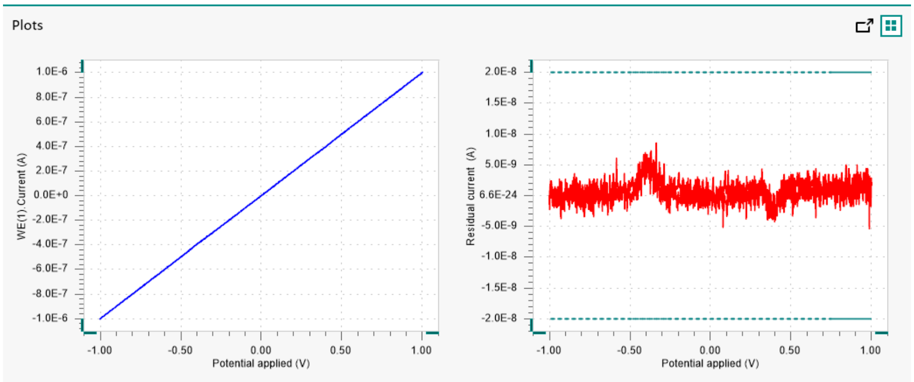
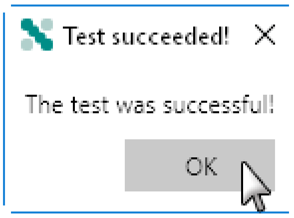


Figure 1078 The data measured by the TestCV procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestCV automatic evaluation of the data requires the following tests to succeed:

1. The residual current, determined by subtracting the expected current from the measured current, must be smaller or equal than ± 20 nA.
2. The inverted slope of the measured current versus the applied potential must be equal to 1000100 Ω± 5 %.
3. The intercept of the measured current versus the applied potential must be equal to ± 4 mV divided by 1000100 Ω± 5 %.

All three conditions must be valid for the test to succeed.

16.2.2.6.2 Autolab PGSTAT302F testing in Floating mode

NOVA is shipped with a procedure which can be used, alongside the **Diagnostics** application, to verify that the instrument is working as expected.

NOTE

For more information on the **Diagnostics** application, please refer to *Chapter 17*.

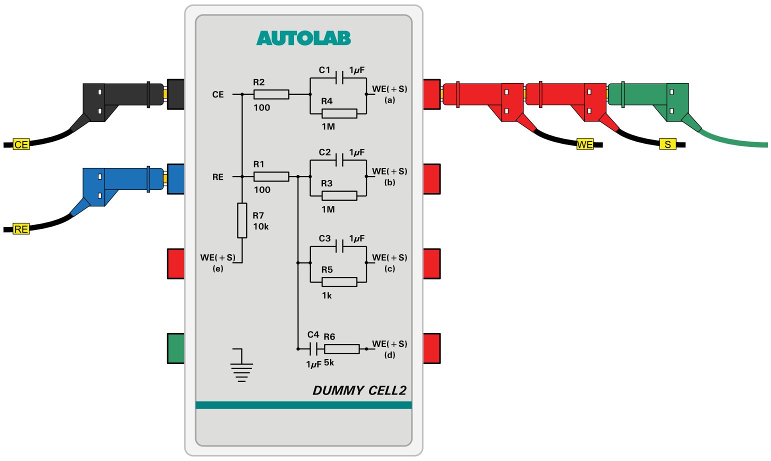
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestCV PGSTAT302F** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test\TestCV PGSTAT302F.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (a).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1079*.

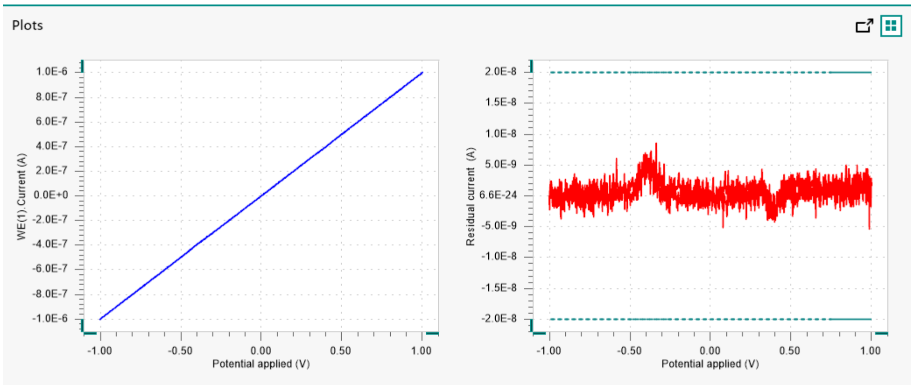
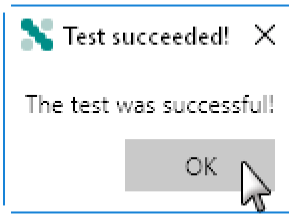


Figure 1079 The data measured by the TestCV procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestCV automatic evaluation of the data requires the following tests to succeed:

1. The residual current, determined by subtracting the expected current from the measured current, must be smaller or equal than ± 20 nA.
2. The inverted slope of the measured current versus the applied potential must be equal to 1000100 Ω± 5 %.
3. The intercept of the measured current versus the applied potential must be equal to ± 4 mV divided by 1000100 Ω± 5 %.

All three conditions must be valid for the test to succeed.

16.2.2.7 Autolab F Series specifications

The specifications of the F Series Autolab are provided in *Table 28*.

Table 28 Specifications of the F Series Autolab instruments

Instrument	PGSTAT302F
Maximum current	± 2 A
Compliance voltage	± 30 V, ± 10 V (with default cables)
Potential range	± 10 V

Instrument	PGSTAT302F
Applied potential accuracy	± 0.2 % ± 2 mV
Applied potential resolution	150 µV
Measured potential resolution	300 nV (gain 1000)
Current ranges	10 nA to 1 A, 9 decades
Current accuracy	± 0.2 % of current range
Applied current resolution	0.015 % of current range
Measured current resolution	0.00003 % of current range (gain 1000)
Potentiostat bandwidth	100 kHz
Potentiostat rise/fall time	< 250 ns
Input impedance of electrometer	> 1 TΩ, 8 pF
Input bias current	< 1 pA
Electrometer bandwidth	> 4 MHz
iR compensation	2 Ω - 200 MΩ
iR compensation resolution	0.025 %
Analog output	Potential and current
Analog voltage input	Yes
External inputs	2
External outputs	2

Instrument	PGSTAT302F
Digital input/ output	48
Interface	USB
Warm-up time	30 minutes
Pollution degree	2
Installation category	II
External dimensions (without cables and accessories)	52x42x16 cm^3
Weight	18 kg
Power requirements	300 W
Power supply	100 - 240 V ± 10% in four ranges 100 V: [90 - 121 V] 120 V: [104 - 139 V] 230 V: [198 - 242 V] 240 V: [207 - 264 V]
Power line frequency	47-63 Hz
Fuse	100 V, 120 V: 3.15 A (slow-slow) 230 V, 240 V: 1.6 A (slow-slow)
Operating environment	0 °C to 40 °C, 80 % relative humidity without derating
Storage environment	-10 °C to 60 °C

16.2.3 Autolab MBA N Series (AUT8) instruments

The Autolab MBA N Series is a special version of the modular potentiostat/ galvanostat produced by Metrohm Autolab. These instruments, identified by a serial number starting with **AUT8**, are based on a modular concept that allows the instrument to be complemented by internal or external extension modules.

The following instruments belong to the Autolab MBA N Series:

- **Autolab PGSTAT302N MBA:** modular PGSTAT with 30 V compliance and 2 A maximum current.
- **Autolab PGSTAT128N MBA:** modular PGSTAT with 12 V compliance and 800 mA maximum current.

NOTE

The **Autolab MBA N Series** derives from the **Autolab N Series**. This chapter will only provide details on instrumental properties that deviate from the properties of the **Autolab N Series**. Please refer to *Chapter 16.2.1* for additional information on the common properties of the instruments.

Unlike the Autolab N Series instruments which can accommodate a wide range of internal extension modules, the Autolab N MBA Series can only accommodate the following modules:

- **FRA32M or FRA2 module:** impedance spectroscopy module (please refer to *Chapter 16.3.2.13* and *Chapter 16.3.2.12* for more information).
- **BA module:** dual mode bipotentiostat module. Up to five BA modules can be placed in each MBA instrument *(see Chapter 16.3.2.3, page 990)*.

The BA modules installed in the MBA instrument can be used to control up to five additional working electrodes, sharing a common counter electrode and reference electrode. These instruments can therefore be used to work with sensor arrays or with electrochemical cells in which more than one working electrode is located.

Each of the five BA modules are identified by a specified MBA module label *(see Figure 1080, page 905)*.

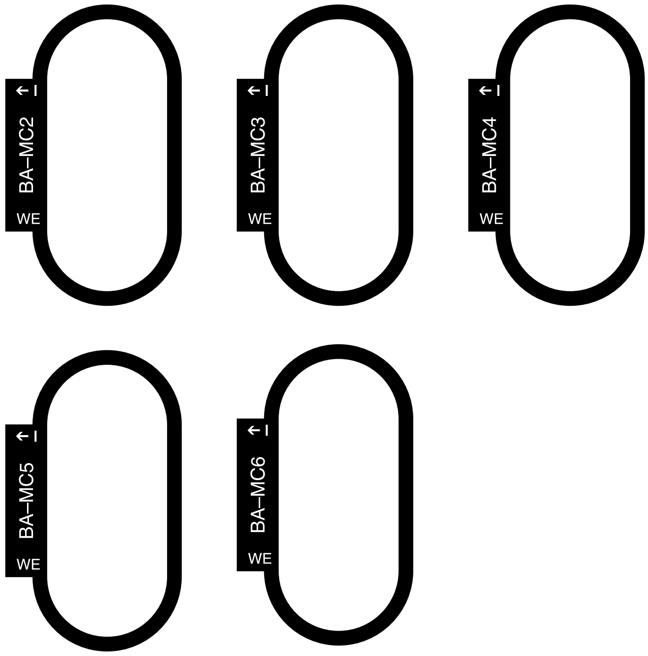


Figure 1080 The module labels used to identify the BA modules installed in a MBA instrument

16.2.3.1 N MBA Series Autolab front panel

NOTE

The front panel of the **Autolab MBA N Series** instrument is arranged differently from the **Autolab N Series** instruments.

The front panel of the N MBA Series Autolab provides a number of connections, controls and indicators *(see Figure 1081, page 905)*.

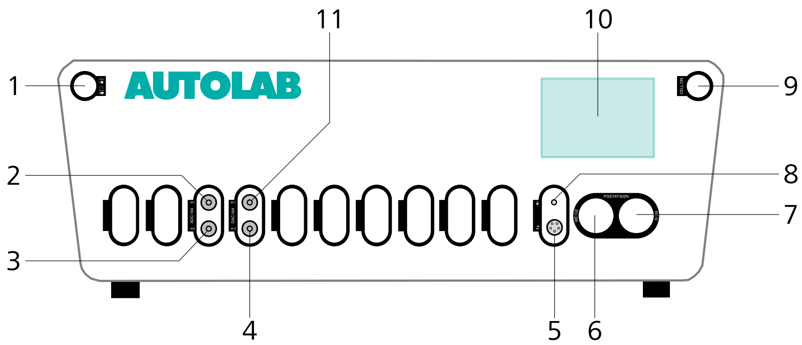


Figure 1081 Overview of the front panel of the N MBA Series Autolab

1 On/Off button
For switching the Autolab on or off.

2 ADC164 →1
Analog input for recording external signals (ADC164 →1).

3	**ADC164 →2** Analog input for recording external signals (ADC164 →2).	**4**	**DAC164 ←2** Analog output for controlling external signals (DAC164 ←2).
5	**Monitor cable connector ⇄** For connecting the monitor cable.	**6**	**CE/WE connector** For connecting the Autolab cell cable, providing connections to the counter electrode (CE), working electrode (WE) and ground.
7	**RE/S connector** For connecting the Autolab differential amplifier, providing connections to the reference electrode (RE) and sense electrode (S).	**8**	**Ground connector** Additional ground connector for connecting external devices to the Autolab ground.
9	**Cell ON button** For enabling and disabling the cell.	**10**	**Display** Display indicating real-time information on the measured current and potential and instrumental settings.
11	**DAC164 ←1** Analog output for controlling external signals (DAC164 ←1).		

The display (item 10 in *Figure 1081*) is identical to the display of the Autolab N Series *(see Figure 1070, page 882)*.

16.2.4 Autolab Compact Series (AUT4/AUT5) instruments

The Autolab Compact Series provides potentiostat/galvanostat instruments with a very small footprint.

The following instruments belong to the Autolab Compact Series:

- **Autolab PGSTAT101:** PGSTAT with 10 V compliance and 100 mA maximum current, identified with a serial number starting with **AUT4**. The PGSTAT101 can be complemented by **external** extension modules only.
- **Autolab PGSTAT204:** modular PGSTAT with 20 V compliance and 400 mA maximum current, identified with a serial number starting with **AUT5**. The PGSTAT204 can be complemented by a selection of **internal** and **external** extension modules.

16.2.4.1 Compact Series Autolab scope of delivery

The Compact Series Autolab systems are supplied with the following items:

- Autolab potentiostat/galvanostat
- Cell cable (RE/S/WE/CE/GND)
- Power cable
- USB cable
- Set of four alligator clips

The PGSTAT204 is also supplied with the Autolab dummy cell.

The monitor cable, used to interface to external devices, is also available for the Compact Series Autolab, as an option.

16.2.4.2 Instrument power-up state

The power-up state of the instrument is hardware defined. The following settings are automatically selected whenever the instrument is powered on or whenever the connection to the instrument is reset by the software.

- Cell: off
- Mode: potentiostatic
- Control bandwidth: high stability
- iR compensation: off
- Current range: 1 µA
- Optional modules: off
- DIO ports: low state
- Summation point inputs: off
- Internal dummy cell: off

16.2.4.3 Autolab Compact Series front panel

The front panel of the Autolab PGSTAT101 provides a number of connections and indicators *(see Figure 1082, page 908)*.

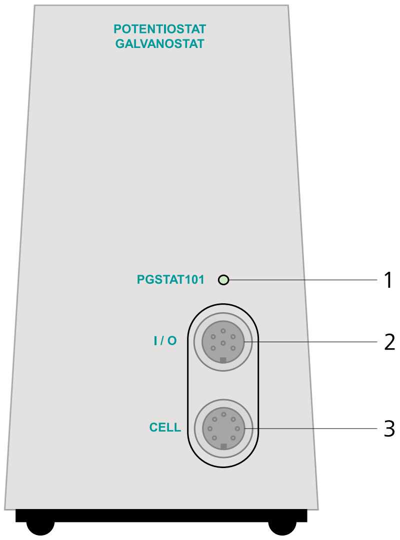


Figure 1082 The front panel of the PGSTAT101

1 Status LED
Dual color LED used to indicate the status of the PGSTAT101. When the LED is red, an overload is detected. When the LED is green, the cell is on and no overloads are detected. When the LED is switched off, the cell is off and no overloads are detected.

2 I/O connector
Used to connect the optional monitor cable providing connections for E_{out}, I_{out}, V_{out} and V_{in}.

3 CELL connector
Used to connect the cell cable providing connections for the counter (CE), reference (RE), working (WE) and sense (S) electrode as well as the ground.

The front panel of the Autolab PGSTAT204 provides a number of connections and indicators *(see Figure 1083, page 909)*.

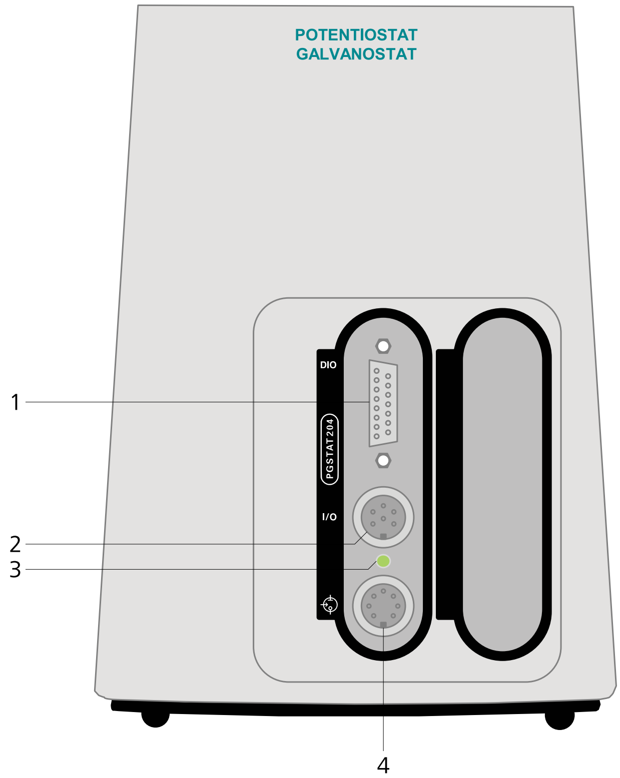


Figure 1083 The front panel of the PGSTAT204

1 DIO connector
For connecting a DIO cable or interfacing to external devices through the digital inputs and outputs.

2 I/O connector
Used to connect the optional monitor cable providing connections for E_{out}, I_{out}, V_{out} and V_{in}.

3 Status LED
Dual color LED used to indicate the status of the PGSTAT204. When the LED is red, an overload is detected. When the LED is green, the cell is on and no overloads are detected. When the LED is switched off, the cell is off and no overloads are detected.

4 CELL connector
Used to connect the cell cable providing connections for the counter (CE), reference (RE), working (WE) and sense (S) electrode as well as the ground. The cell is represented by the symbol ⯐.

16.2.4.4 Autolab Compact Series back plane

The back plane of the Autolab PGSTAT101 provides a number of connections and controls, shown in *Figure 1084*.

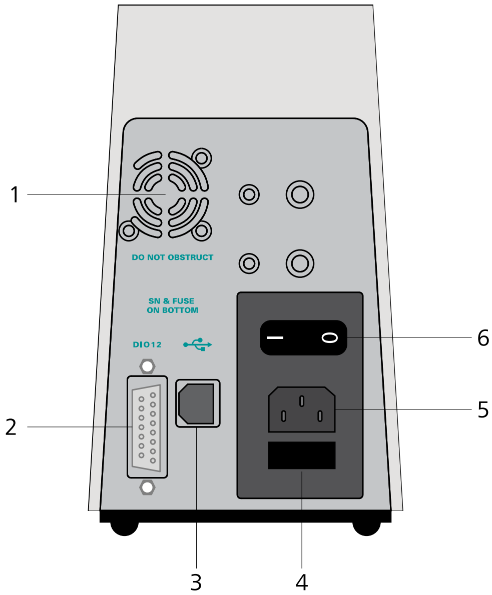


Figure 1084 The back plane of the PGSTAT101

1	**Fan** Required for cooling the Autolab during operation.	**2**	**DIO port** For connecting a DIO cable or interfacing to external devices through the digital inputs and outputs.
3	**USB connector** Type B USB plug for connecting the USB cable to the host computer.	**4**	**Fuse holder** Holds the mains connection socket fuse.
5	**Mains connection socket** For connecting the Autolab to the mains supply.	**6**	**On/Off switch** For switching the Autolab on or off.

The back plane of the Autolab PGSTAT204 provides a number of connections and controls, shown in *Figure 1085*.

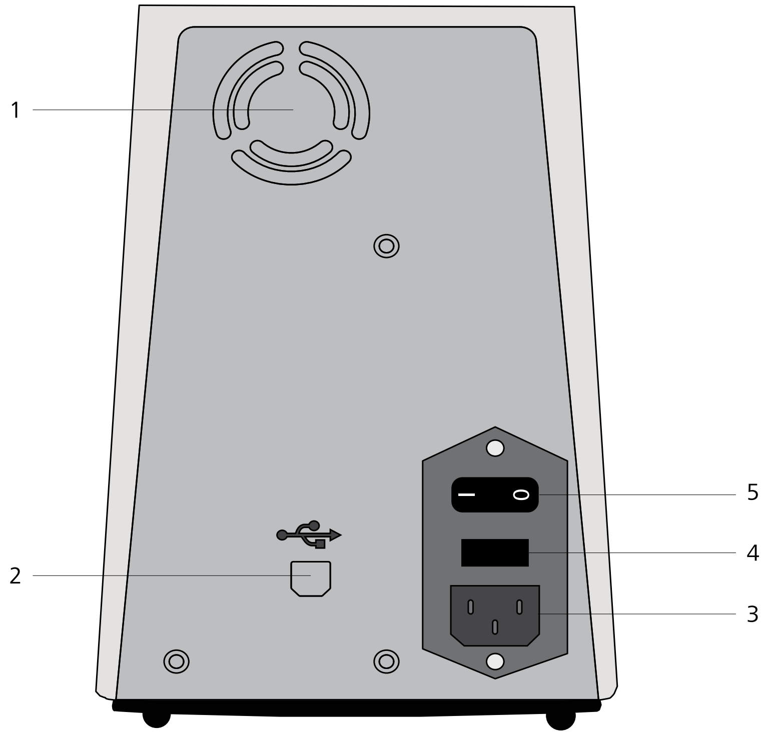


Figure 1085 The back plane of the PGSTAT204

1	**Fan** Required for cooling the Autolab during operation.	**2**	**USB connector** Type B USB plug for connecting the USB cable to the host computer.
3	**Mains connection socket** For connecting the Autolab to the mains supply.	**4**	**Fuse holder** Holds the mains connection socket fuse.
5	**On/Off switch** For switching the Autolab on or off.		

16.2.4.5 Connections for analog signals

The Autolab Compact Series instruments provide connections for analog signals through an optional monitor cable.

NOTE

The monitor cable is not supplied with the instrument and it need to be ordered separately. A dedicated cable is available for each instrument type.

CAUTION

Avoid creating ground loops when connecting the Autolab to external signals as this will degrade the performance of the instrument.

16.2.4.5.1 Monitor cable for Autolab PGSTAT101

The **monitor cable** for **PGSTAT101** provides additional connections for analog signals, through BNC connectors. All the connections are with respect to the Autolab ground directly and indirectly with respect to the protective earth *(see Figure 1086, page 912)*.

Figure 1086 The monitor cable for PGSTAT101

To use the monitor cable, connect the cable to the matching connector located on the front panel of the Autolab. This connector, labeled **I/O**, is located on the front panel of the instrument (item 2 in *Figure 1082*).

The following connections are provided through the monitor cable:

- **Eout:** this output corresponds to the differential potential of the reference electrode (RE) with respect to the sense electrode (S). The output voltage will vary between ± 10 V. The output impedance is 1 k Ω, so a correction should be made if a load smaller than 2 MΩ is connected to this output. The minimum load is 4000 Ω.
- **iout:** this output corresponds to the inverted output of the current-to-voltage converter circuit of the Autolab. The output corresponds to the inverted measured current divided by the current range. The output voltage will vary between ± 10 V. The output impedance is 50 Ω, so a correction should be made if a load smaller than 100 kΩ is connected to this output. The minimum load is 200 Ω.

- **Vout:** this output corresponds to the output of the on-board DAC of the PGSTAT101. It can be used to generate any analog signal with a ± 10 V value range. The output impedance is 1 Ω and this output is capable of supplying a current of up to 5 mA. Corrections should be made with loads smaller than 2 kΩ. More information on the on-board DAC is provided in *Chapter 16.3.1.2*.
- **Vin:** this input corresponds to the input of the on-board ADC of the PGSTAT101. It can be used to record any analog signal with a ± 10 V value range. The input impedance is ≥ 1 GΩ. More information on the on-board ADC is provided in *Chapter 16.3.1.1*.

16.2.4.5.2 Monitor cable for Autolab PGSTAT204

The **monitor cable** for **PGSTAT204** provides additional connections for analog signals, through BNC connectors. All the connections are with respect to the Autolab ground directly and indirectly with respect to the protective earth *(see Figure 1087, page 913)*.

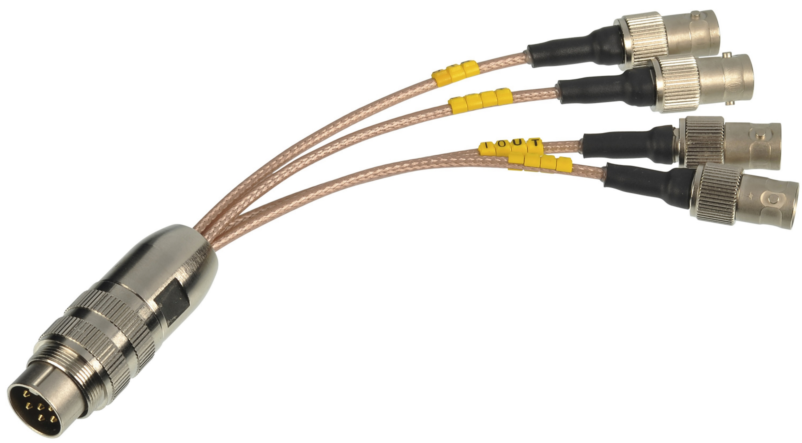


Figure 1087 The monitor cable for PGSTAT204

To use the monitor cable, connect the cable to the matching connector located on the front panel of the Autolab. This connector, labeled **I/O**, is located on the front panel of the instrument (item 2 in *Figure 1083*).

The following connections are provided through the monitor cable:

- **Eout:** this output corresponds to the differential potential of the reference electrode (RE) with respect to the sense electrode (S). The output voltage will vary between ± 10 V. The output impedance is 1 k Ω, so a correction should be made if a load smaller than 2 MΩ is connected to this output. The minimum load is 4000 Ω.

- **iout:** this output corresponds to the inverted output of the current-to-voltage converter circuit of the Autolab. The output corresponds to the inverted measured current divided by the current range. The output voltage will vary between ± 10 V. The output impedance is 50 Ω, so a correction should be made if a load smaller than 100 kΩ is connected to this output. The minimum load is 200 Ω.
- **Vout:** this output corresponds to the output of the on-board DAC of the PGSTAT204. It can be used to generate any analog signal with a ± 10 V value range. The output impedance is 1 Ω and this output is capable of supplying a current of up to 5 mA. Corrections should be made with loads smaller than 2 kΩ. More information on the on-board DAC is provided in *Chapter 16.3.1.2*.
- **Vin:** this input corresponds to the input of the on-board ADC of the PGSTAT204. It can be used to record any analog signal with a ± 10 V value range. The input impedance is ≥ 1 GΩ. More information on the on-board ADC is provided in *Chapter 16.3.1.1*.

16.2.4.6 Compact Autolab Series restrictions

Restrictions apply when using the Compact Autolab Series potentiostat/galvanostat:

- **Intended use:** the Compact Autolab Series potentiostat/galvanostat is intended to be used for electrochemical research only.
- **Service:** there are **no** serviceable parts inside. Servicing of the instrument can only be carried out by qualified personnel.

CAUTION

All attempts to service the instrument will lead to the immediate voiding of any warranty.

16.2.4.7 Compact Series Autolab testing

The Autolab PGSTAT101 and PGSTAT204 can be tested using the following procedures:

1. For the Autolab PGSTAT101, a dedicated test with the internal dummy cell is available. Please refer to *Chapter 16.2.4.7.1* for more information.
2. For the Autolab PGSTAT204, the standard TestCV procedure with the Autolab dummy cell. Please refer to *Chapter 16.2.4.7.2* for more information.

16.2.4.7.1 Autolab PGSTAT101 testing

NOVA is shipped with a procedure which can be used, alongside the **Diagnostics** application, to verify that the instrument is working as expected.

NOTE

For more information on the **Diagnostics** application, please refer to *Chapter 17*.

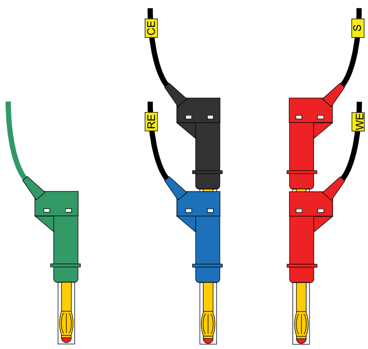
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestCV PGSTAT101** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test\TestCV PGSTAT101.nox)

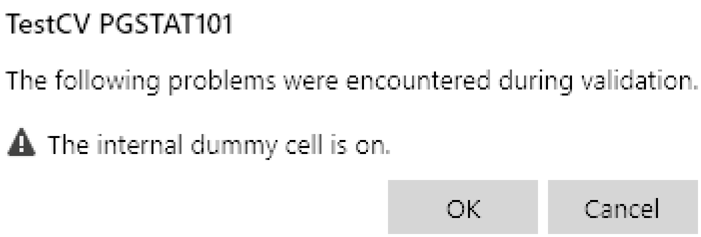
2 Connect the cell cable connectors

Connect the counter electrode (CE) and reference electrode (RE) together and the working electrode (WE) and sense electrode (S) together.



3 Ignore the warning

Ignore the warning message shown when the procedure is loaded and when the procedure is started.



4 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1088*.

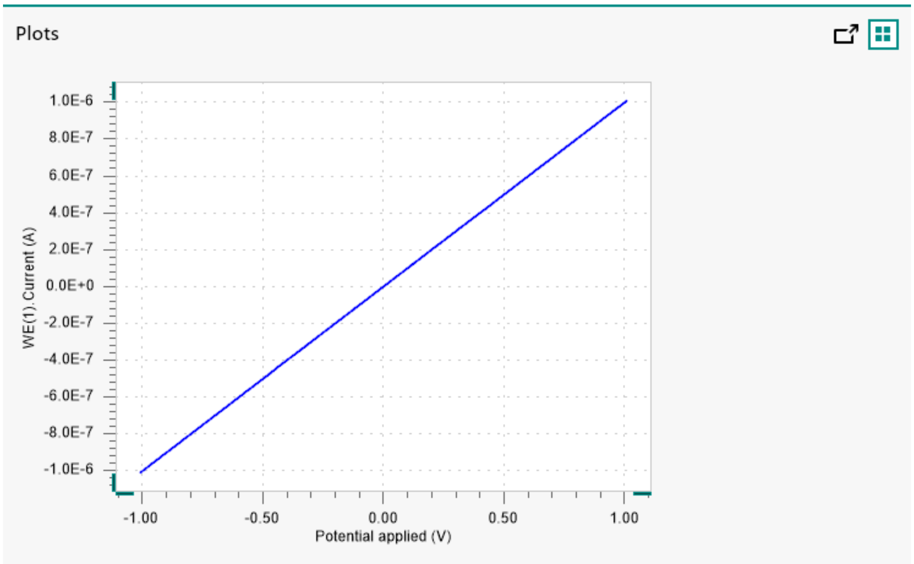
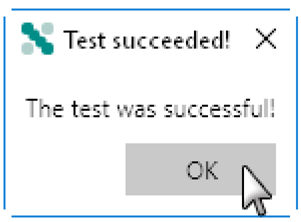


Figure 1088 The data measured by the TestCV procedure

5 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestCV automatic evaluation of the data requires the following tests to succeed:

1. The inverted slope of the measured current versus the applied potential must be equal to 1000100 Ω± 5 %.
2. The intercept of the measured current versus the applied potential must be equal to ± 5 mV divided by 1000100 Ω± 5 %.

Both conditions must be valid for the test to succeed.

16.2.4.7.2 Autolab PGSTAT204 testing

NOVA is shipped with a procedure which can be used, alongside the **Diagnostics** application, to verify that the instrument is working as expected.

NOTE

For more information on the **Diagnostics** application, please refer to *Chapter 17*.

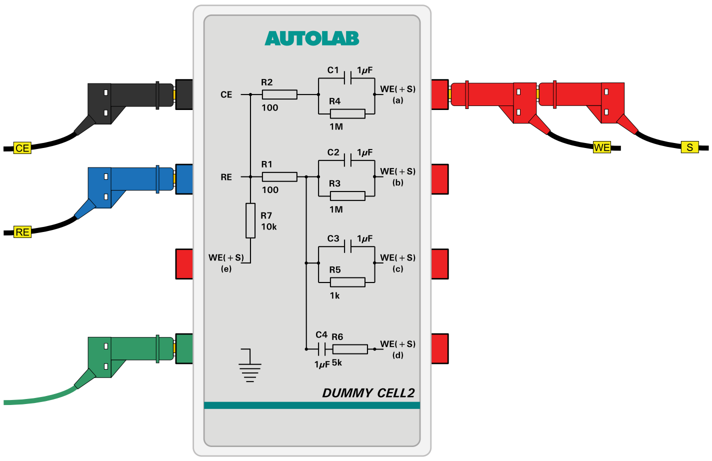
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestCV** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test \TestCV.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (a).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1089*.

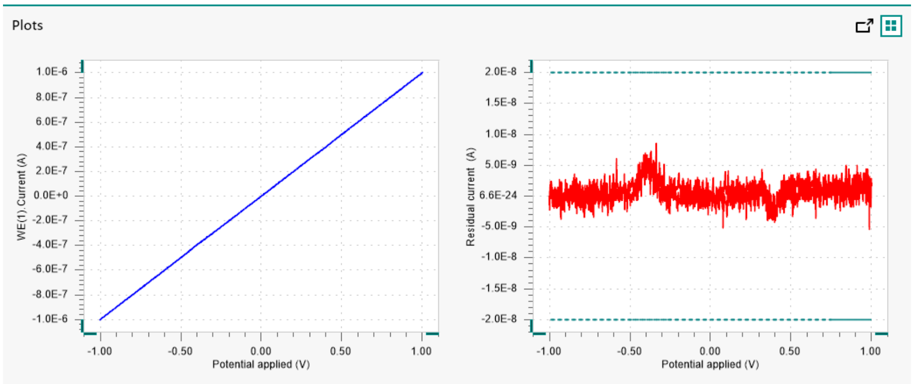
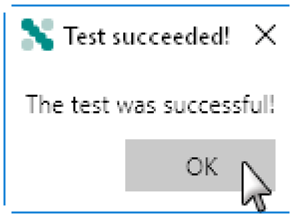


Figure 1089 The data measured by the TestCV procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestCV automatic evaluation of the data requires the following tests to succeed:

1. The residual current, determined by subtracting the expected current from the measured current, must be smaller or equal than ± 20 nA.
2. The inverted slope of the measured current versus the applied potential must be equal to 1000100 Ω± 5 %.
3. The intercept of the measured current versus the applied potential must be equal to ± 4 mV divided by 1000100 Ω± 5 %.

All three conditions must be valid for the test to succeed.

16.2.4.8 Autolab Compact Series specifications

The specifications of the Autolab Compact Series are provided in *Table 29*.

Table 29 Specifications of the Autolab Compact Series instruments

Instrument	PGSTAT101	PGSTAT204
Maximum current	± 100 mA	± 400 mA
Compliance voltage	± 10 V	± 20 V

Instrument	PGSTAT101	PGSTAT204
Potential range	± 10 V	
Applied potential accuracy	± 0.2 % ± 2 mV	
Applied potential resolution	150 µV	
Measured potential resolution	300 nV (gain 1000)	
Current ranges	10 nA to 10 mA, 7 decades	10 nA to 100 mA, 8 decades
Current accuracy	± 0.2 % of current range	
Applied current resolution	0.015 % of current range	
Measured current resolution	0.00003 % of current range (gain 1000)	
Potentiostat bandwidth	1 MHz	
Potentiostat rise/fall time	< 300 ns	
Input impedance of electrometer	> 100 GΩ, 8 pF	
Input bias current	< 1 pA	
Electrometer bandwidth	> 4 MHz	
iR compensation	20 mΩ - 200 MΩ	200 mΩ - 200 MΩ
iR compensation resolution	0.025 %	
Analog output	Potential and current	
Analog voltage input	Yes	
External inputs	2	
External outputs	2	
Digital input/output	12	
Interface	USB	
Warm-up time	30 minutes	
Pollution degree	2	

Instrument	PGSTAT101	PGSTAT204
Installation category	II	
External dimensions (without cables and accessories)	9x21x15 cm^3	15x26x20 cm^3
Weight	2.1 kg	4.1 kg
Power requirements	40 W	75 W
Power supply	100 - 240 V ± 10% in four ranges (auto select)	
Power line frequency	47-63 Hz	
Fuse	2 A (slow-slow)	3.5 A (slow-slow)
Operating environment	0 °C to 40 °C, 80 % relative humidity without derating	
Storage environment	-10 °C to 60 °C	

16.2.5 Multi Autolab Series (MAC8/MAC9) instruments

The Multi Autolab Series provides cabinets that can accommodate up to 12 potentiostat/galvanostat channels or a combination of potentiostat/galvanostat modules with expansion modules.

The following instruments belong to the Multi Autolab Series:

- **Multi Autolab M101:** Multi Autolab cabinet designed to accommodate up to 12 M101 potentiostat/galvanostat modules with 10 V compliance and 100 mA maximum current. The cabinet is identified with a serial number starting with **MAC8**. The M101 can be complemented by **internal** or **external** extension modules only.
- **Multi Autolab M204:** Multi Autolab cabinet designed to accommodate up to 12 M204 potentiostat/galvanostat modules with 20 V compliance and 400 mA maximum current. The cabinet is identified with a serial number starting with **MAC9**. The M204 can be complemented by **internal** or **external** extension modules only.

CAUTION

M101 modules can only be installed in a M101 Multi Autolab cabinet.

M204 modules can only be installed in a M204 Multi Autolab cabinet.

The Multi Autolab cabinet is fitted with twelve module bays, labeled 1 to 6 (from left to right) and A to F (from left to right). Potentiostat/galvanostat modules can be installed in any available module bays. Internal expansion module can be installed in one of the six (A-F) module bays if the module immediately on the left is fitted with a potentiostat/galvanostat module *(see Figure 1090, page 921)*.

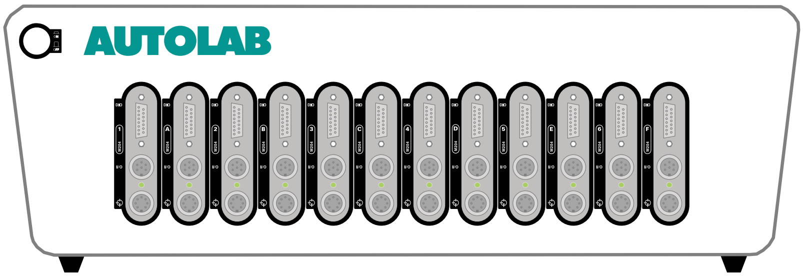


Figure 1090 The Multi Autolab cabinet can accommodate up to 12 potentiostat/galvanostat modules

16.2.5.1 Multi Autolab Series scope of delivery

The Multi Autolab Series systems are supplied with the following items:

- Multi Autolab cabinet
- Autolab potentiostat/galvanostat module (M101 or M204)
- Cell cable (RE/S/WE/CE/GND), (one per M101 or M204)
- Power cable
- 3 USB cables
- Set of four alligator clips, (one per M101 or M204)
- Autolab dummy cell

The monitor cable, used to interface to external devices, is also available for the Multi Autolab Series, as an option.

16.2.5.2 Instrument power-up state

The power-up state of the instrument is hardware defined. The following settings are automatically selected whenever the instrument is powered on or whenever the connection to the instrument is reset by the software.

- Cell: off
- Mode: potentiostatic
- Control bandwidth: high stability
- iR compensation: off
- Current range: 1 µA
- Optional modules: off
- DIO port: low state
- Summation point inputs: off

16.2.5.3 Multi Autolab Series front panel

The front panel of the Multi Autolab M101 provides a number of connections and indicators *(see Figure 1091, page 922)*.

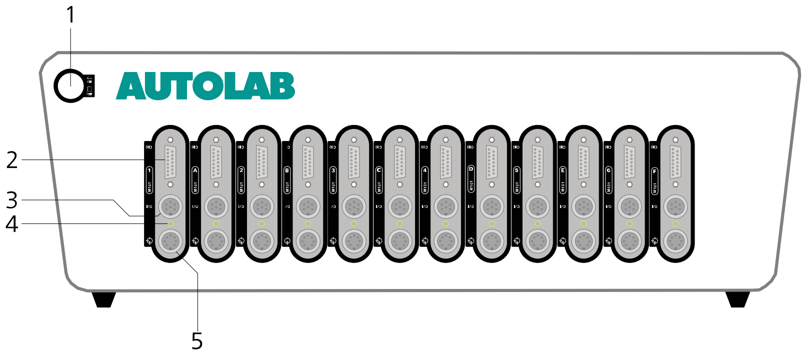


Figure 1091 The front panel of the Multi Autolab M101

1 On/Off button
For switching the Multi Autolab on or off.

2 DIO connector
For connecting a DIO cable or interfacing to external devices through the digital inputs and outputs.

3 I/O connector
Used to connect the optional monitor cable providing connections for E_{out}, I_{out}, V_{out} and V_{in}.

4 Status LED
Dual color LED used to indicate the status of the M101. When the LED is red, an overload is detected. When the LED is green, the cell is on and no overloads are detected. When the LED is switched off, the cell is off and no overloads are detected.

5 CELL connector
Used to connect the cell cable providing connections for the counter (CE), reference (RE), working (WE) and sense (S) electrode as well as the ground. The cell is represented by the symbol ⯐.

The front panel of the Multi Autolab M204 provides a number of connections and indicators *(see Figure 1092, page 923)*.

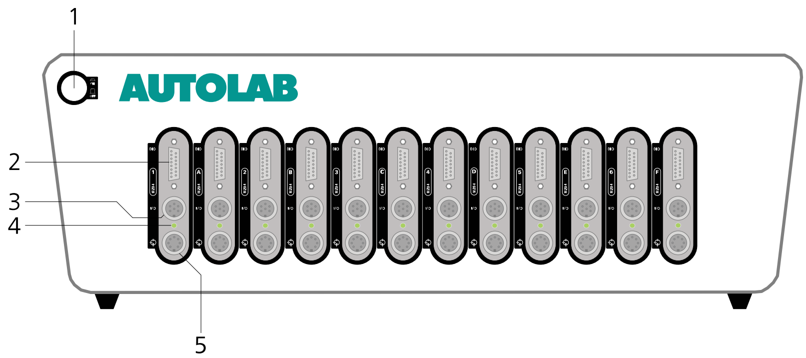


Figure 1092 The front panel of the Multi Autolab M204

1 On/Off button
For switching the Multi Autolab on or off.

2 DIO connector
For connecting a DIO cable or interfacing to external devices through the digital inputs and outputs.

3 I/O connector
Used to connect the optional monitor cable providing connections for E_{out}, I_{out}, V_{out} and V_{in}.

4 Status LED
Dual color LED used to indicate the status of the M204. When the LED is red, an overload is detected. When the LED is green, the cell is on and no overloads are detected. When the LED is switched off, the cell is off and no overloads are detected.

5 CELL connector
Used to connect the cell cable providing connections for the counter (CE), reference (RE), working (WE) and sense (S) electrode as well as the ground. The cell is represented by the symbol ⯐.

16.2.5.4 Multi Autolab Series back plane

The back plane of the Multi Autolab provides a number of connections and controls, shown in *Figure 1093*.

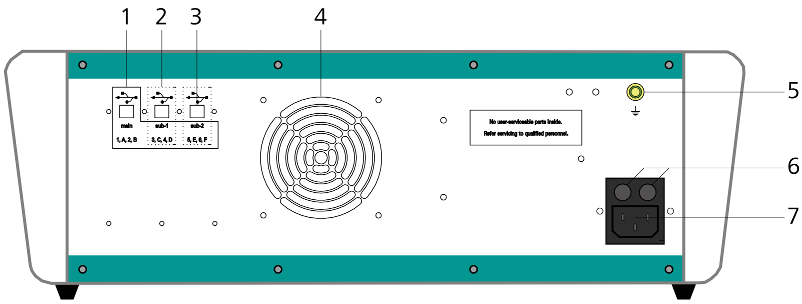


Figure 1093 The back plane of the Multi Autolab

1 USB hub main connector
Type B USB plug for connecting the USB cable to the host computer providing connections to all modules.

2 USB hub sub-1 connector
Type B USB plug for connecting the USB cable to the host computer providing connections module bays 3, C, 4 and D.

3 USB hub sub-2 connector
Type B USB plug for connecting the USB cable to the host computer providing connections module bays 5, E, 6 and F.

4 Fan
Required for cooling the Multi Autolab during operation.

5 Ground plug
For grounding the Multi Autolab cabinet

6 Fuse holders
Holds the mains connection socket fuses.

7 Mains connection socket
For connecting the Autolab to the mains supply.

16.2.5.5 Connections for analog signals

The Multi Autolab Series instruments provide connections for analog signals through an optional monitor cable.

NOTE

The monitor cable is not supplied with the instrument and it need to be ordered separately.

CAUTION

Avoid creating ground loops when connecting the Autolab to external signals as this will degrade the performance of the instrument.

The **monitor cable** for **Multi Autolab M101** and **Multi Autolab M204** provides additional connections for analog signals, through BNC connectors. All the connections are with respect to the Autolab ground directly and indirectly with respect to the protective earth *(see Figure 1087, page 913)*.

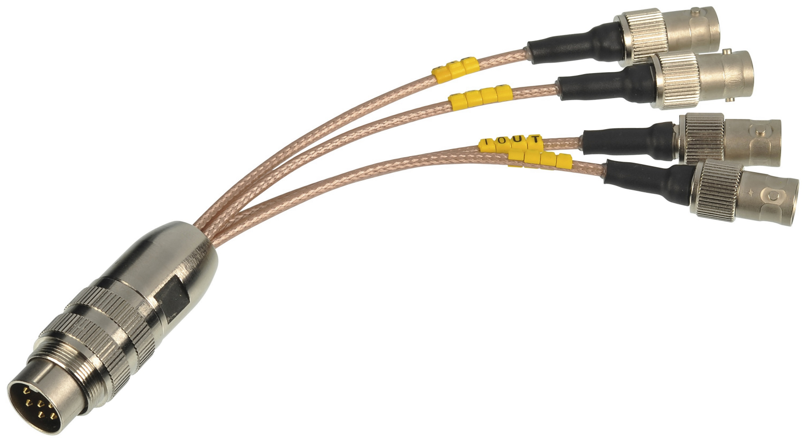


Figure 1094 The monitor cable for Multi Autolab M101 and Multi Autolab M204

To use the monitor cable, connect the cable to the matching connector located on the front panel of the M101 or M204 module in the Multi Autolab instrument. This connector, labeled **I/O**, is located on the front panel of the potentiostat/galvanostat module (item 3 in *Figure 1091* and item 3 in *Figure 1092*).

The following connections are provided through the monitor cable:

- **Eout:** this output corresponds to the differential potential of the reference electrode (RE) with respect to the sense electrode (S). The output voltage will vary between ± 10 V. The output impedance is 1 k Ω, so a correction should be made if a load smaller than 2 MΩ is connected to this output. The minimum load is 4000 Ω.
- **iout:** this output corresponds to the inverted output of the current-to-voltage converter circuit of the Autolab. The output corresponds to the inverted measured current (for the M101) or the measured current (for the M204) divided by the current range. The output voltage will vary between ± 10 V. The output impedance is 50 Ω, so a correction should be made if a load smaller than 100 kΩ is connected to this output. The minimum load is 200 Ω.
- **Vout:** this output corresponds to the output of the on-board DAC of the M101 or M204. It can be used to generate any analog signal with a ± 10 V value range. The output impedance is 1 Ω and this output is capable of supplying a current of up to 5 mA. Corrections should be made with loads smaller than 2 kΩ. More information on the on-board DAC is provided in *Chapter 16.3.1.2*.

- **Vin:** this input corresponds to the input of the on-board ADC of the M101 or M204. It can be used to record any analog signal with a ± 10 V value range. The input impedance is ≥ 1 GΩ. More information on the ADC164 is provided in *Chapter 16.3.1.1*.

16.2.5.6 Multi Autolab Series connection hub

The Multi Autolab cabinet is fitted with an internal USB hub that allows the Multi Autolab to be shared in between a maximum of three computers running NOVA.

When the Multi Autolab is connected to the **main** USB connector, the host computer controls all twelve module bays. If an additional host computer is connected to the either one of the sub USB connectors (**sub-1** or **sub-2**), that computer will take over the control of the module bays indicated in the labels below the sub USB connectors *(see Figure 1095, page 926)*.

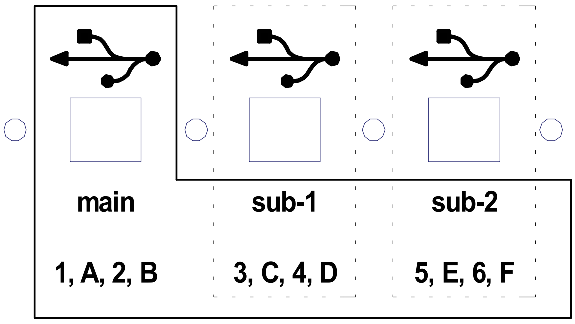


Figure 1095 The Multi Autolab connection hub

CAUTION

Never change the USB connections while an experiment is running on any of the Multi Autolab channels since this could lead to a loss of control of an experiment and a loss of data.

Depending on the connection to the USB hub, the host computer can control the following module bays:

- **main USB:** all the module bays in the Multi Autolab cabinet are controlled.
- **sub-1 USB:** module bays 3, C, 4 and D are controlled.
- **sub-2 USB:** module bays 5, E, 6 and F are controlled.

For example, connecting a computer to the sub-2 USB connector on the back plane will provide control over the modules installed in bays 5, E, 6 and F, exclusively. Connecting an additional computer to the main USB connector will provide control over all the remaining modules (1, A, 2, B, 3, C, 4, D). If a third PC is connected to the sub-1 USB connector, control over the modules in bays 3, C, 4 and D will be transferred from the computer connected to the main USB connector to the computer connected to the sub-1 connector.

16.2.5.7 Multi Autolab Series restrictions

Restrictions apply when using the Multi Autolab Series potentiostat/galvanostat:

- **Intended use:** the Multi Autolab potentiostat/galvanostat is intended to be used for electrochemical research only.
- **Service:** there are **no** serviceable parts inside. Servicing of the instrument can only be carried out by qualified personnel.

CAUTION

All attempts to service the instrument will lead to the immediate voiding of any warranty.

16.2.5.8 Multi Autolab Series Autolab testing

NOVA is shipped with a procedure which can be used, alongside the **Diagnostics** application, to verify that the instrument is working as expected.

NOTE

For more information on the **Diagnostics** application, please refer to *Chapter 17*.

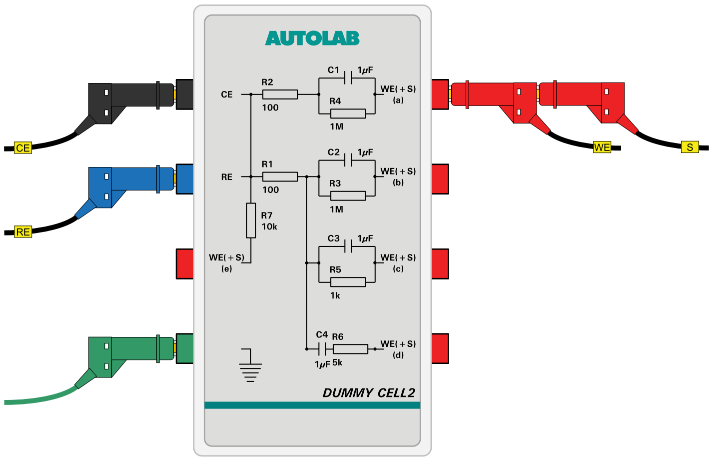
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestCV** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test \TestCV.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (a).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1096*.

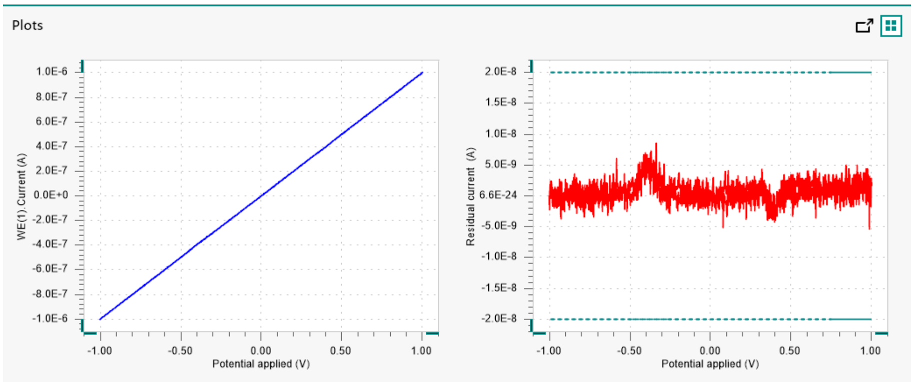
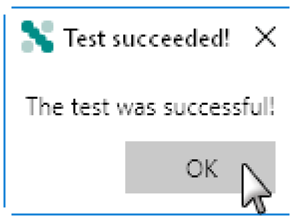


Figure 1096 The data measured by the TestCV procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestCV automatic evaluation of the data requires the following tests to succeed:

1. The residual current, determined by subtracting the expected current from the measured current, must be smaller or equal than ± 20 nA.
2. The inverted slope of the measured current versus the applied potential must be equal to 1000100 Ω± 5 %.
3. The intercept of the measured current versus the applied potential must be equal to ± 4 mV divided by 1000100 Ω± 5 %.

All three conditions must be valid for the test to succeed.

16.2.5.9 Multi Autolab Series specifications

The specifications of the Multi Autolab Series are provided in *Table 30*.

Table 30 Specifications of the Multi Autolab Series instruments

Instrument	M101	M204
Maximum current	± 100 mA	± 400 mA
Compliance voltage	± 10 V	± 20 V
Potential range	± 10 V	
Applied potential accuracy	± 0.2 % ± 2 mV	
Applied potential resolution	150 µV	
Measured potential resolution	300 nV (gain 1000)	
Current ranges	10 nA to 10 mA, 7 decades	10 nA to 100 mA, 8 decades
Current accuracy	± 0.2 % of current range	
Applied current resolution	0.015 % of current range	
Measured current resolution	0.00003 % of current range (gain 1000)	
Potentiostat bandwidth	1 MHz	
Potentiostat rise/fall time	< 300 ns	
Input impedance of electrometer	> 100 GΩ, 8 pF	
Input bias current	< 1 pA	
Electrometer bandwidth	> 4 MHz	
iR compensation	20 mΩ - 200 MΩ	200 mΩ - 200 MΩ

Instrument	M101	M204
iR compensation resolution	0.025 %	
Analog output	Potential and current	
Analog voltage input	Yes	
External inputs	2	
External outputs	2	
Digital input/output	12	
Interface	USB	
Warm-up time	30 minutes	
Pollution degree	2	
Installation category	II	
External dimensions (without cables and accessories)	52x42x17 cm^3	
Weight	13 kg	14 kg
Power requirements	200 W	700 W
Power supply	100 - 240 V ± 10% in four ranges (auto select)	
Power line frequency	47-63 Hz	
Fuse	8 A (slow-slow)	8 A (slow-slow)
Operating environment	0 °C to 40 °C, 80 % relative humidity without derating	
Storage environment	-10 °C to 60 °C	

16.2.6 Autolab 7 Series (AUT7) instruments

The Autolab 7 Series is the predecessor version of the Autolab N Series modular potentiostat/galvanostat produced by Metrohm Autolab *(see Chapter 16.2.1, page 879)*. These instruments, identified by a serial number starting with **AUT7**, are based on modular concept that allows the instrument to be complemented by internal or external extension modules.

NOTE

The Autolab 7 Series instruments are no longer available.

The following instruments belong to the Autolab 7 Series:

- **Autolab PGSTAT302:** modular PGSTAT with 30 V compliance and 2 A maximum current. This instrument is now replaced by the **PGSTAT302N**.
- **Autolab PGSTAT30:** modular PGSTAT with 30 V compliance and 2 A maximum current. This instrument is now replaced by the **PGSTAT302N**.
- **Autolab PGSTAT12:** modular PGSTAT with 12 V compliance and 250 mA maximum current. This instrument is now replaced by the **PGSTAT128N**.
- **Autolab PGSTAT100:** modular PGSTAT with 100 V compliance and 250 mA maximum current. This instrument is now replaced by the **PGSTAT100N**.

16.2.6.1 Scope of delivery

The 7 Series Autolab systems are supplied with the following items:

- Autolab potentiostat/galvanostat
- ADC164 (installed)
- DAC164 (installed)
- Cell cable (WE/CE)
- Ground cable
- Differential amplifier (RE/S)
- Monitor cable
- Power cable
- BNC cable (50 cm)
- USB cable
- Set of four alligator clips
- Autolab dummy cell

16.2.6.2 Instrument power-up state

The power-up state of the instrument is hardware defined. The following settings are automatically selected whenever the instrument is powered on or whenever the connection to the instrument is reset by the software.

- Cell: off
- Mode: potentiostatic
- Control bandwidth: high stability
- iR compensation: off
- Current range: 10 mA
- Optional modules: off

- DIO ports: write mode, low state
- Summation point inputs: off
- Oscillation protection: on

16.2.6.3 7 Series Autolab front panel

The front panel of the 7 Series Autolab provides a number of connections, controls and indicators *(see Figure 1097, page 932)*.

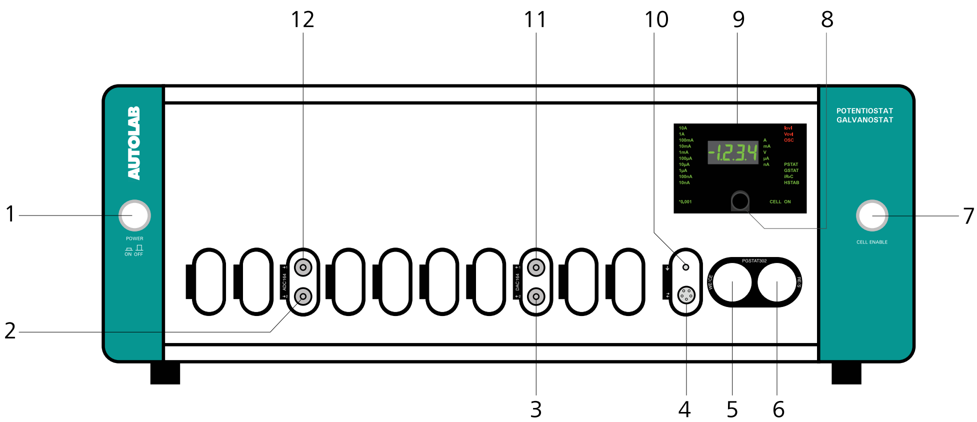


Figure 1097 Overview of the front panel of the 7 Series Autolab

1	**On/Off button** For switching the Autolab on or off.	**2**	**ADC164 →2** Analog input for recording external signals (ADC164 →2).
3	**DAC164 ←2** Analog output for controlling external signals (DAC164 ←2).	**4**	**Monitor cable connector ⇄** For connecting the monitor cable.
5	**CE/WE connector** For connecting the Autolab cell cable, providing connections to the counter electrode (CE) and working electrode (WE).	**6**	**RE/S connector** For connecting the Autolab differential amplifier, providing connections to the reference electrode (RE) and sense electrode (S).
7	**CELL ENABLE button** For enabling and disabling the cell.	**8**	**Display switch mode button** For switching between the voltage and the current on the display.
9	**Display** Display indicating real-time information on the measured current and potential and instrumental settings.	**10**	**Ground connector** Additional ground connector for connecting external devices to the Autolab ground.
11	**DAC164 ←1** Analog output for controlling external signals (DAC164 ←1).	**12**	**ADC164 →2** Analog input for recording external signals (ADC164 →1).

The display (item 9 in *Figure 1097*) is used to provide information about the Autolab to the user. *Figure 1098* shows a detail of this display.

Figure 1098 Overview of the display of the Autolab

1 Booster current range
Indicate that a current range provided by a Booster is active when lit.

2 Autolab current ranges
The current range indicator which is lit corresponds to the active current of the Autolab.

3 ECD current range
Indicates that the ECD module is used when lit. The actual current range is corresponds to the active Autolab current range divided by 1000.

4 Display switch mode button
For switching between the voltage and the current on the display.

5 CELL ON
Indicates that the cell is on when lit.

6 Operation mode indicators
Indicate the operation settings of the Autolab. From top to bottom:

PSTAT indicates that the Autolab is operating in potentiostatic mode when lit.

GSTAT indicates that the Autolab is operating in galvanostatic mode when lit.

iR-C indicates that the ohmic drop compensation is on when lit

HSTAB indicates that the Autolab is operating in high stability mode when lit.

7 OSC indicator
Indicates that oscillations are detected when lit.

8 V ovl indicator
Indicates that a voltage overload is detected when lit.

9 I ovl indicator
Indicates that a current overload is detected when lit.

10 Unit indicator
Indicates the units used for the value shown on the display.

11 Voltage/Current indicator
Displays the measured voltage or current.

16.2.6.4 Autolab 7 Series back plane

The back plane of the Autolab 7 Series provides a number of connections, shown in *Figure 1099*.

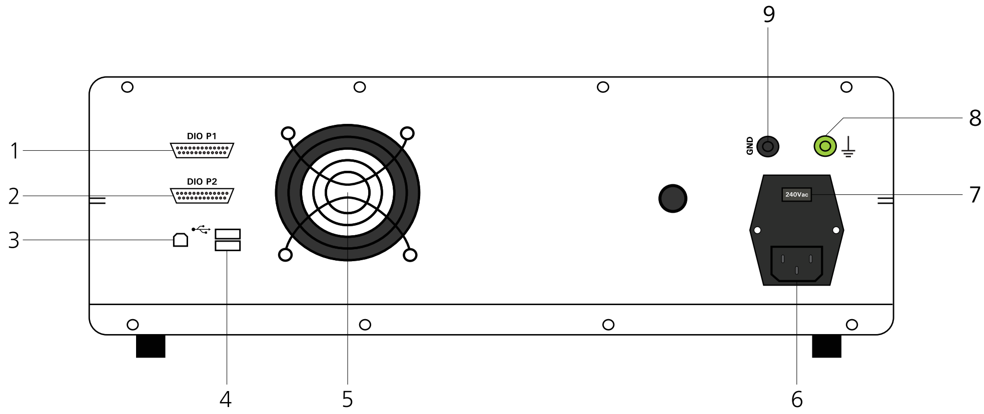


Figure 1099 Overview of the back plane of the Autolab 7 Series

1 DIO P1 connector Digital input/output connector **P1** for sending and receiving external TTL triggers.	**2 DIO P2 connector** Digital input/output connector **P2** for sending and receiving external TTL triggers.
3 USB connector Type B USB plug for connecting the USB cable to the host computer.	**4 USB Hub** For connecting additional USB devices.
5 Fan Required for cooling the Autolab during operation.	**6 Mains connection socket** For connecting the Autolab to the mains supply.
7 Mains voltage indicator Indicates the mains voltage settings of the Autolab.	**8 Earth plug** For connections to the protective earth
9 GND plug For connections to the Autolab ground.	

CAUTION

Make sure that the mains voltage indicator is set properly before switching the Autolab on.

Some of the first Autolab 7 Series instruments are not fitted with an internal USB interface. These instruments are controlled through an external USB interface adapter. The back plane of these instruments is different *(see Figure 1100, page 935)*.

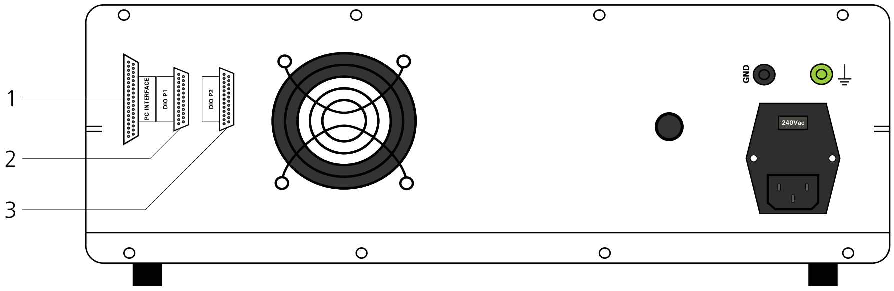


Figure 1100 *Overview of the back plane of the Autolab 7 Series (without USB)*

1 **PC INTERFACE connector** For connecting the external USB interface adapter.	**2** **DIO P1 connector** Digital input/output connector **P1** for sending and receiving external TTL triggers.
3 **DIO P2 connector** Digital input/output connector **P2** for sending and receiving external TTL triggers.	

NOTE

All other items present on the back plane of *Figure 1100* are the same as in *Figure 1099*.

16.2.6.5 Connections for analog signals

The Autolab 7 Series instruments provide connections for analog signals through two different types of connectors:

- BNC connectors directly located on the front panel of the instrument *(see Chapter 16.2.6.5.1, page 935)*.
- BNC connectors located on the monitor cable *(see Chapter 16.2.6.5.2, page 936)*.

CAUTION

Avoid creating ground loops when connecting the Autolab to external signals as this will degrade the performance of the instrument.

16.2.6.5.1 Front panel connections for analog signals

The **ADC164** module and the **DAC164** module, installed in all the 7 Series Autolab instruments, are fitted with two analog inputs and two analog outputs, respectively *(see Figure 1069, page 881)*.

- **ADC164:** the ADC164 inputs, labeled →1 and →2 on the front panel, can be used to record any analog signal with a ± 10 V value range. The input impedance of the two analog inputs is ≥ 1 GΩ. More information on the ADC164 is provided in *Chapter 16.3.1.1*.
- **DAC164** the DAC164 outputs, labeled ←1 and ←2 on the front panel, can be used to generate any analog signal with a ± 10 V value range. The output impedance of these two inputs is 50 Ω. Corrections should be made with loads smaller than 100 kΩ. Because of dissipation, the minimum load impedance should be 200 Ω. More information on the DAC164 is provided in *Chapter 16.3.1.2*.

16.2.6.5.2 Monitor cable connections for analog signals

The **monitor cable**, supplied with the instrument, provides additional connections for analog signals, through BNC connectors. All the connections are with respect to the Autolab ground directly and indirectly with respect to the protective earth *(see Figure 1101, page 936)*.

Figure 1101 The monitor cable provided with the 7 Series Autolab instruments

To use the monitor cable, connect the cable to the matching connector located on the front panel of the Autolab. This connector, labeled ⇄, is located below the front panel display (item 4 in *Figure 1097*).

The following connections are provided through the monitor cable:

- **Eout:** this output corresponds to the differential potential of the reference electrode (RE) with respect to the sense electrode (S). The output voltage will vary between ± 10 V. The output impedance is 50 Ω, so a correction should be made if a load smaller than 100 kΩ is connected to this output. The minimum load is 200 Ω.

- **iout:** this output corresponds to the output of the current-to-voltage converter circuit of the Autolab. The output corresponds to the measured current divided by the current range. The output voltage will vary between ± 10 V. The output impedance is 50 Ω, so a correction should be made if a load smaller than 100 kΩ is connected to this output. The minimum load is 200 Ω.
- **Ein:** this input corresponds to an analog voltage input, directly connected to the summation point of the Autolab. This input is disabled by default. When it is enabled, it can be used to control the Autolab through an external waveform generator. In potentiostatic mode, 1 V provided on this input will add 1 V to the applied potential. In galvanostat mode, 1 V provided on this input will add an extra current equal to 1 multiplied by the current range. In both cases, the converted signal is added to the value already applied by the potentiostat or the galvanostat circuit. The input range is ± 10 V and the input impedance is 1 kΩ when the connection is enabled, so a correction should be made when the source impedance is larger than 1 Ω.

The **Ein** input is enabled and disabled using the **Autolab control** command *(see Figure 1102, page 937)*.

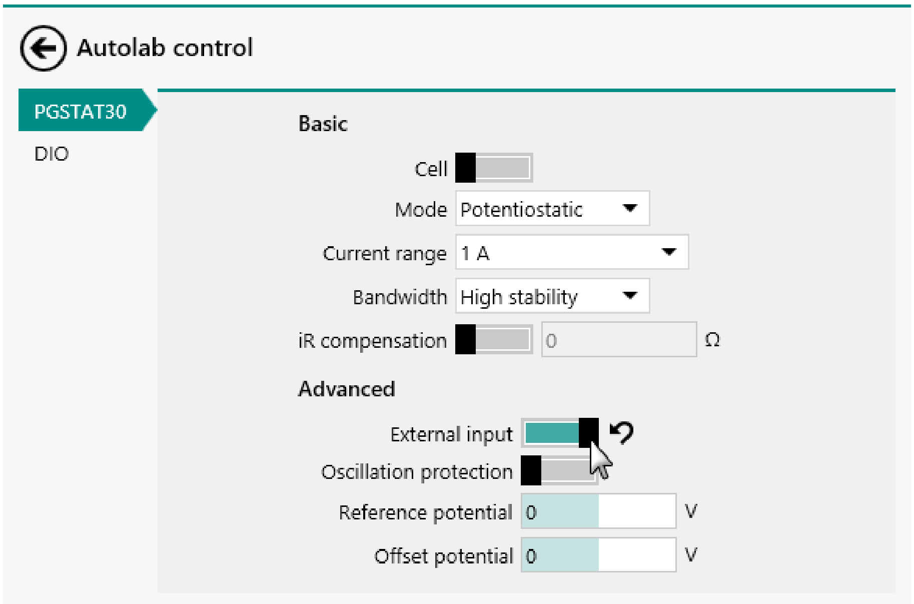


Figure 1102 The Ein connection is enable or disabled in the Autolab control command

CAUTION

Do not leave the **Ein** connection enabled unnecessarily to prevent noise pickup by the Autolab.

16.2.6.6 7 Series Autolab restrictions

Restrictions apply when using the 7 Series Autolab potentiostat/galvanostat:

- **Intended use:** the Autolab potentiostat/galvanostat is intended to be used for electrochemical research only.
- **Service:** there are **no** serviceable parts inside. Servicing of the instrument can only be carried out by qualified personnel.

CAUTION

All attempts to service the instrument will lead to the immediate voiding of any warranty.

WARNING

The PGSTAT100 is fitted with a control amplifier capable of generating up to 100 V potential difference between the counter electrode (CE) and the working electrode (WE). Take all necessary precautions when working with this instrument and use the supplied warning laminated sheet to warn others.

16.2.6.7 7 Series Autolab testing

NOVA is shipped with a procedure which can be used, alongside the **Diagnostics** application, to verify that the instrument is working as expected.

NOTE

For more information on the **Diagnostics** application, please refer to *Chapter 17*.

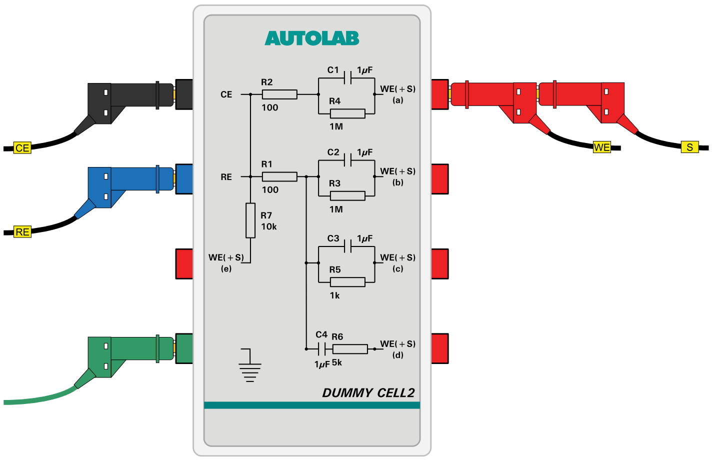
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestCV** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test \TestCV.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (a).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1103*.

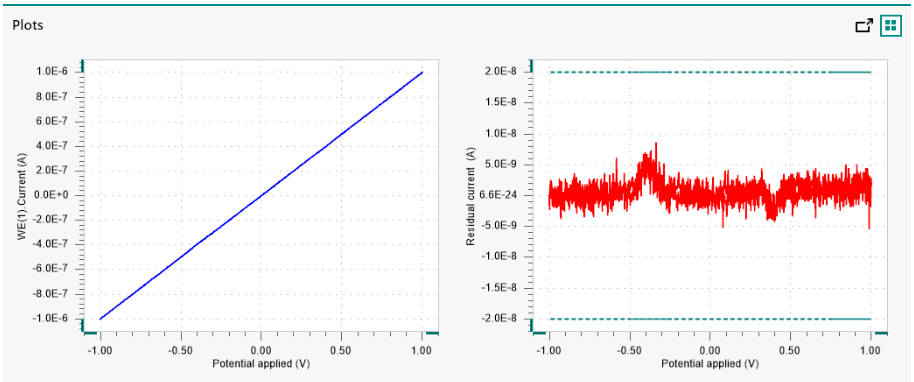
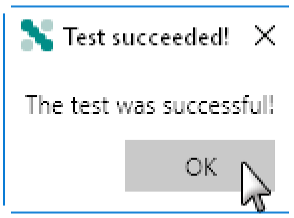


Figure 1103 The data measured by the TestCV procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestCV automatic evaluation of the data requires the following tests to succeed:

1. The residual current, determined by subtracting the expected current from the measured current, must be smaller or equal than ± 20 nA.
2. The inverted slope of the measured current versus the applied potential must be equal to 1000100 Ω± 5 %.
3. The intercept of the measured current versus the applied potential must be equal to ± 4 mV divided by 1000100 Ω± 5 %.

All three conditions must be valid for the test to succeed.

16.2.6.8 7 Series Autolab specifications

The specifications of the 7 Series Autolab are provided in *Table 31*.

Table 31 Specifications of the 7 Series Autolab instruments

Instrument	PGSTAT12	PGSTAT302/30	PGSTAT100
Maximum current	± 250 mA	± 2 A/± 1 A	± 250 mA
Compliance voltage	± 12 V	± 30 V	± 100 V
Potential range	± 10 V		
Applied potential accuracy	± 0.2 % ± 2 mV		
Applied potential resolution	150 µV		
Measured potential resolution	300 nV (gain 1000)		
Current ranges	10 nA to 100 mA, 8 decades	10 nA to 1 A, 9 decades	10 nA to 100 mA, 8 decades
Current accuracy	± 0.2 % of current range		
Applied current resolution	0.015 % of current range		
Measured current resolution	0.00003 % of current range (gain 1000)		
Potentiostat bandwidth	500 kHz	1 MHz	400 kHz
Potentiostat rise/fall time	< 500 ns	< 250 ns	< 500 ns

Instrument	PGSTAT12	PGSTAT302/30	PGSTAT100
Input impedance of electrometer	> 100 GΩ, 8 pF	> 1 TΩ, 8 pF	> 100 GΩ, 8 pF
Input bias current	< 1 pA		
Electrometer bandwidth	> 4 MHz		
iR compensation	2 Ω - 200 MΩ		200 mΩ - 200 MΩ
iR compensation resolution	0.025 %		
Analog output	Potential and current		
Analog voltage input	Yes		
External inputs	2		
External outputs	2		
Digital input/output	48		
Interface	USB		
Warm-up time	30 minutes		
Pollution degree	2		
Installation category	II		
External dimensions (without cables and accessories)	52x42x17 cm^3		
Weight	22 kg	25 kg	25 kg
Power requirements	247 W	247 W	247 W

Instrument	PGSTAT12	PGSTAT302/30	PGSTAT100
Power supply	100 - 240 V ± 10% in four ranges 100 V: [90 - 121 V] 120 V: [104 - 139 V] 230 V: [198 - 242 V] 240 V: [207 - 264 V]		
Power line frequency	47-63 Hz		
Fuse	100 V, 120 V: 3.15 A (slow-slow) 230 V, 240 V: 1.6 A (slow-slow)		100 V, 120 V: 3.15 A (slow-slow) 230 V, 240 V: 1.25 A (slow-slow)
Operating environment	0 °C to 40 °C, 80 % relative humidity without derating		
Storage environment	-10 °C to 60 °C		

16.2.7 µAutolab Series instruments

The µAutolab Series is the predecessor version of the Autolab Compact Series potentiostat/galvanostat produced by Metrohm Autolab *(see Chapter 16.2.4, page 906)*. These instruments are identified by a serial number starting with **µ2AUT7** (for the µAutolab type II) or **µ3AUT7** (for the µAutolab type III).

NOTE

The µAutolab Series instruments are no longer available.

The following instruments belong to the µAutolab Series:

- **µAutolab type II:** compact potentiostat/galvanostat with 12 V compliance and 80 mA current. It is now replaced by the **PGSTAT101** or **PGSTAT204**.
- **µAutolab type III:** compact potentiostat/galvanostat with 12 V compliance and 80 mA current. It is now replaced by the **PGSTAT101** or **PGSTAT204**.

- **µAutolab type III/FRA2:** compact potentiostat/galvanostat with 12 V compliance and 80 mA current with **FRA2** impedance analyzer module. It is now replaced by the **PGSTAT204** with the **FRA32M** module *(see Chapter 16.3.2.13, page 1091)*.

16.2.7.1 Scope of delivery

The µAutolab series systems are supplied with the following items:

- Autolab potentiostat/galvanostat
- On-board ADC
- On-board DAC
- On-board analog integrator
- Cell cable (WE/CE/RE/Ground)
- Power cable
- USB cable (µAutolab type III only)
- FRA2 module (µAutolab type III/FRA2 only)
- Set of three alligator clips
- Autolab dummy cell

16.2.7.2 Instrument power-up state

The power-up state of the instrument is hardware defined. The following settings are automatically selected whenever the instrument is powered on or whenever the connection to the instrument is reset by the software.

- Cell: off
- Mode: potentiostatic
- Control bandwidth: high stability
- Current range: 1 µA
- Optional modules: off
- DIO ports: write mode, low state

16.2.7.3 µAutolab Series front panel

The front panel of the µAutolab type II and type III provides a number of connections, controls and indicators *(see Figure 1104, page 944)*.

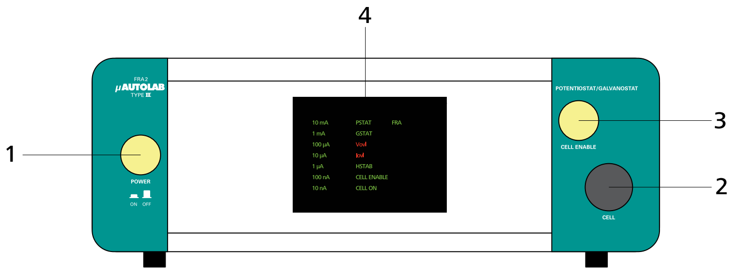


Figure 1104   Overview of the front panel of the µAutolab type II and type III

1 On/Off button
For switching the µAutolab on or off.

2 Cell cable connector
For connecting the cell cable providing connections to the counter electrode (CE), reference electrode (RE), working electrode (WE) and ground.

3 CELL ENABLE button
For enabling and disabling the cell.

4 Display
Display indicating real-time information on the measured current and potential and instrumental settings.

The display (item 4 in *Figure 1104*) is used to provide information about the Autolab to the user. *Figure 1105* shows a detail of this display.

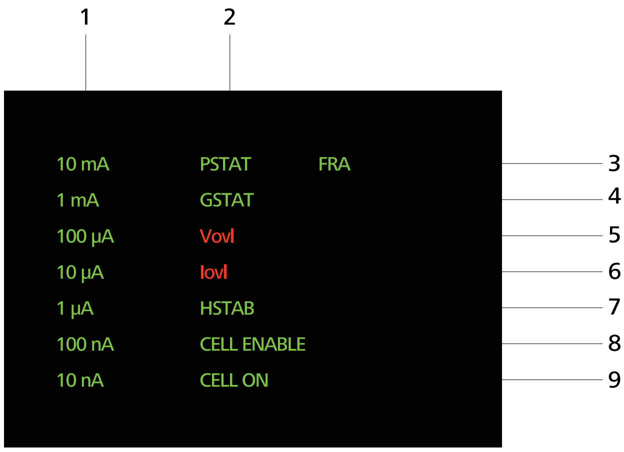


Figure 1105   Overview of the display of the µAutolab type II and type III

1 µAutolab current ranges
The current range indicator which is lit corresponds to the active current of the µAutolab.

2 PSTAT indicator
Indicates that the µAutolab is operating in potentiostatic mode when lit.

3	**FRA indicator** Indicates that the FRA2 module is in use when lit. This indicator is only available for the µAutolab type III fitted with the **FRA2** module.	**4**	**GSTAT indicator** Indicates that the µAutolab is operating in galvanostatic mode when lit.
5	**V ovl indicator** Indicates that a voltage overload is detected when lit.	**6**	**I ovl indicator** Indicates that a current overload is detected when lit.
7	**HSTAB indicator** Indicates that the µAutolab is operating in high stability mode when lit.	**8**	**CELL ENABLE indicator** Indicate the cell is enabled when lit.
9	**CELL ON indicator** Indicates that the cell is on when lit.		

16.2.7.4 µAutolab Series back plane

The back plane of the µAutolab type III provides a number of connections, shown in *Figure 1106*.

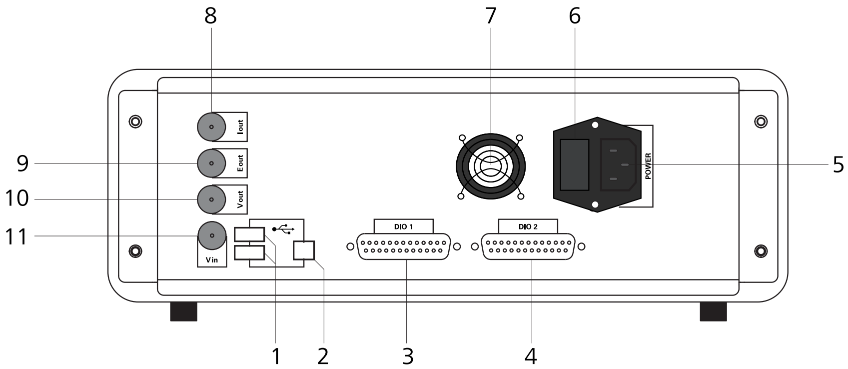


Figure 1106 Overview of the back plane of the µAutolab type III

1	**USB hub** For connecting additional USB devices.	**2**	**USB connector** Type B USB plug for connecting the USB cable to the host computer.
3	**DIO P1 connector** Digital input/output connector **P1** for sending and receiving external TTL triggers.	**4**	**DIO P2 connector** Digital input/output connector **P2** for sending and receiving external TTL triggers.
5	**Mains connection socket** For connecting the µAutolab to the mains supply.	**6**	**Fuse holder** Holds the mains connection socket fuse.
7	**Fan** Required for cooling the µAutolab during operation.	**8**	**I out BNC connector** Connector providing the output of the current to voltage converter of the µAutolab.

9 E out BNC connector
Connector providing the output of the voltage follower of the µAutolab.

10 V out BNC connector
Connector providing the output of the onboard DAC of the µAutolab.

11 V in BNC connector
Connector providing the input of the onboard ADC of the µAutolab.

The back plane of the µAutolab type II provides a number of connections, shown in *(see Figure 1107, page 946)*

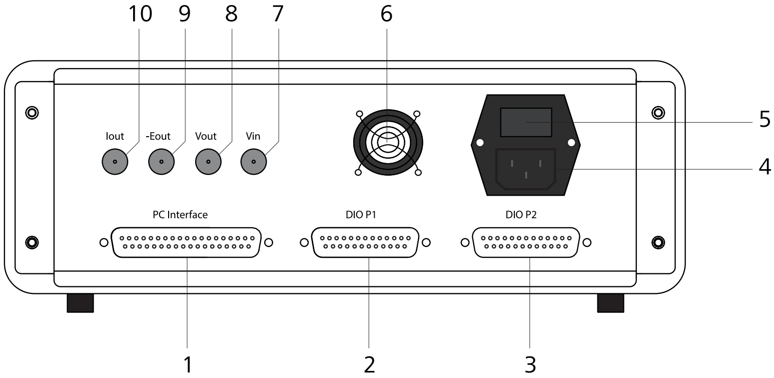


Figure 1107 Overview of the back plane of the µAutolab type II

1 PC INTERFACE connector
For connecting the external USB interface adapter.

2 DIO P1 connector
Digital input/output connector **P1** for sending and receiving external TTL triggers.

3 DIO P2 connector
Digital input/output connector **P2** for sending and receiving external TTL triggers.

4 Mains connection socket
For connecting the µAutolab to the mains supply.

5 Fuse holder
Holds the mains connection socket fuse.

6 Fan
Required for cooling the µAutolab during operation.

7 V in BNC connector
Connector providing the input of the onboard ADC of the µAutolab.

8 V out BNC connector
Connector providing the output of the onboard DAC of the µAutolab.

9 -E out BNC connector
Connector providing the inverted output of the voltage follower of the µAutolab.

10 I out BNC connector
Connector providing the output of the current to voltage converter of the µAutolab.

16.2.7.5 Connections for analog signals

Four connectors, located on the back plane of the µAutolab type II and µAutolab type III, can be used as connections for analog signals *(see Chapter 16.2.7.4, page 945)*. All the connections are provided through BNC connectors. All the connections are with respect to the µAutolab ground directly and indirectly with respect to the protective earth.

The following connections are provided on the back plane of the µAutolab type II and µAutolab type III:

- **Eout:** this output corresponds to the differential potential of the working electrode (WE) with respect to the reference electrode (RE). The output voltage will vary between ± 5 V. The output impedance is 50 Ω, so a correction should be made if a load smaller than 100 kΩ is connected to this output. The minimum load is 200 Ω.
- **iout:** this output corresponds to the inverted output of the current-to-voltage converter circuit of the Autolab. The output corresponds to the inverted measured current divided by the current range. The output voltage will vary between ± 10 V. The output impedance is 50 Ω, so a correction should be made if a load smaller than 100 kΩ is connected to this output. The minimum load is 200 Ω.
- **Vout:** this output corresponds to the output of the on-board DAC of the µAutolab type II or µAutolab type III. It can be used to generate any analog signal with a ± 10 V value range. The output impedance is 50 Ω. Corrections should be made with loads smaller than 100 kΩ. The minimum load is 200 Ω. More information on the on-board DAC is provided in *Chapter 16.3.1.2*.
- **Vin:** this input corresponds to the input of the on-board ADC of the µAutolab type II or µAutolab type III. It can be used to record any analog signal with a ± 10 V value range. The input impedance is ≥ 1 GΩ. More information on the on-board ADC is provided in *Chapter 16.3.1.1*.

16.2.7.6 µAutolab Series restrictions

Restrictions apply when using the µAutolab Series potentiostat/galvanostat:

- **Intended use:** the µAutolab potentiostat/galvanostat is intended to be used for electrochemical research only.
- **Service:** there are **no** serviceable parts inside. Servicing of the instrument can only be carried out by qualified personnel.

CAUTION

All attempts to service the instrument will lead to the immediate voiding of any warranty.

16.2.7.7 µAutolab Series testing

NOVA is shipped with a procedure which can be used, alongside the **Diagnostics** application, to verify that the instrument is working as expected.

NOTE

For more information on the **Diagnostics** application, please refer to *Chapter 17*.

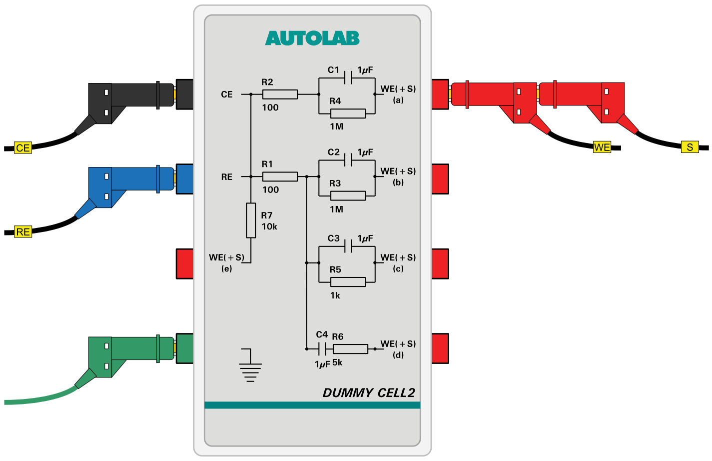
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestCV** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test \TestCV.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (a).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1108*.

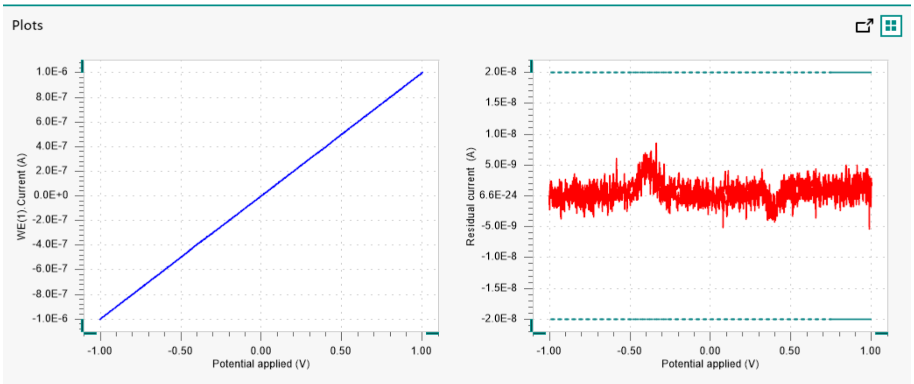
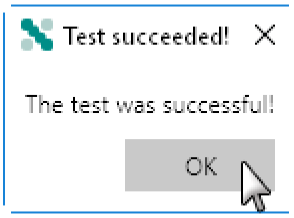


Figure 1108 The data measured by the TestCV procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestCV automatic evaluation of the data requires the following tests to succeed:

1. The residual current, determined by subtracting the expected current from the measured current, must be smaller or equal than ± 20 nA.
2. The inverted slope of the measured current versus the applied potential must be equal to 1000100 Ω± 5 %.
3. The intercept of the measured current versus the applied potential must be equal to ± 4 mV divided by 1000100 Ω± 5 %.

All three conditions must be valid for the test to succeed.

16.2.7.8 μAutolab Series specifications

The specifications of the μAutolab Series are provided in *Table 32*.

Table 32 Specifications of the μAutolab Series instruments

Instrument	μAutolab type II	μAutolab type III
Maximum current	± 80 mA	
Compliance voltage	± 12 V	

Instrument	µAutolab type II	µAutolab type III
Potential range	± 5 V	
Applied potential accuracy	± 0.2 % ± 2 mV	
Applied potential resolution	150 µV	
Measured potential resolution	300 nV (gain 1000)	
Current ranges	10 nA to 10 mA, 7 decades	
Current accuracy	± 0.2 % of current range	
Applied current resolution	0.015 % of current range	
Measured current resolution	0.00003 % of current range (gain 1000)	
Potentiostat bandwidth	500 kHz	
Potentiostat rise/fall time	< 500 ns	
Input impedance of electrometer	> 100 GΩ, 8 pF	
Input bias current	< 1 pA	
Electrometer bandwidth	> 4 MHz	
Analog output	Potential and current	
Analog voltage input	No	
External inputs	1	
External outputs	1	
Digital input/ output	48	

Instrument	µAutolab type II	µAutolab type III
Interface	External USB	Internal USB
Warm-up time	30 minutes	
Pollution degree	2	
Installation category	II	
External dimensions (without cables and accessories)	27x27x9 cm^3	
Weight	3.6 kg	3.6 kg or 4.4 kg with FRA2
Power requirements	75 W	144 W
Power supply	100 - 240 V ± 10% in four ranges (auto select)	
Power line frequency	47-63 Hz	
Fuse	1.6 A (slow-slow)	
Operating environment	0 °C to 40 °C, 80 % relative humidity without derating	
Storage environment	-10 °C to 60 °C	

16.3 Module description

This chapter describes the extension modules available for the Autolab potentiostat/galvanostat instruments. The modules are grouped into two groups:

- **Common modules:** these modules are included standard in all Autolab systems.
- **Optional modules:** these internal or external optional modules can be installed in the Autolab or connected to the Autolab to extend the functionality of the instrument.

NOTE

Some of the modules described in this chapter are no longer available but are still supported in the software. Whenever applicable, the successor module is specified.

16.3.1 Common modules

Common modules are always present in all Autolab instruments. These modules are either present as exchangeable modules or built-in modules. Depending on the type of Autolab system, these modules can be slightly different. However, the functionality these modules provide is common to all instruments.

The following common modules are available:

- **ADC164 or on-board ADC:** the *analog-to-digital* converter module of the Autolab. It is used to convert measured values into digital words that can be recorded by the host computer *(see Chapter 16.3.1.1, page 952)*.
- **DAC164 or on-board DAC:** the *digital-to-analog* converter module of the Autolab. It is used to convert digital words generated by the host computer into analog values that can be used to control the Autolab or external devices connected to the Autolab *(see Chapter 16.3.1.2, page 959)*.
- **DIO48 or DIO12:** the digital input/output module of the Autolab. This module can be used to send or receive TTL (*Transistor-Transistor Logic*) triggers in order to synchronize the Autolab control with external devices *(see Chapter 16.3.1.3, page 965)*.

16.3.1.1 ADC164 or on-board ADC

The ADC164 or on-board ADC is the *analog-to-digital* converter used by the Autolab instrument to perform all analog control actions during measurements. The ADC164 or on-board ADC used by the Autolab is a multichannel *analog-to-digital* converter. Each channel is fitted with a 16 bit converter, with an input range of ± 10 V.

The resolution of the ADC164 or the on-board ADC is given by:

$$\frac{20\,\text{V}}{2^{16}} = 305.175\,\mu\text{V}$$

The ADC164 or on-board ADC is also fitted with a gain circuit with three settings: gain 1, gain 10 and gain 100. Each of these circuits divides the resolution by a factor equal to the gain. This means that in gain 100, the resolution is 3.05 µV.

The ADC164 provides two inputs for external signals, while the on-board ADC provides an input for a single external signal. One or two signals are provided in the Sampler *(see Figure 1109, page 953)*.

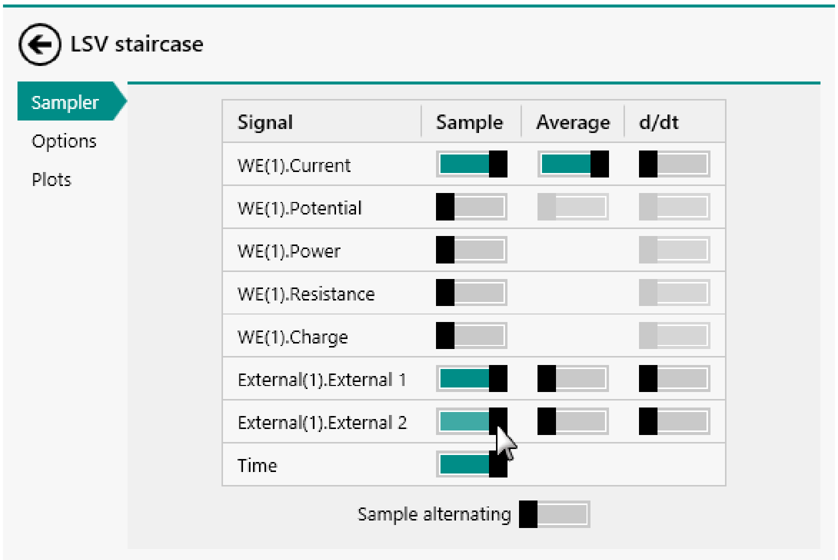


Figure 1109 The ADC164 module provides the External(1),External 1 and External(1).External 2 signals

In the case of the ADC164 the following signals are provided:

- External(1).External 1
- External(1).External 2

In the case of the on-board ADC, the following signal is provided:

- External(1).External 1

NOTE

The names of the signals can be modified in the hardware setup *(see Chapter 16.3.1.1.3, page 954)*.

16.3.1.1.1 ADC164 module front panel connections

The ADC164 module is fitted with two female BNC connectors, labeled →1 and →2 *(see Figure 1110, page 954)*.

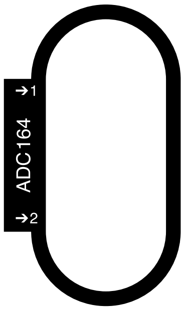


Figure 1110 The front panel label of the ADC164 module

These two connectors provide inputs that can be used to record external signals. They have an input range of ± 10 V and an input impedance of 50 Ω.

16.3.1.1.2 On-board ADC connections

Instruments that are fitted with an on-board ADC provide connections either through a dedicated connector located on the back plane of the instrument or through an optional monitor cable.

- **For the µAutolab type II and µAutolab type III:** the on-board ADC provides a connection, labeled **V in**, on the back plane of the instrument. Please refer to *Chapter 16.2.7.4* for more information.
- **For the PGSTAT101, PGSTAT204, M101 and M204:** the on-board ADC provides a connection, labeled **Vin**, through the optional monitor cable. Please refer to *Chapter 16.2.4.5.1*, *Chapter 16.2.4.5.2* and *Chapter 16.2.5.5* for more information.

16.3.1.1.3 ADC164 and on-board ADC hardware setup

To use the ADC164 or the on-board ADC for the measurement of external signals, the hardware setup needs to be adjusted. The External devices checkbox, provided in the Additional modules panel, adds the ADC164 or on-board ADC to the hardware setup for the purpose of recording external signals *(see Figure 1111, page 955)*.

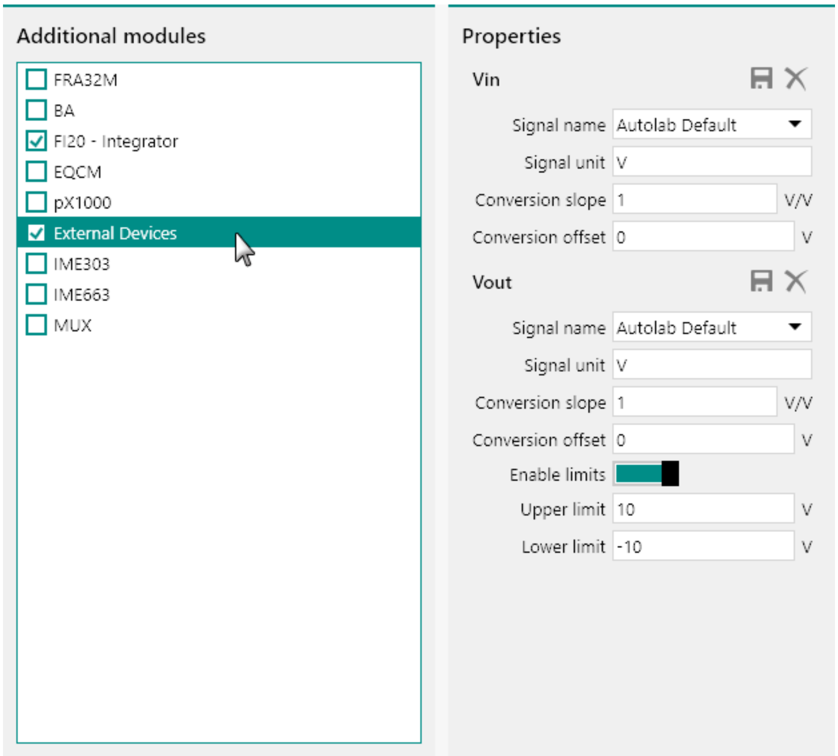


Figure 1111 The External Devices module adds the ADC164 or on-board ADC to the hardware setup

For each available input, the following properties can be defined:

- **Signal name:** the name of the signal to record.
- **Signal unit:** the units of the signal to record.
- **Conversion slope:** the slope of the conversion function used to convert the signal.
- **Conversion offset:** the offset of the conversion function used to convert the signal.

Predefined settings are available using the drop-down list provided for the **Signal name** property *(see Figure 1112, page 956)*.

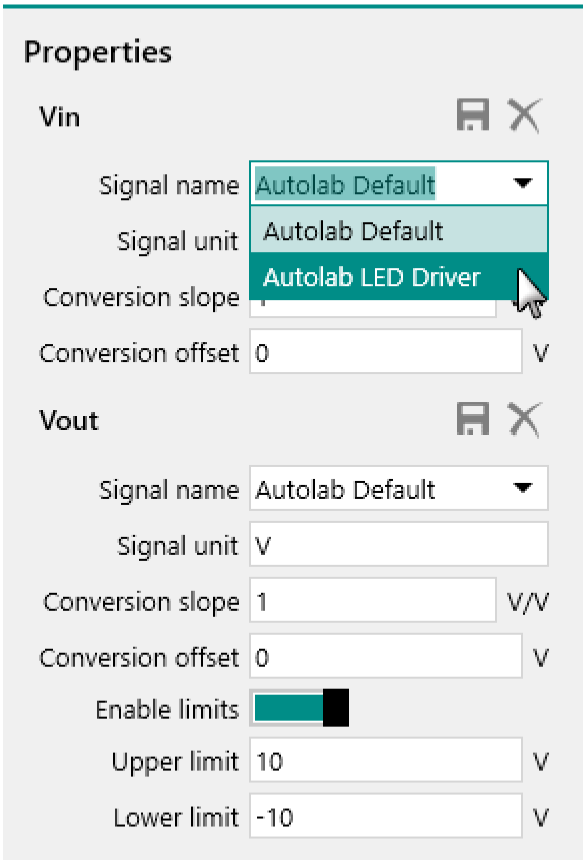


Figure 1112 Predefined settings are available

All the properties are automatically adjusted when one of the predefined setting is selected *(see Figure 1113, page 957)*.

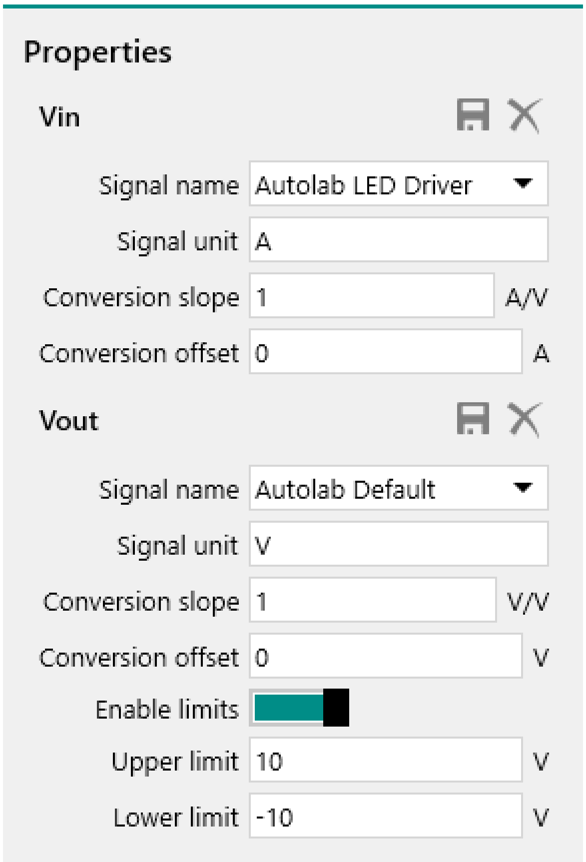


Figure 1113 The LED Driver settings

It is possible to define properties for other devices and to save these as a new preset by clicking the button located above the properties *(see Figure 1120, page 964)*.

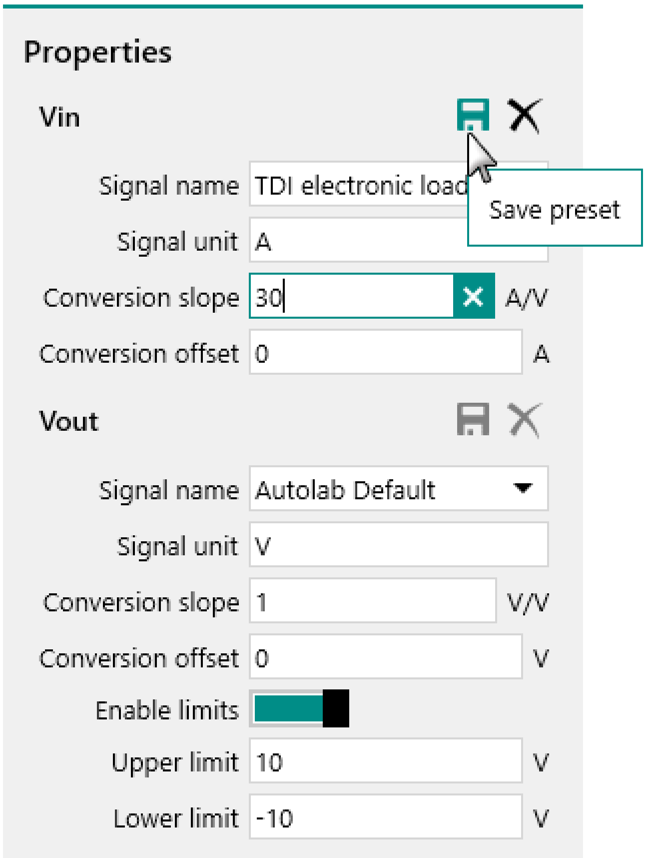


Figure 1114 Saving a new preset in the hardware setup

NOTE

Once a new preset is saved, it can be reused with other instruments connected to the computer.

NOTE

Clicking the button ✕ deletes the preset from the computer. It is not possible to delete predefined presets.

16.3.1.1.4 ADC164 and on-board ADC settings

The ADC164 and on-board ADC have no user-definable settings, except the Sampler settings *(see Chapter 9.1, page 595)*.

The Sampler settings define which signals are sampled during an electrochemical measurement.

The on-board ADC located in the Autolab PGSTAT101, PGSTAT204 and in the M101 and M204 modules of the Multi Autolab systems have an additional property, which can be defined through the Autolab control command *(see Figure 1115, page 959)*.

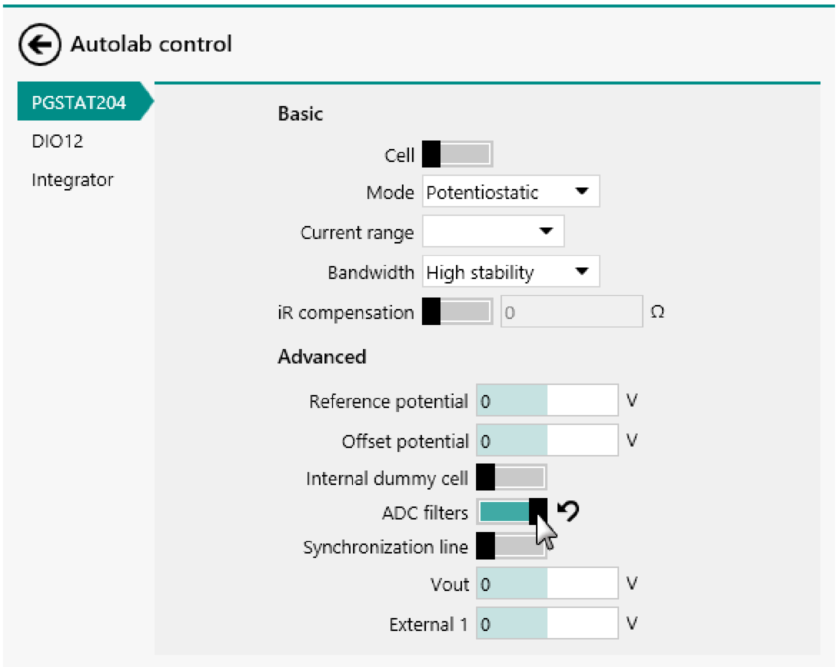


Figure 1115 The ADC filter property is provided in the Autolab control command

The following property can be specified:

- **ADC filters:** a  which can be used to switch the ADC filter on or off. When the filter is on, a low pass filter with a cutoff frequency of 22 kHz is applied on the input signals of the ADC.

16.3.1.1.5 ADC164 and on-board ADC restrictions

The following restrictions apply to the ADC164 and the on-board ADC:

- **Input impedance:** the input impedance of the inputs is 50 Ω. The ADC164 and on-board ADC cannot be used to record unbuffered signals.
- **Input range:** the input range if the inputs is ± 10 V.

16.3.1.2 DAC164 or on-board DAC

The DAC164 or on-board DAC is the *digital-to-analog* converter used by the Autolab instrument to perform all analog control actions during measurements. The DAC164 or on-board DAC used by the Autolab is a multichannel *digital-to-analog* converter. Each channel is fitted with a 16 bit converter, with an output range of ± 10 V.

The resolution of the DAC164 or the on-board DAC is given by:

$$\frac{20\,\text{V}}{2^{16}} = 305.175\,\mu\text{V}$$

16.3.1.2.1 DAC164 module front panel connections

The DAC164 module is fitted with two female BNC connectors, labeled ←1 and ←2 *(see Figure 1116, page 960)*.

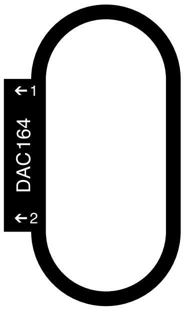


Figure 1116 The front panel label of the DAC164 module

These two connectors provide outputs that can be used to generate signals suitable for controlling external devices. They have an output range of ± 10 V and an output impedance of 50 Ω.

CAUTION

DAC164 ←2 is reserved for AC voltammetry measurements. This means that this output cannot be used to control an external device, unless the instrument is modified.

16.3.1.2.2 On-board DAC connections

Instruments that are fitted with an on-board DAC provide connections either through a dedicated connector located on the back plane of the instrument or through an optional monitor cable.

- **For the µAutolab type II and µAutolab type III:** the on-board DAV provides a connection, labeled **V out**, on the back plane of the instrument. Please refer to *Chapter 16.2.7.4* for more information.
- **For the PGSTAT101, PGSTAT204, M101 and M204:** the on-board DAC provides a connection, labeled **Vout**, through the optional monitor cable. Please refer to *Chapter 16.2.4.5.1*, *Chapter 16.2.4.5.2* and *Chapter 16.2.5.5* for more information.

16.3.1.2.3 DAC164 and on-board DAC hardware setup

To use the DAC164 or the on-board DAC for the generating signals to control external devices, the hardware setup needs to be adjusted. The External devices checkbox, provided in the Additional modules panel, adds the DAC164 or on-board DAC to the hardware setup for the purpose of generating external signals *(see Figure 1117, page 961)*.

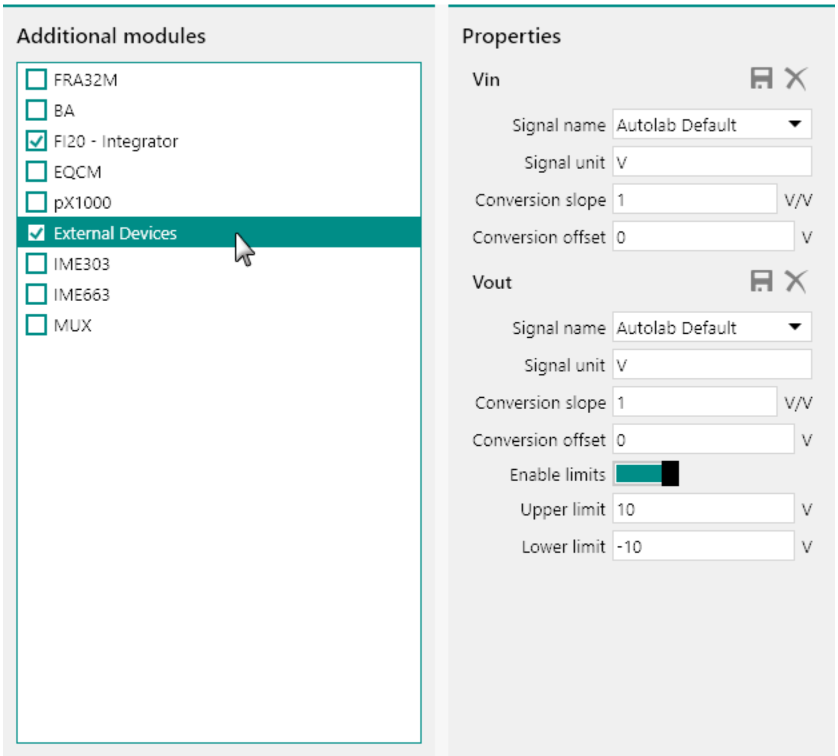


Figure 1117 The External Devices module adds the DAC164 or on-board DAC to the hardware setup

For each available output, the following properties can be defined:

- **Signal name:** the name of the signal to generate.
- **Signal unit:** the units of the signal to generate.
- **Conversion slope:** the slope of the conversion function used to generate the signal.
- **Conversion offset:** the offset of the conversion function used to generate the signal.
- **Enable limits:** a toggle that is provided to enable or disable limits for the generated signal.
- **Upper limit:** the upper limit for the generated signal. This limit is only used if the *Enable limits* property is set to on.
- **Lower limit:** the lower limit for the generated signal. This limit is only used if the *Enable limits* property is set to on.

Predefined settings are available using the drop-down list provided for the **Signal name** property *(see Figure 1118, page 962)*.

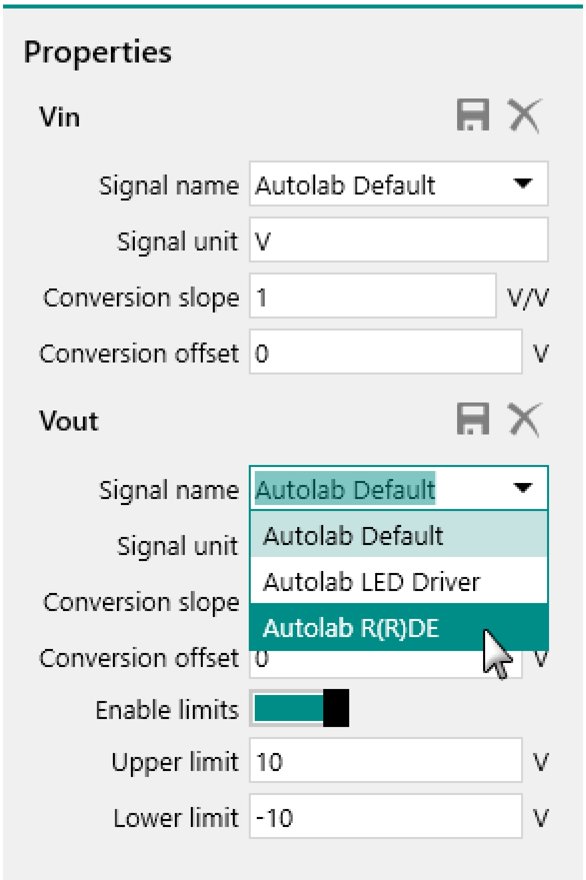


Figure 1118 Predefined settings are available

All the properties are automatically adjusted when one of the predefined setting is selected *(see Figure 1119, page 963)*.

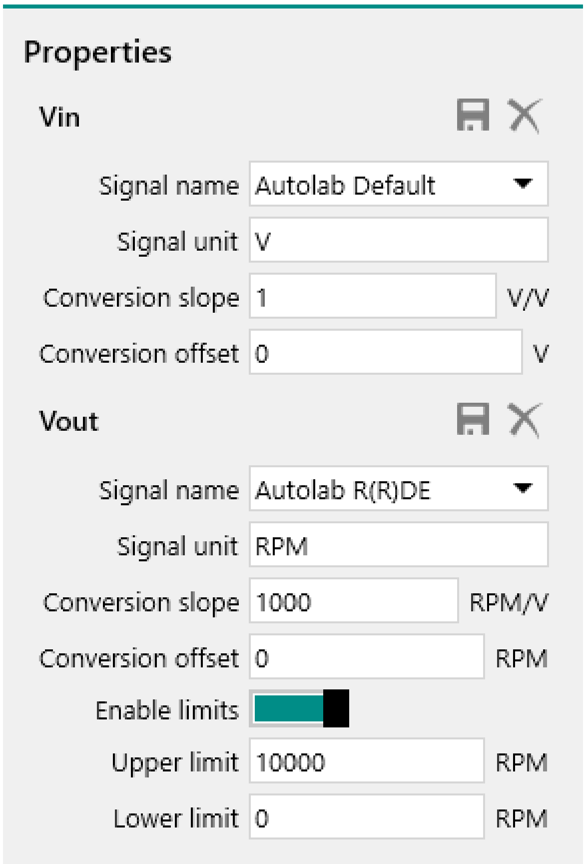


Figure 1119 The R(R)DE settings

It is possible to define properties for other devices and to save these as a new preset by clicking the 💾 button located above the properties *(see Figure 1120, page 964)*.

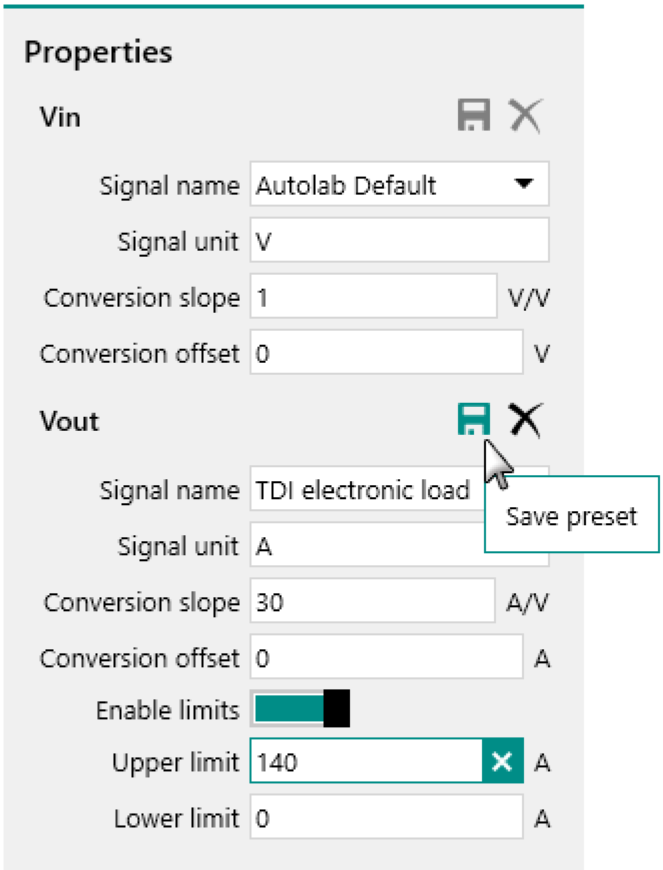


Figure 1120 Saving a new preset in the hardware setup

NOTE

Once a new preset is saved, it can be reused with other instruments connected to the computer.

NOTE

Clicking the button deletes the preset from the computer. It is not possible to delete predefined presets.

16.3.1.2.4 DAC164 and on-board DAC settings

The DAC164 or on-board DAC module settings are completely defined in the NOVA software. The following user-definable settings are available, through the **Autolab control** command *(see Figure 1121, page 965)*:

- **DAC164 ←1/Vout:** this setting defines the output of the DAC164 output 1 (DAC164 ←1) or on-board DAC (Vout), unconverted, as a voltage in the ± 10 V range.

- **Signal name:** this setting defines the output of the DAC164 output 1 (DAC164 ←1) or on-board DAC (Vout), converted to the units of the signal, units and conversion function defined in the hardware setup *(see Chapter 16.3.1.2.3, page 960)*.

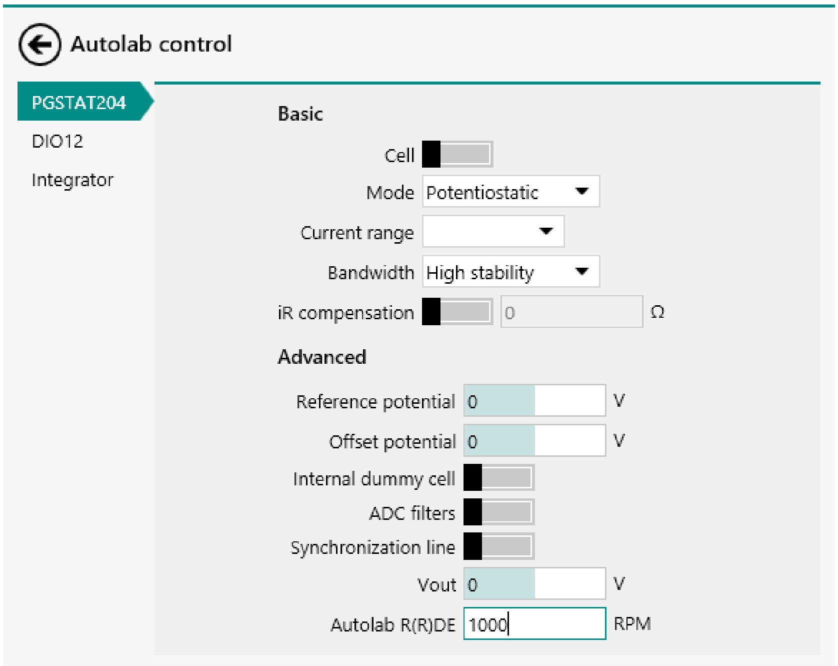


Figure 1121 The settings of the DAC164 or the on-board DAC are defined in the Autolab control command

16.3.1.2.5 DAC164 and on-board DAC restrictions

The following restrictions apply to the use of the **DAC164** and the **on-board DAC**:

- **Output impedance:** the output impedance of the DAC164 and on-board DAC is 50 Ω.

The following restrictions apply to the **DAC164**:

- **Reserved use:** the DAC164 ←2 is reserved for use by the AC voltammetry circuit of the Autolab potentiostat/galvanostat.
- **Shared use:** the DAC164 ←1 is used by optional extension modules (BIPOT, ARRAY. ECD). It is not possible to use this output when these extension modules are in use.

16.3.1.3 TTL Triggers

The Digital Input/Output (DIO) of the Autolab offers the possibility of synchronizing measurements with external devices that can be controlled by TTL signals (*Transistor-Transistor Logic*) or controlling electrode systems, motorburettes or other equipment that can be controlled by transistor-transistor logic.

The Autolab is able to send and receive triggers, using the **Autolab control** command *(see Chapter 7.2.1, page 221)* and the **Wait** command *(see Chapter 7.2.4.2, page 227)*, respectively.

Every Autolab instrument is equipped with one or two digital input/output connectors (DIO) that can be used to receive or send a digital TTL trigger. Depending on the instrument type, two different connector layouts are available:

- **For all other Autolab instruments:** two programmable, 25 pin SUB-D connectors located on the front panel or the back plane of the instrument are available for TTL triggering. Both connectors are identified as a **DIO48**.
- **For the PGSTAT101 or M101 module and the PGSTAT204 and M204 module:** a single, female, 15 pin SUD-D connector located on the front panel or the back plane of the instrument or module is available for TTL triggering. This connector is identified as **DIO12**.

CAUTION

There is a chance of introducing a ground loop when connecting external devices to the Autolab DIO. This can result in higher than expected noise levels during measurements. It is recommended to disconnect external devices from the DIO connector(s) of the Autolab when TTL triggering is not required.

CAUTION

Although the Autolab **PGSTAT302F** is fitted with two DIO ports on the back plane, these ports **cannot** be used for TTL triggering.

16.3.1.3.1 DIO48 type connectors

The DIO48 connectors for TTL triggering is consist of two, 25 pin, female SUB-D connectors. Each connector has a total of 24 user-addressable input/output pins, grouped in three sections:

- **Section A:** pins 1 to 8.
- **Section B:** pins 17 to 14.
- **Section C:** pins 9 to 16.

NOTE

Each section can be programmed to *write* mode or *read* mode by the user.

The pins located in the connector are numbered as shown in *Figure 1122*:

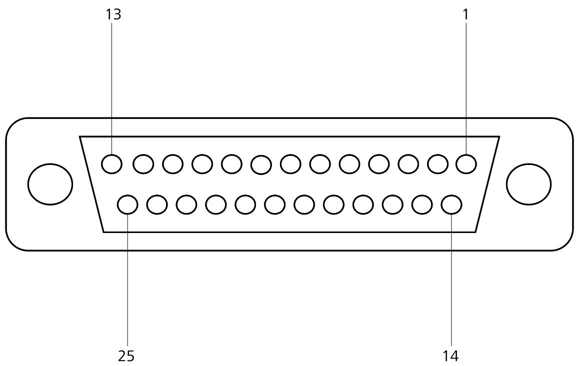


Figure 1122 The DIO48 connector layout

Pin 25 of the connector is used as a digital ground.

CAUTION

The write lines of the **DIO48** connector are capable of supplying a maximum current of **2.5 mA**. Pull-down resistors are usually not required. Please refer to the user manual of the external device connected to the instrument for more information.

16.3.1.3.2 DIO12 type connector

The DIO12 connector for TTL triggering consists of a single, 15 pin, female SUB-D connector. This connector has a total of 12 user-addressable input/output pins, grouped in two sections:

- **Section A:** 8 write pins.
- **Section B:** 4 read pins.

NOTE

The 4 pins of Section B are galvanically isolated.

The pins located in the connector are numbered as shown in *Figure 1123*:

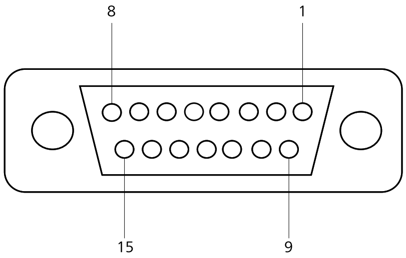


Figure 1123 The DIO12 connector layout

The pin layout is detailed in *Table 33*.

Table 33 Inputs and outputs of the DIO12 connector

Assignment	Pin number	Section
Input 1	1	B
Input 2	9	B
Input 3	2	B
Input 4	10	B
Output 1	12	A
Output 2	5	A
Output 3	13	A
Output 4	6	A
Output 5	14	A
Output 6	7	A
Output 7	15	A
Output 8	8	A
Digital ground	4	
Digital ground	11	
Isolated ground	3	

CAUTION

The write lines of the DIO12 connector are capable of supplying a maximum current of **200 mA**. Suitable pull-down resistors should be placed in the write lines of the DIO12. A typical value for the pull-down resistance is about 1 kΩ. Please refer to the user manual of the external device connected to the instrument for more information.

16.3.1.3.3 Sending triggers

Each pin on the DIO connector(s) of the Autolab can be set to two different levels:

- **Low, 0 V:** this status corresponds to a digital **0** state. This is the default power-up state of the Autolab DIO pins.
- **High, 5 V:** this status corresponds to a digital **1** state.

Depending on the device type, an external device connected to the Autolab can be triggered by a *rising edge* transition or a *falling edge* transition.

A *rising edge* TTL trigger is generated by transitioning from low state to high state (0 → 1), as shown in *Figure 1124*. If required, the involved pin can be reprogrammed to low state.

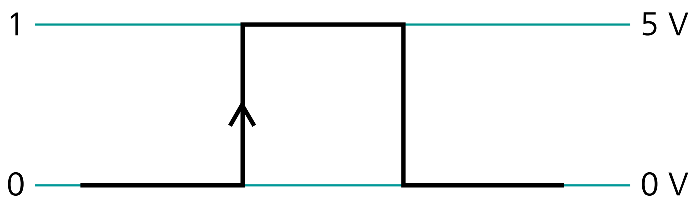


Figure 1124 Rising edge TTL trigger

A *falling edge* TTL trigger is generated by transitioning from high state to low state (1 → 0), as shown in *Figure 1125*. If required, the involved pin can be reprogrammed to high state.

Figure 1125 Falling edge TTL trigger

Each section of a DIO connector located on the Autolab can be set to two different modes, using the **Autolab control** command *(see Chapter 7.2.1, page 221)*:

- **Read (R):** the section is initialized to *read* mode and will be used to receive TTL triggers.
- **Write (W):** the section is initialized to *write* mode and will be used to send TTL triggers. This is the default power-up state of the DIO48 connectors.

NOTE

The **DIO12** connector has pre-defined read and write pins *(see Chapter 16.3.1.3.2, page 967)*.

16.3.1.4 Dummy cell

For testing purposes and for illustration purposes, the Autolab potentiostat/galvanostat systems are supplied with a **dummy cell**. These cells can be used to carry out specially designed tests or diagnostics of the instrument, as explained in *Chapter 17* and can be used to perform measurements on a known circuit.

Depending on the type of instrument, the following dummy cells are available:

- **Dummy cell 2:** all Autolab potentiostat/galvanostat systems are supplied with the standard Autolab dummy cell *(see Chapter 16.3.1.4.1, page 970)*.
- **Internal dummy cell:** all the PGSTAT101, M101, PGSTAT204 and M204 Autolab systems are supplied with an internal dummy cell *(see Chapter 16.3.1.4.2, page 972)*.
- **Option ECI10M dummy cell:** this dummy cell is an optional item that can be used in combination with the ECI10M module *(see Chapter 16.3.1.4.3, page 974)*.
- **Booster10A test cell:** all BOOSTER10A systems are supplied with a dedicated high power test cell *(see Chapter 16.3.1.4.4, page 975)*.
- **Booster20A test cell:** all BOOSTER20A systems are supplied with a dedicated high power test cell *(see Chapter 16.3.1.4.5, page 976)*.

CAUTION

All the **dummy cells** supplied with or available for the Autolab instruments are uncalibrated. These cells **cannot** be used to verify that the instrument is reaching all of the specifications. These cells should only be used to carry out qualitative measurements unless otherwise specified in this manual.

16.3.1.4.1 Autolab Dummy cell 2

The Autolab dummy cell 2 is the standard dummy cell, supplied with all instruments except the PGSTAT101. The Dummy cell is shown in *Figure 1126*, schematically.

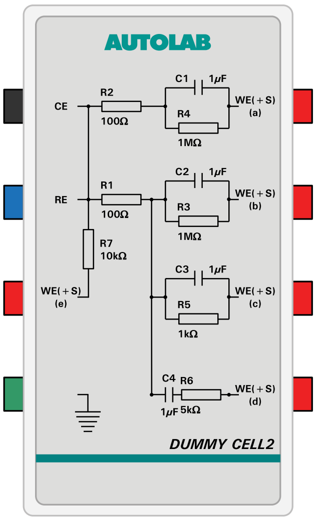


Figure 1126 The Autolab dummy cell 2

This dummy cell is fitted with five circuits, consisting of resistors and capacitors. The actual values and the tolerances of these components are shown in *Figure 1127*.

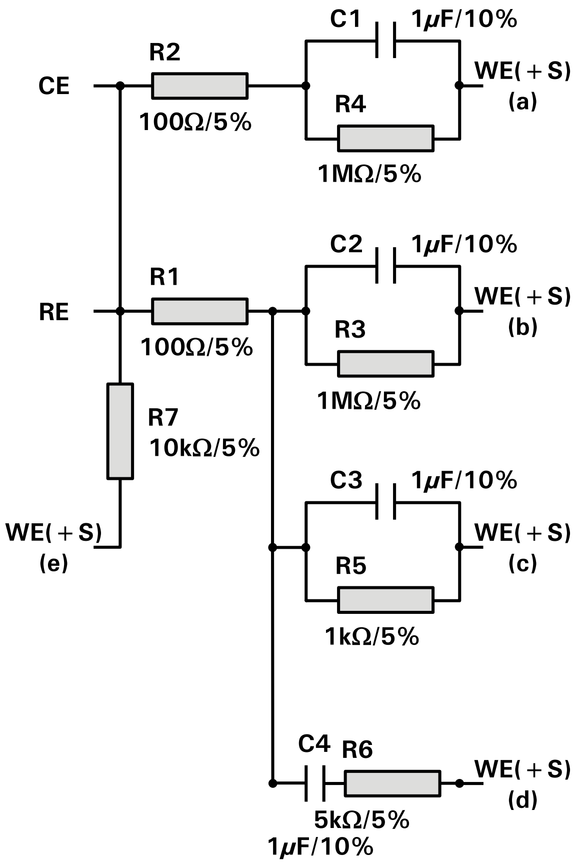


Figure 1127 Component values and tolerances used in the Autolab dummy cell 2

All resistors have a tolerance of 5% and all capacitors have a tolerance of 10%.

CAUTION

The **Dummy cell 2** is not calibrated. This cell **cannot** be used to verify that the instrument is reaching all of the specifications. This cell should only be used to carry out qualitative measurements unless otherwise specified in this manual.

16.3.1.4.2 Internal dummy cell

The Autolab PGSTAT101, M101, PGSTAT204 and M204 are all fitted with an internal dummy cell. This cell is built inside the instrument and cannot be removed from the instrument. To use the internal dummy cell, it is necessary to connect the cell cable to the instrument and to shorten the electrode connectors as shown in *(see Figure 1128, page 972)*.

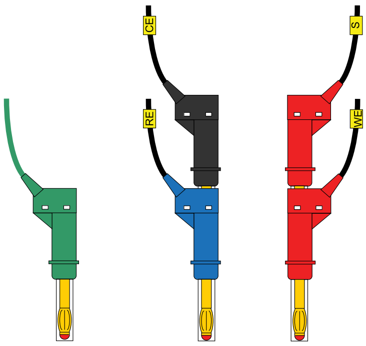


Figure 1128 The electrode connections used in combination with the internal dummy cell

The internal dummy cell can be activated using the dedicated switch available through the **Autolab control** command *(see Figure 1129, page 973)*.

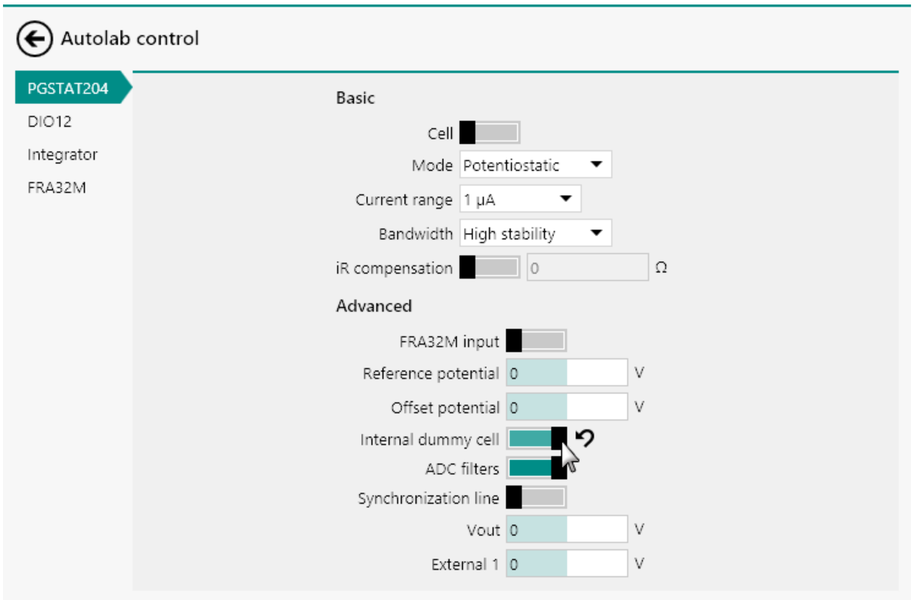


Figure 1129 The internal dummy cell can be activated using the dedicated toggle in the Autolab control command

This dummy cell is fitted with a single circuit, consisting of two resistors and one capacitor. The actual values and the tolerances of these components are shown in *Figure 1130*.

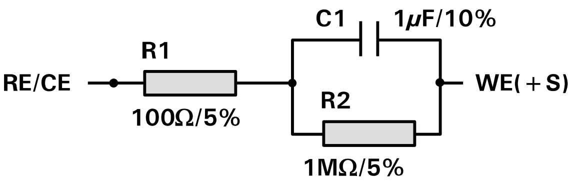


Figure 1130 Component values and tolerances used in the internal dummy cell

Both resistors have a tolerance of 5% and the capacitor have a tolerance of 10%.

CAUTION

The **Internal dummy cell** is not calibrated. This cell **cannot** be used to verify that the instrument is reaching all of the specifications. This cell should only be used to carry out qualitative measurements unless otherwise specified in this manual.

16.3.1.4.3 ECI10M optional dummy cell

The ECI10M test cell is an optional dummy cell, designed for measurements in combination with the ECI10M module. The Dummy cell is shown in *Figure 1131*, schematically.

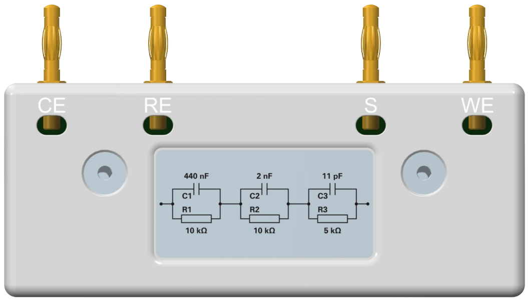


Figure 1131 The ECI10M test cell

NOTE

This dummy cell must be directly connected to the front panel of the ECI10M external interface!

This dummy cell is fitted with a single circuit, consisting of resistors and capacitors. The actual values and the tolerances of these components are shown in *Figure 1132*.

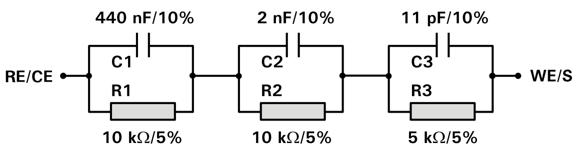


Figure 1132 Component values and tolerances used in the ECI10M test cell

All resistors have a tolerance of 5% and all capacitors have a tolerance of 10%.

CAUTION

The **ECI10M test cell** is not calibrated. This cell **cannot** be used to verify that the instrument is reaching all of the specifications. This cell should only be used to carry out qualitative measurements unless otherwise specified in this manual.

16.3.1.4.4 Booster10A test cell

The BOOSTER10A systems are supplied with a high power test cell. This cell is mounted on a heat sink to dissipate heat while the cell is used *(see Figure 1133, page 975)*.

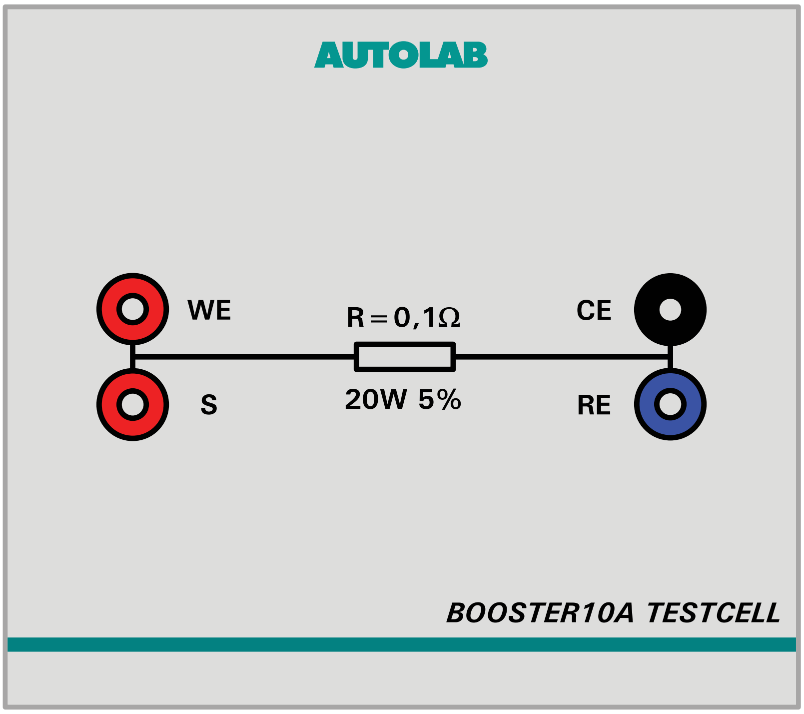


Figure 1133 The BOOSTER10A test cell

NOTE

More information on the BOOSTER10A is available in *Chapter 16.3.2.5*.

The dummy cell supplied with the BOOSTER10A is fitted with a single resistor of 100 mΩ. This resistor has a tolerance of 5% and can dissipate up to 20 W of power.

CAUTION

The **BOOSTER10A test cell** is not calibrated. This cell **cannot** be used to verify that the instrument is reaching all of the specifications. This cell should only be used to carry out qualitative measurements unless otherwise specified in this manual.

16.3.1.4.5 Booster20A test cell

The BOOSTER20A systems are supplied with a high power test cell. This cell is mounted on a heat sink to dissipate heat while the cell is used *(see Figure 1134, page 976)*.

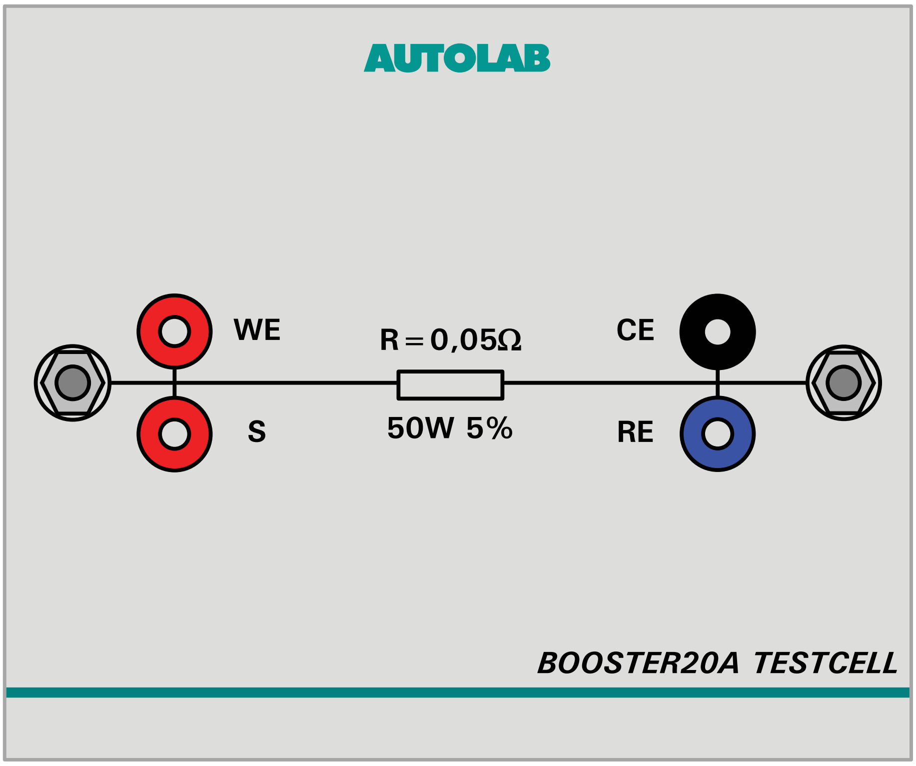


Figure 1134 The Booster20A test cell

NOTE

More information on the BOOSTER20A is available in *Chapter 16.3.2.6*.

The dummy cell supplied with the BOOSTER20A is fitted with a single resistor of 50 mΩ. This resistor has a tolerance of 5% and can dissipate up to 50 W of power.

CAUTION

The **Booster20A test cell** is not calibrated. This cell **cannot** be used to verify that the instrument is reaching all of the specifications. This cell should only be used to carry out qualitative measurements unless otherwise specified in this manual.

16.3.2 Optional modules

Optional modules can be used to extend the functionality of the Autolab system. These modules included in the default configuration and they can be either installed in (internal module) or connected to (external module) the Autolab. Each optional module is designed to provide specific functionality.

CAUTION

Internal modules can only be installed by qualified personnel.

CAUTION

All attempts to service the instrument will lead to the immediate voiding of any warranty.

16.3.2.1 ADC10M module

The ADC10M is a dual channel, synchronous, fast sampling A/D converters. This module can be used to sample the values of up to two signals at the same time at a sampling rate of up to 10,000,000 samples per second.

The ADC10M module can be used for chrono measurements using the smallest possible interval time. This modules can also be used in combination with the linear scan generator modules (SCAN250 or SCANGEN).

16.3.2.1.1 ADC10M module compatibility

The ADC10M module is available for the following instruments:

- PGSTAT302N, PGSTAT302 and PGSTAT30
- PGSTAT128N and PGSTAT12
- PGSTAT100N and PGSTAT100

NOTE

The ADC10M module is **not** compatible with the Autolab instruments not listed above.

16.3.2.1.2 ADC10M scope of delivery

The ADC10M module is supplied with the following items:

- ADC10M module
- ADC10M module label

16.3.2.1.3 ADC10M hardware setup

To use the **ADC10M** module, the hardware setup needs to be adjusted. The checkbox for the module needs to be ticked *(see Figure 1135, page 978)*.

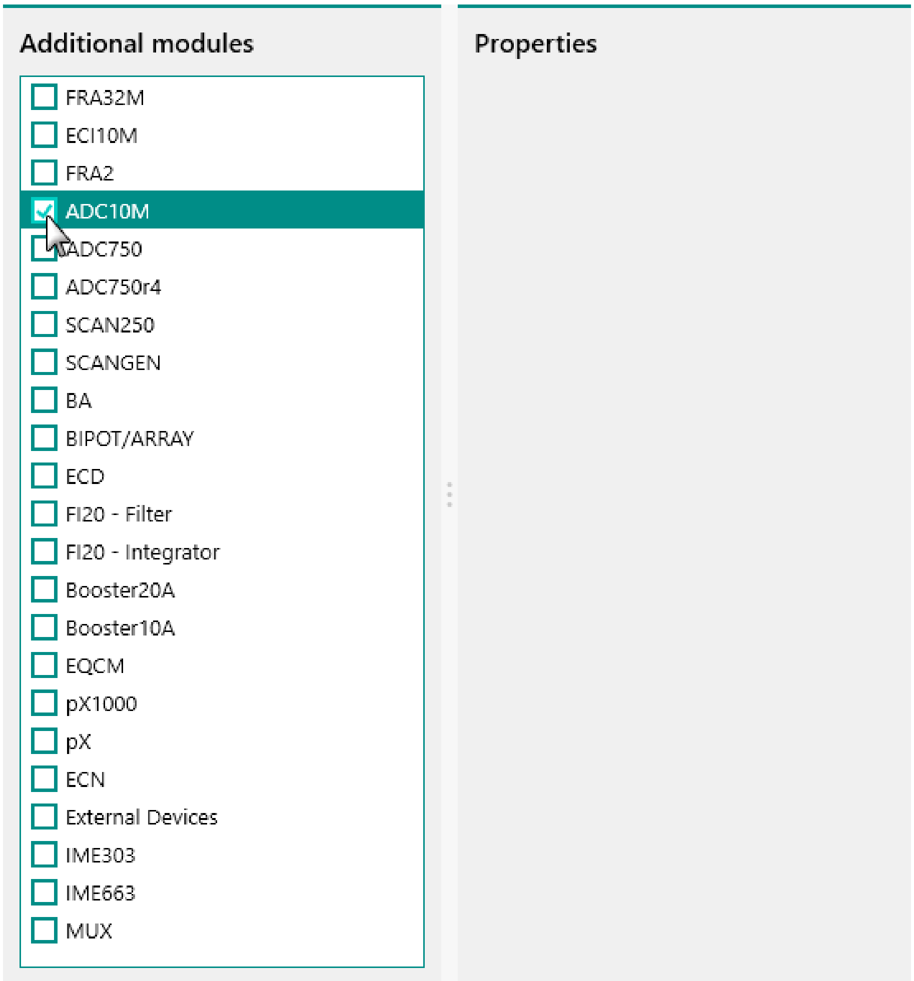


Figure 1135 The ADC10M module is selected in the hardware setup

16.3.2.1.4 ADC10M module settings

The ADC10M module can be used in combination with the Cyclic voltammetry linear scan and the Chrono methods command.

In the case of the Cyclic voltammetry linear scan command, all the module settings are automatically controlled by the measurement command.

In the case of the Chrono methods command, additional module settings can be specified in the additional properties of the command. These settings can only be defined when the Chrono methods command is used in High speed mode *(see Figure 1136, page 979)*.

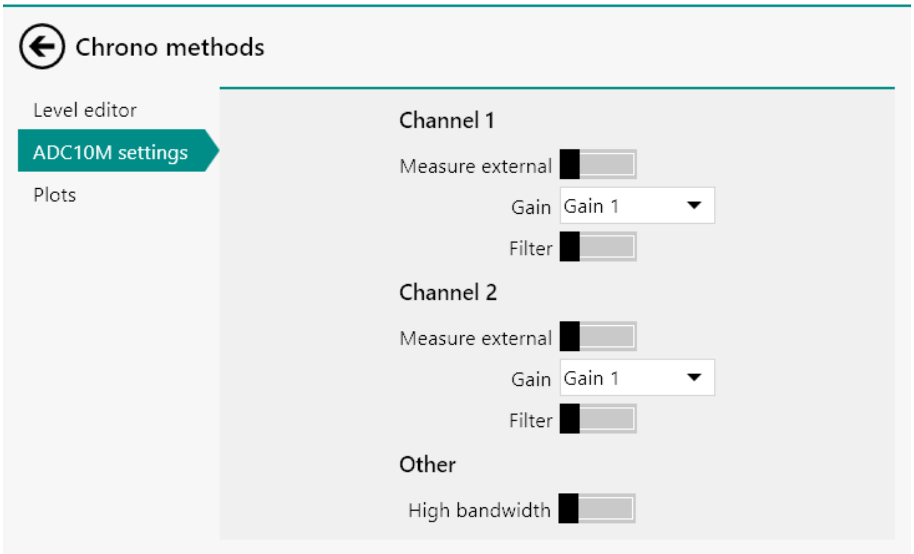


Figure 1136 Additional settings of the ADC10M are available in the Chrono methods command

The following settings are available:

- **Channel 1**
  - **Measure external:** specifies the input signal for Channel 1 of the ADC10M module, using the provided toggle. When this toggle is off, the WE(1).Potential signal is measured through Channel 1 of the ADC10M. When this toggle is on, the signal provided on the →1 BNC input on the front panel of the ADC10M module is sampled *(see Chapter 16.3.2.1.6, page 981)*.
  - **Gain:** specifies the amplification gain for the signal measured by Channel 1 of the ADC10M module, using the drop-down list. The default gain is 1 and optional gains 5, 10 and 20 are available.
  - **Filter:** specifies if a filter should be applied on the signal measured on Channel 1 of the ADC10M, using the provided toggle. When this filter is on, the bandwidth of Channel 1 of the ADC10M is reduced to 200 kHz.

- **Channel 2**
  - **Measure external:** specifies the input signal for Channel 2 of the ADC10M module, using the provided toggle. When this toggle is off, the WE(1).Current signal is measured through Channel 2 of the ADC10M. When this toggle is on, the signal provided on the →2 BNC input on the front panel of the ADC10M module is sampled *(see Chapter 16.3.2.1.6, page 981)*.
  - **Gain:** specifies the amplification gain for the signal measured by Channel 2 of the ADC10M module, using the drop-down list. The default gain is 1 and optional gains 5, 10 and 20 are available.
  - **Filter:** specifies if a filter should be applied on the signal measured on Channel 2 of the ADC10M, using the provided toggle. When this filter is on, the bandwidth of Channel 2 of the ADC10M is reduced to 200 kHz.
- **Other**
  - **High bandwidth:** specifies if the high bandwidth mode of the ADC10M should be used, using the provided toggle. When this setting is off, the bandwidth of both ADC10M channels is set to 600 kHz. When this setting is on, the bandwidth of both ADC10M channels is increased to 1.2 MHz.

NOTE

Select the gain carefully to avoid exceeding the measurable range of the ADC10M.

NOTE

The Filters provided for Channel 1 and Channel 2 can be used to overrule the settings defined by the High bandwidth toggle.

16.3.2.1.5 ADC10M module restrictions

Restrictions apply when using the ADC10M module:

- **No real-time data display:** the ADC10M is fitted with an on-board memory that can be used to store up to 1,024,000 data points. When the ADC10M module is used in an experiment, each new data point is stored in the on-board memory of the module until the experiment is finished. At the end of the measurement, all the stored data points are transferred to the computer for data analysis.

16.3.2.1.6 ADC10M module front panel connections

The ADC10M module is fitted with two female BNC connectors, labeled →1 and →2 *(see Figure 1137, page 981)*.

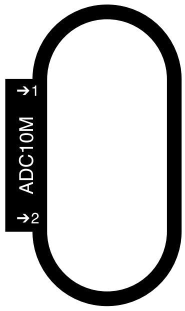


Figure 1137 The front panel label of the ADC10M module

These two connectors provide inputs that can be used to record external signals. They have an input range of ± 10 V and an input impedance of 50 Ω.

NOTE

The input signals used by the ADC10M are defined in the **Chrono methods** and **CV linear scan** commands.

16.3.2.1.7 ADC10M module testing

NOVA is shipped with a procedure which can be used to verify that the **ADC10M** module is working as expected.

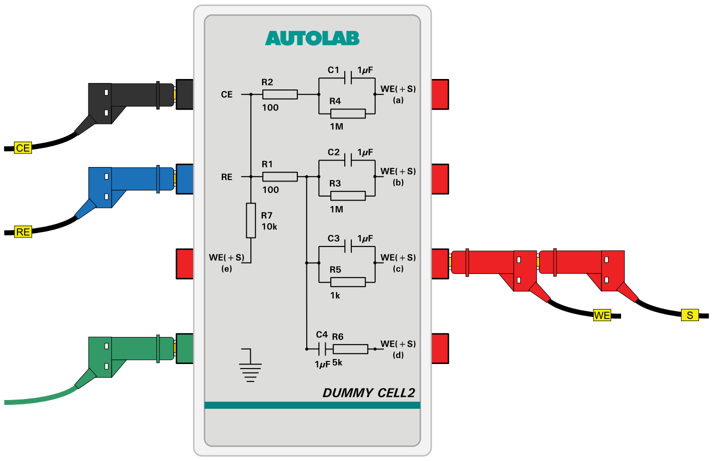
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestADC** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test \TestADC.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (c).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test uses a high-speed chrono methods measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1138*.

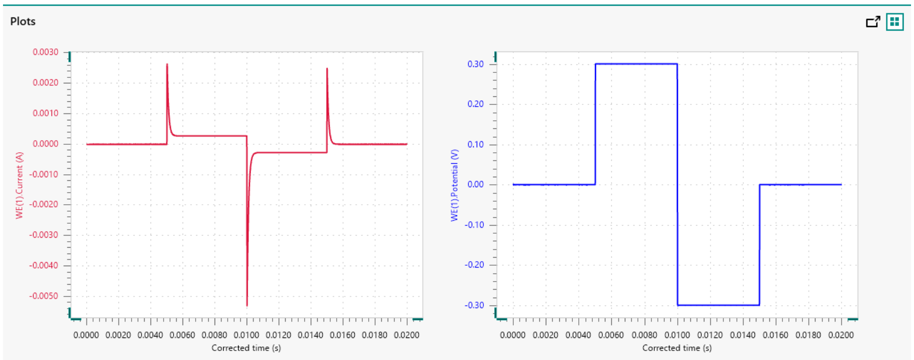
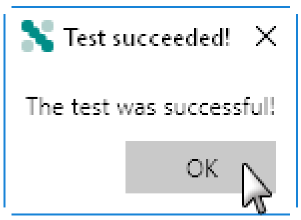


Figure 1138 The results of the TestADC procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestADC automatic evaluation of the data requires the following tests to succeed:

1. The applied potential in Step 1 of the measurement must be 0 V ± 10 mV.
2. The applied potential in Step 2 of the measurement must be 0.3 V ± 10 mV.
3. The applied potential in Step 3 of the measurement must be -0.3 V ± 10 mV.
4. The applied potential in Step 4 of the measurement must be 0 V ± 10 mV.

All four conditions must be valid for the test to succeed.

16.3.2.1.8 ADC10M module specifications

The specifications of the ADC10M module are provided in *Table 34*.

Table 34 Specifications of the ADC10M module

Specification	Value
Number of channels	2
Maximum sampling rate	10,000,000 samples/second
Shortest interval time	100 ns
ADC resolution	14 bit
Maximum resolution, potential	100 µV (gain 10)
Maximum resolution, current	0.0006 % of current range (gain 10)
Maximum number of points	1,024,000
Input range	± 10 V
Input impedance	≥ 100 kΩ

16.3.2.2 ADC750 module

The ADC750 is a dual channel, synchronous, fast sampling A/D converters. This module can be used to sample the values of up to two signals at the same time at a sampling rate of up to 750,000 samples per second.

NOTE

The ADC750 module is no longer available and it is now replaced by its successor module, the ADC10M.

The ADC750 module can be used for chrono measurements using a much smaller interval time than with the ADC164 module. This module can also be used in combination with the linear scan generator modules (SCAN250 or SCANGEN).

Two versions of the ADC750 are available: the ADC750 (revision 5) and the ADC750 (revision 4). The former is identified as ADC750 in the hardware setup while the latter is identified as ADC750r4.

The module revision is not indicated on the front panel of the instrument. It is therefore recommended to declare the ADC750 module in the hardware setup *(see Figure 1139, page 984)*.

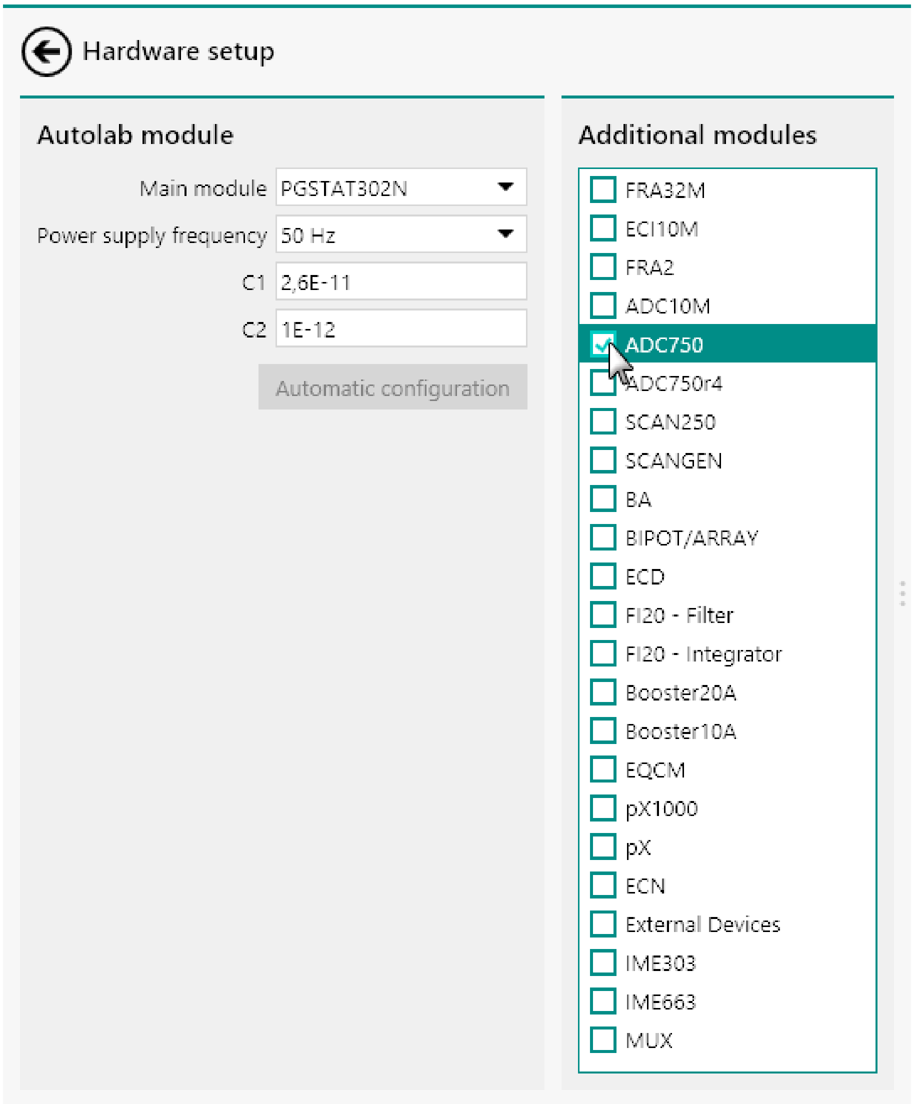


Figure 1139 Declaring the ADC750 in the hardware setup

If an error message is shown after adjusting the hardware setup, then the ADC750 is a revision 4 module. The hardware setup should then be adjusted accordingly *(see Figure 1140, page 985)*.

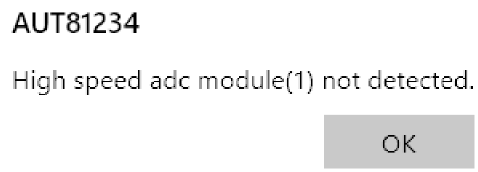


Figure 1140 An error message is shown if a ADC750 is declared instead of a ADC750r4

16.3.2.2.1 ADC750 module compatibility

The ADC750 module is available for the following instruments:

- PGSTAT302N, PGSTAT302 and PGSTAT30
- PGSTAT128N and PGSTAT12
- PGSTAT100N and PGSTAT100
- PGSTAT20
- PGSTAT10

NOTE

The ADC750 module is **not** compatible with the Autolab instruments not listed above.

16.3.2.2.2 ADC750 scope of delivery

The ADC750 module is supplied with the following items:

- ADC750 module
- ADC750 module label

16.3.2.2.3 ADC750 module settings

The ADC750 module can be used in combination with the Cyclic voltammetry linear scan and the Chrono methods command.

In the case of the Cyclic voltammetry linear scan command, all the module settings are automatically controlled by the measurement command.

In the case of the Chrono methods command, additional module settings can be specified in the additional properties of the command. These settings can only be defined when the Chrono methods command is used in High speed mode *(see Figure 1141, page 986)*.

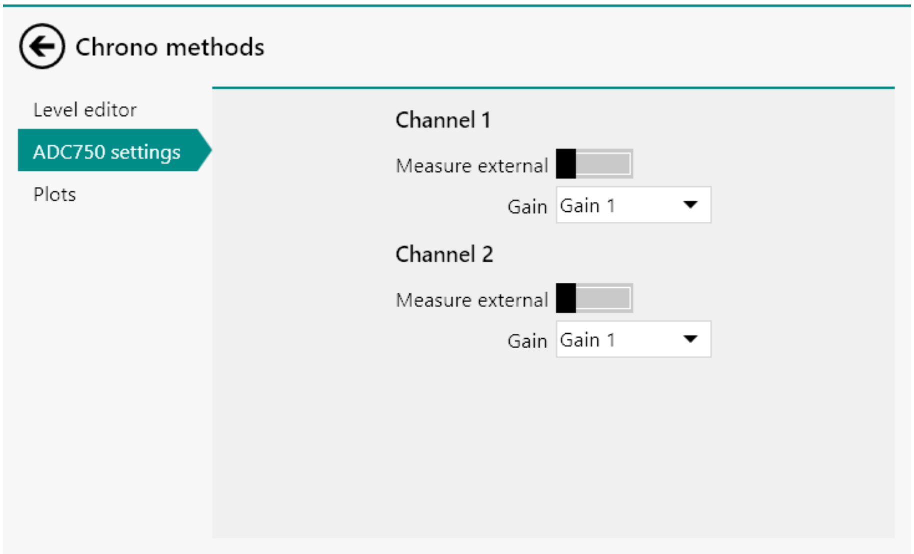


Figure 1141 Additional settings of the ADC750 are available in the Chrono methods command

The following settings are available:

- **Channel 1**
  - **Measure external:** specifies the input signal for Channel 1 of the ADC750 module, using the provided toggle. When this toggle is off, the WE(1).Potential signal is measured through Channel 1 of the ADC750. When this toggle is on, the signal provided on the →1 BNC input on the front panel of the ADC750 module is sampled *(see Chapter 16.3.2.2.5, page 987)*.
  - **Gain:** specifies the amplification gain for the signal measured by Channel 1 of the ADC750 module, using the drop-down list. The default gain is 1 and optional gains 10 and 100 are available.
- **Channel 2**
  - **Measure external:** specifies the input signal for Channel 2 of the ADC750 module, using the provided toggle. When this toggle is off, the WE(1).Current signal is measured through Channel 2 of the ADC750. When this toggle is on, the signal provided on the →2 BNC input on the front panel of the ADC750 module is sampled *(see Chapter 16.3.2.2.5, page 987)*.
  - **Gain:** specifies the amplification gain for the signal measured by Channel 2 of the ADC750 module, using the drop-down list. The default gain is 1 and optional gains 10 and 100 are available.

NOTE

Select the gain carefully to avoid exceeding the measurable range of the ADC750.

16.3.2.2.4 ADC750 module restrictions

Restrictions apply when using the ADC750 module:

- **No real-time data display:** the ADC750 is fitted with an on-board memory that can be used to store up to 512,000 data points. When the ADC750 module is used in an experiment, each new data point is stored in the on-board memory of the module until the experiment is finished. At the end of the measurement, all the stored data points are transferred to the computer for data analysis.

16.3.2.2.5 ADC750 module front panel connections

The ADC750 module is fitted with two female BNC connectors, labeled →1 and →2 *(see Figure 1142, page 987)*.

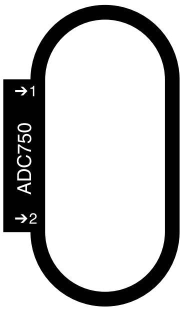


Figure 1142 The front panel label of the ADC750 module

These two connectors provide inputs that can be used to record external signals. They have an input range of ± 10 V and an input impedance of 50 Ω.

NOTE

The input signals used by the ADC750 are defined in the **Chrono methods** and **CV linear scan** commands.

16.3.2.2.6 ADC750 module testing

NOVA is shipped with a procedure which can be used to verify that the **ADC750** module is working as expected.

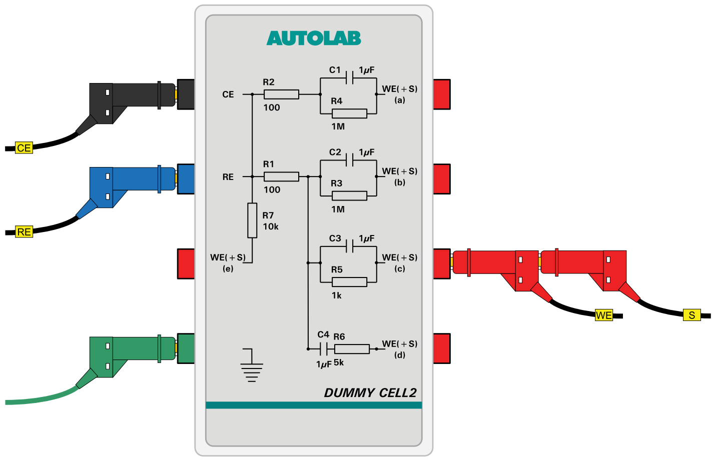
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestADC** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test \TestADC.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (c).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test uses a high-speed chrono methods measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1143*.

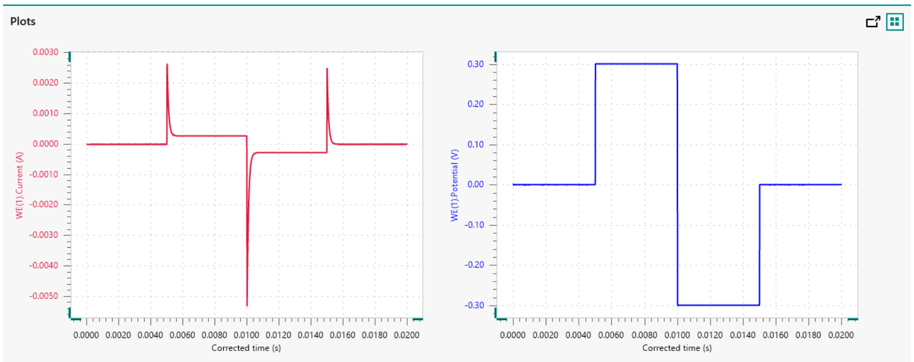
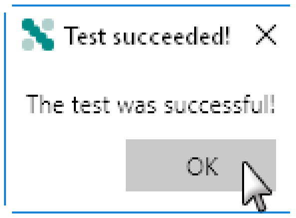


Figure 1143 The results of the TestADC procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestADC automatic evaluation of the data requires the following tests to succeed:

1. The applied potential in Step 1 of the measurement must be 0 V ± 10 mV.
2. The applied potential in Step 2 of the measurement must be 0.3 V ± 10 mV.
3. The applied potential in Step 3 of the measurement must be -0.3 V ± 10 mV.
4. The applied potential in Step 4 of the measurement must be 0 V ± 10 mV.

All four conditions must be valid for the test to succeed.

16.3.2.2.7 ADC750 module specifications

The specifications of the ADC750 module are provided in *Table 35*.

Table 35 Specifications of the ADC750 module

Specification	Value
Number of channels	2
Maximum sampling rate	750,000 samples/second
Shortest interval time	1.33 µs

Specification	Value
ADC resolution	12 bit
Maximum resolution, potential	500 µV (gain 10)
Maximum resolution, current	0.0025 % of current range (gain 10)
Maximum number of points	512,000
Input range	± 10 V
Input impedance	≥ 5 GΩ

16.3.2.3 BA module

The BA module is an extension module for the Autolab PGSTAT and the Multi Autolab. This module provides a second working electrode, **WE(2)**. The BA module has two different operation modes:

- **BIPOT mode:** in this mode, the potential of the second working electrode, WE(2), is defined with respect to the common reference electrode.
- **Scanning BIPOT:** in this mode, the potential of the second working electrode, WE(2), is defined with respect to the main working electrode, WE(1).

NOTE

The BA module only works in potentiostatic mode. The main potentiostat can be set to galvanostatic mode.

The BA module adds the following signal to the Sampler *(see Figure 1144, page 991)*:

- **WE(2).Current (A):** this signal corresponds to the current flowing through the second working electrode, WE(2).
- **WE(2).Charge (C):** this signal corresponds to the charge obtained by numerical integration of the measured WE(2).Current.

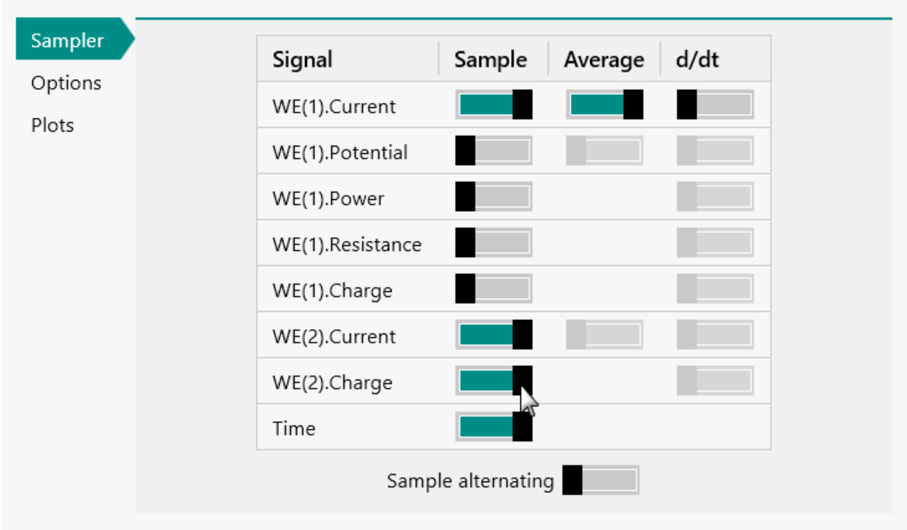


Figure 1144 The BA module provides the WE(2).Charge and WE(2).Current signals

16.3.2.3.1 BA module compatibility

The BA module is available for the following instruments:

- PGSTAT302N, PGSTAT302 and PGSTAT30
- PGSTAT128N and PGSTAT12
- PGSTAT100N and PGSTAT100
- M101
- PGSTAT204/M204

NOTE

The BA module is **not** compatible with the Autolab instruments not listed above.

16.3.2.3.2 BA module scope of delivery

The BA module is supplied with the following items:

- BA module
- BA module label
- WE2 connection cable

16.3.2.3.3 BA hardware setup

To use the **BA** module, the hardware setup needs to be adjusted. The checkbox for the module needs to be ticked *(see Figure 1145, page 992)*.

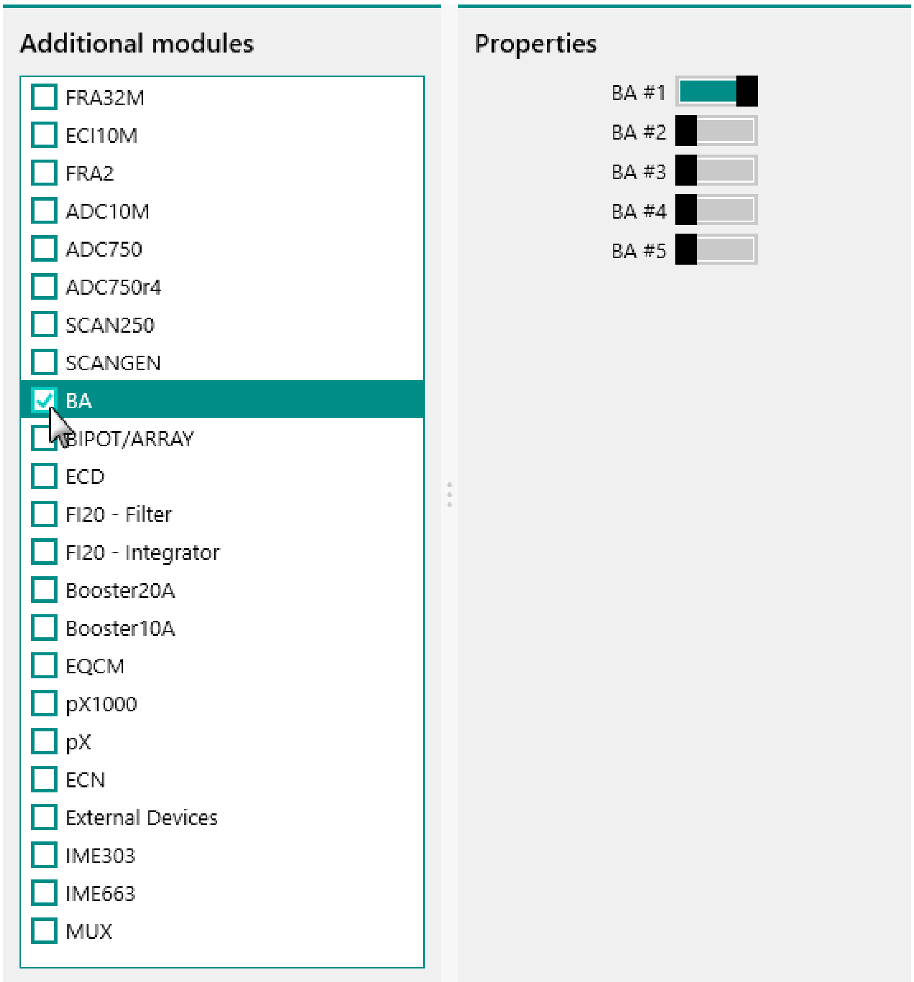


Figure 1145 The BA module is selected in the hardware setup

The toggles provided in the **Properties** panel can be used to specify the number of **BA** modules installed in the instrument.

NOTE

It is only possible to have more than one **BA** module in combination with **MBA** instruments *(see Chapter 16.2.3, page 903)*.

16.3.2.3.4 BA module settings

The BA module settings are completely defined in the NOVA software. The following user-definable settings are available, through the **Autolab control** command *(see Figure 1146, page 993)*:

- **Cell:** a toggle control that can be used to switch the WE(2) on or off.
- **Current range:** a drop-down control that can be used to select the current range of WE(2).

- **Electrode control:** defines the relationship between the cell switch of WE(2) and WE(1). When this setting is set to Linked to WE(1), the cell switch of WE(2) will automatically be set to the same status of the cell switch of WE(1). Using the Cell command, both cell switches will be toggled at the same time. When this setting is set to Independent, the cell switch of WE(2) is decoupled from the cell switch of WE(1). In that case, the cell switch of the WE(2) must be set manually, using the control Cell control provided by the Autolab control command. This setting only affects the transition from cell off to cell on.
- **Mode:** defines the mode for the BA module (BIPOT/Scanning BIPOT).
- **WE(2) potential or Offset potential (V):** defines the potential difference between WE(2) and the reference electrode in BIPOT mode and between WE(2) and WE(1) in Scanning BIPOT mode.

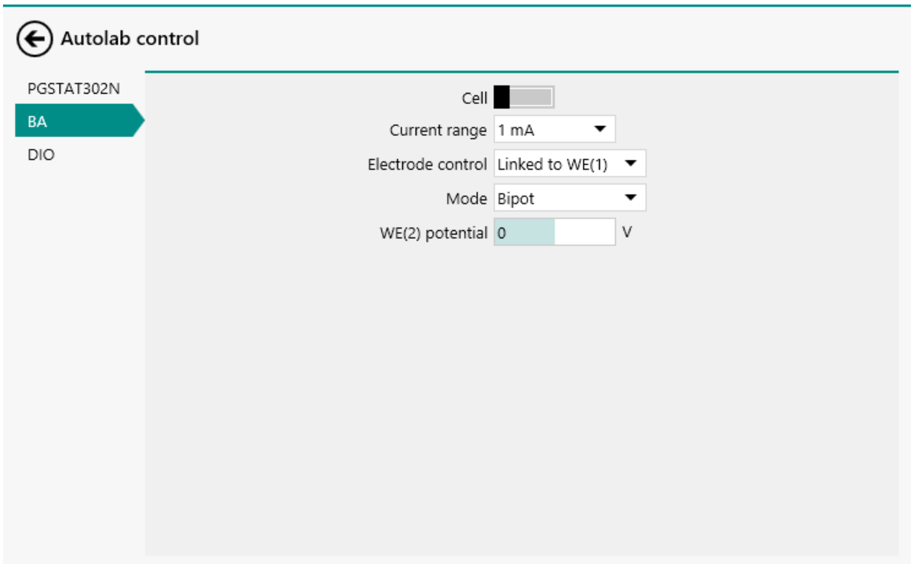


Figure 1146 The BA module settings are defined in the Autolab control command

NOTE

The default startup Electrode control setting is linked to WE(1).

NOTE

The Cell command switches the main potentiostat and the second working electrode off automatically, regardless of the Electrode control setting.

CAUTION

When the electrode control is set to independent, a specific order must be respected to avoid current leakage between WE(1) and WE(2). Always set the cell of WE(2) to ON after the cell has been set to ON for WE(1). Always set the cell of WE(2) to OFF before the cell has been set to OFF for WE(1).

16.3.2.3.5 BA module restrictions

No restrictions apply when using the BA module.

16.3.2.3.6 BA module front panel connections

The BA module is fitted with a single female BNC connector, labeled ←I (in the case of the Autolab potentiostat/galvanostat instrument) or with two female BNC connectors, labeled ←I and WE, respectively *(see Figure 1147, page 994)*.

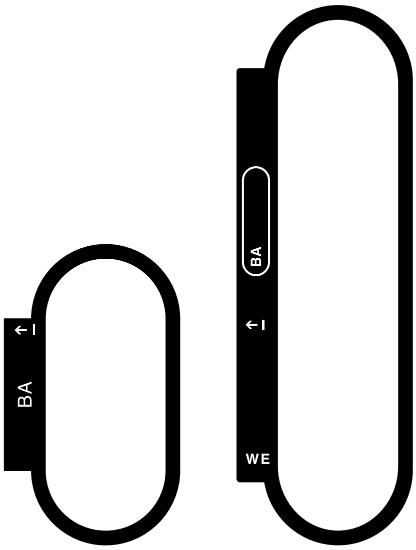


Figure 1147 The front panel labels of the BA module (left: BA module in PGSTAT, right: BA module in Multi Autolab)

NOTE

The WE connector provided in the case of the Multi Autolab instruments is used to connect the WE2 cable.

The signal provided through the ←I connector on the front panel corresponds to the output of the current-to-voltage converter located on the

BA module. The output signal is a voltage, referred to the instrument ground, corresponding to the converted current according to:

$$E_{out}(\leftarrow I) = \frac{i_{(WE2)}}{[CR]}$$

Where $E_{out}(\leftarrow I)$ corresponds to the output voltage signal of the module, in V, $i_{(WE2)}$ corresponds to the current measured by the BA module, in BA and [CR] is the active current range of the BA module.

NOTE

The front panel ←I BNC output is provided for information purposes only.

16.3.2.3.7 BA module testing

NOVA is shipped with a procedure which can be used to verify that the **BA** module is working as expected.

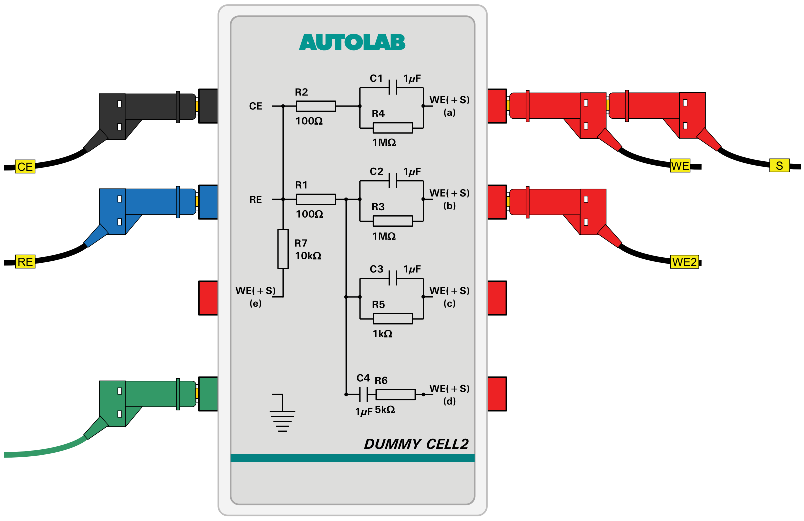
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestBA** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test \TestBA.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (a) and the second working electrode (BA) to the Autolab dummy cell (b).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. Both modes of the BA module are tested. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1148*.

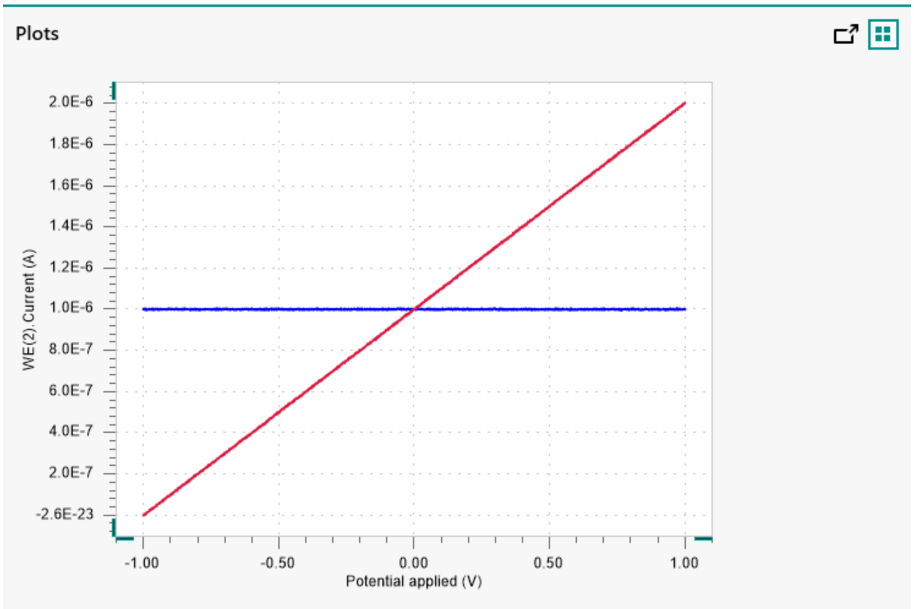
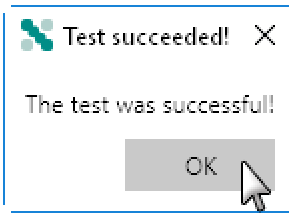


Figure 1148 The results of the TestBA procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestBA automatic evaluation of the data requires the following tests to succeed:

1. The average WE(2).Current measured in Bipotentiostat mode must be equal to 1 V ± 5 mV/1000100 Ω± 5 %.
2. The intercept of the WE(2).Current measured in Bipotentiostat mode must be equal to 1 V ± 5 mV/1000100 Ω± 5 %.
3. The inverted slope of the measured WE(2).Current versus the applied potential must be equal to 1000100 Ω± 5 %.

4. The intercept WE(2).Current measured in Scanning Bipotentiostat mode must be equal to 1 V ± 5 mV/1000100 Ω± 5 %.

All four conditions must be valid for the test to succeed.

16.3.2.3.8 BA module specifications

The specifications of the BA module are provided in *Table 36*.

Table 36 Specifications of the BA module

Specification	Value
Operation mode	BIPOT, Scanning BIPOT (software controlled)
Control DAC	On-board, 16 bit
Maximum current	± 50 mA
Current ranges	10 nA to 10 mA (7 ranges)
Current accuracy	± 0.2 % of current range
Potential range	± 10 V
Potential accuracy	± 2 mV

16.3.2.4 BIPOT/ARRAY module

The BIPOT and ARRAY modules are an extension module for the Autolab potentiostat/galvanostat. These module provides a second working electrode, **WE(2)**. Each of these modules fulfills a specific role:

- **BIPOT module:** this module is designed to control the potential of the second working electrode, WE(2), with respect to the common reference electrode.
- **ARRAY module:** this module is designed to control the potential of the second working electrode, WE(2), with respect to the main working electrode, WE(1).

NOTE

The BIPOT and ARRAY modules are no longer available and it are now replaced by its successor module, the BA.

NOTE

The BIPOT and ARRAY modules only works in potentiostatic mode. The main potentiostat can be set to galvanostatic mode.

The BIPOT and ARRAY modules add the following signal to the Sampler :

- **WE(2).Current (A):** this signal corresponds to the current flowing through the second working electrode, WE(2).
- **WE(2).Charge (C):** this signal corresponds to the charge obtained by numerical integration of the measured WE(2).Current.

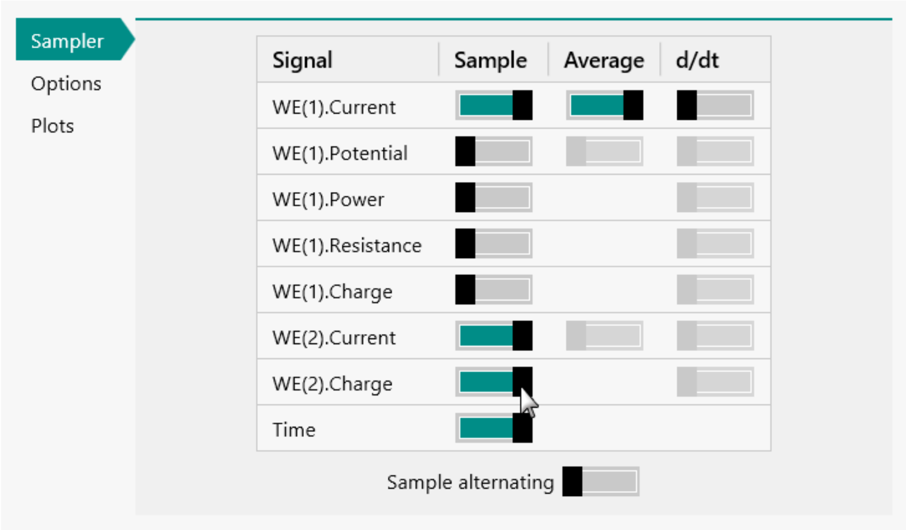


Figure 1149 The BIPOT/ARRAY module provides the WE(2).Charge and WE(2).Current signals

16.3.2.4.1 BIPOT/ARRAY module compatibility

The BIPOT/ARRAY module is available for the following instruments:

- PGSTAT302N, PGSTAT302 and PGSTAT30
- PGSTAT128N and PGSTAT12
- PGSTAT100N and PGSTAT100
- PGSTAT20
- PGSTAT10

NOTE

The BIPOT/ARRAY module is **not** compatible with the Autolab instruments not listed above.

16.3.2.4.2 BIPOT/ARRAY module scope of delivery

The BIPOT/ARRAY module is supplied with the following items:

- BIPOT or ARRAY module
- BIPOT or ARRAY module label
- WE2 connection cable

16.3.2.4.3 BIPOT/ARRAY hardware setup

To use the **BIPOT/ARRAY** module, the hardware setup needs to be adjusted. The checkbox for the module needs to be ticked *(see Figure 1150, page 999)*.

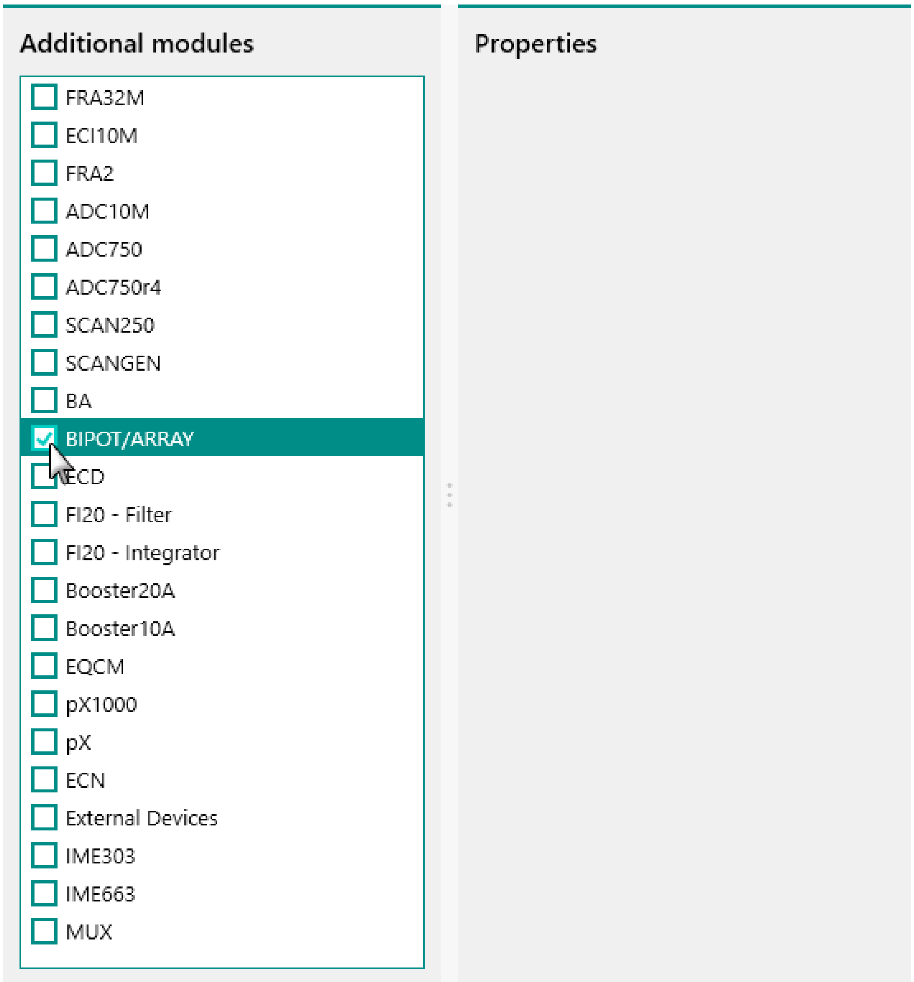


Figure 1150 The BIPOT/ARRAY module is selected in the hardware setup

16.3.2.4.4 BIPOT/ARRAY module settings

The BIPOT/ARRAY module settings are completely defined in the NOVA software. The following user-definable settings are available, through the **Autolab control** command *(see Figure 1151, page 1000)*:

- **Cell:** a toggle control that can be used to switch WE(2) on or off.
- **Current range:** a drop-down control that can be used to select the current range of WE(2).

- **Electrode control:** defines the relationship between the cell switch of WE(2) and WE(1). When this setting is set to Linked to WE(1), the cell switch of WE(2) will automatically be set to the same status of the cell switch of WE(1). Using the Cell command, both cell switches will be toggled at the same time. When this setting is set to Independent, the cell switch of WE(2) is decoupled from the cell switch of WE(1). In that case, the cell switch of the WE(2) must be set manually, using the control Cell control provided by the Autolab control command. This setting only affects the transition from cell off to cell on.
- **Potential (V):** defines the potential difference between WE(2) and the reference electrode for the BIPOT module or between WE(2) and WE(1) in for the ARRAY mode.

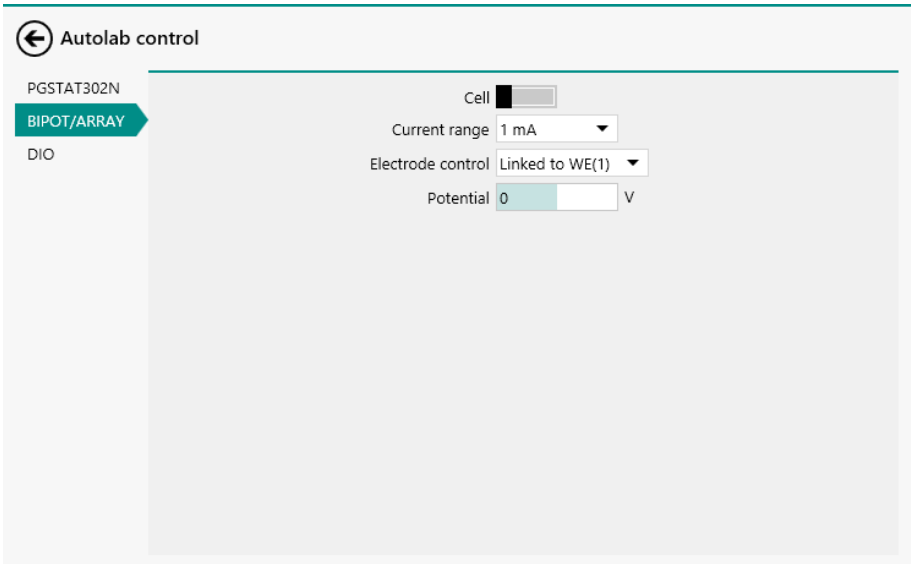


Figure 1151 The BIPOT/ARRAY module settings are defined in the Autolab control command

NOTE

The default startup Electrode control setting is linked to WE(1).

NOTE

The Cell command switches the main potentiostat and the second working electrode off automatically, regardless of the Electrode control setting.

CAUTION

When the electrode control is set to independent, a specific order must be respected to avoid current leakage between WE(1) and WE(2). Always set the cell of WE(2) to ON after the cell has been set to ON for WE(1). Always set the cell of WE(2) to OFF before the cell has been set to OFF for WE(1).

16.3.2.4.5 BIPOT/ARRAY module restrictions

Restrictions apply when using the BIPOT/ARRAY module. Both BIPOT and ARRAY modules are controlled directly from the DAC164 located in the instrument. This forces the following restrictions:

- **ECD module:** the ECD module cannot be used at the same time as the BIPOT/ARRAY module.
- **RDE or RRDE:** the remote control option of the rotating disc electrode (RDE) or rotating ring-disc electrode (RRDE) is not possible when the BIPOT/ARRAY module is used.

16.3.2.4.6 BIPOT/ARRAY module front panel connections

The BIPOT/ARRAY module is fitted with a single female BNC connector, labeled ←I.

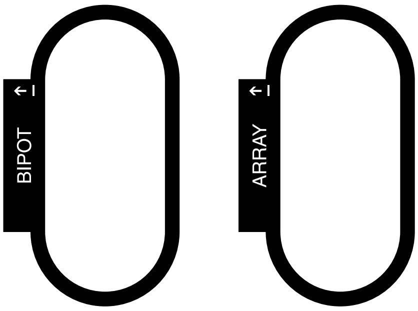


Figure 1152 The front panel labels of the BIPOT/ARRAY module (left: BIPOT module, right: ARRAY module)

The signal provided through the ←I connector on the front panel corresponds to the output of the current-to-voltage converter located on the BIPOT/ARRAY module. The output signal is a voltage, referred to the instrument ground, corresponding to the converted current according to:

$$E_{out}(\leftarrow I) = \frac{i_{(WE2)}}{[CR]}$$

Where $E_{out}(\leftarrow I)$ corresponds to the output voltage signal of the module, in V, $i_{(WE2)}$ corresponds to the current measured by the BIPOT/ARRAY module, in A and [CR] is the active current range of the BIPOT/ARRAY module.

> **NOTE**
>
> The front panel ←I BNC output is provided for information purposes only.

16.3.2.4.7 BIPOT/ARRAY module testing

Two test procedures are provided for testing the BIPOT/ARRAY module:

- For the BIPOT module, please refer to *Chapter 16.3.2.4.7.1*.
- For the ARRAY module, please refer to *Chapter 16.3.2.4.7.2*.

16.3.2.4.7.1 BIPOT module testing

NOVA is shipped with a procedure which can be used to verify that the **BIPOT** module is working as expected.

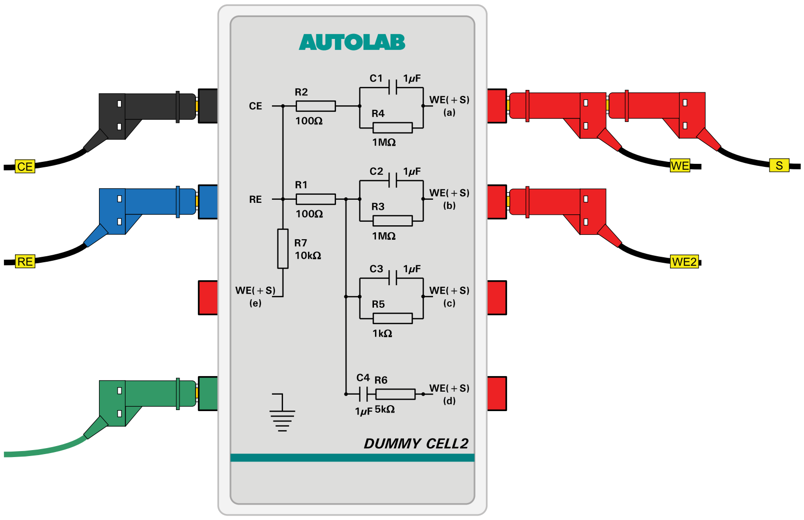
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestBIPOT** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test\TestBIPOT.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (a) and the second working electrode (BIPOT) to the Autolab dummy cell (b).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1153*.

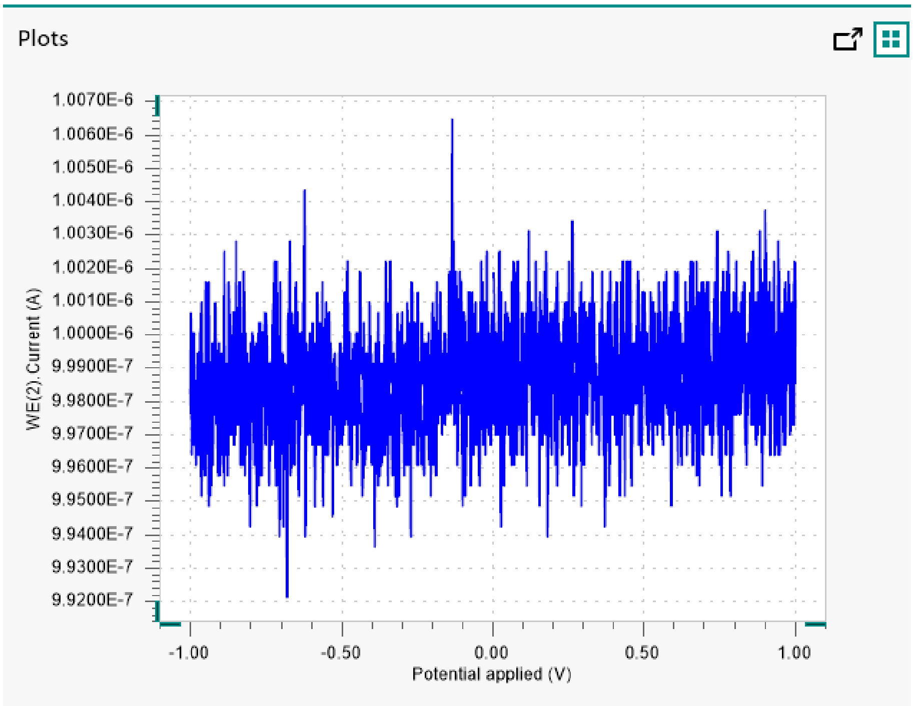
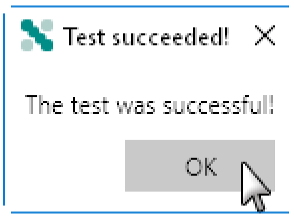


Figure 1153 The results of the TestBIPOT procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestBIPOT automatic evaluation of the data requires the following tests to succeed:

1. The average WE(2).Current measured in Bipotentiostat mode must be equal to 1 V ± 5 mV/1000100 Ω± 5 %.
2. The intercept of the WE(2).Current measured in Bipotentiostat mode must be equal to 1 V ± 5 mV/1000100 Ω± 5 %.

Both conditions must be valid for the test to succeed.

16.3.2.4.7.2 ARRAY module testing

NOVA is shipped with a procedure which can be used to verify that the **ARRAY** module is working as expected.

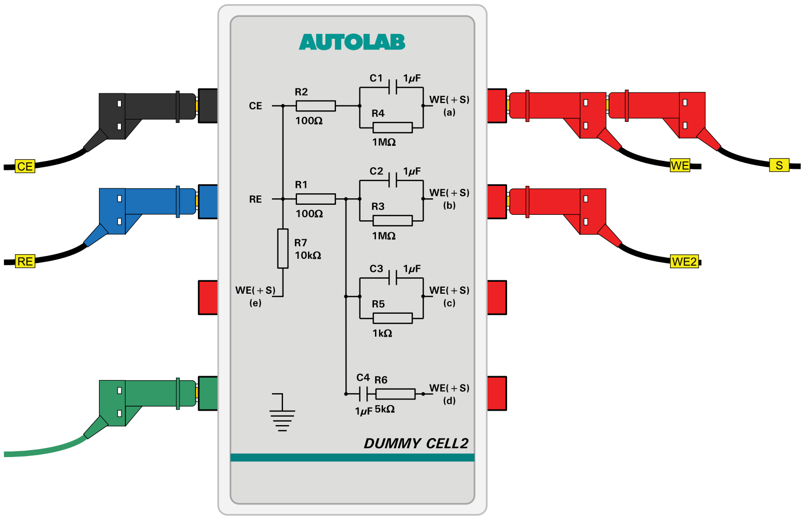
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestARRAY** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test\TestARRAY.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (a) and the second working electrode (ARRAY) to the Autolab dummy cell (b).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1154*.

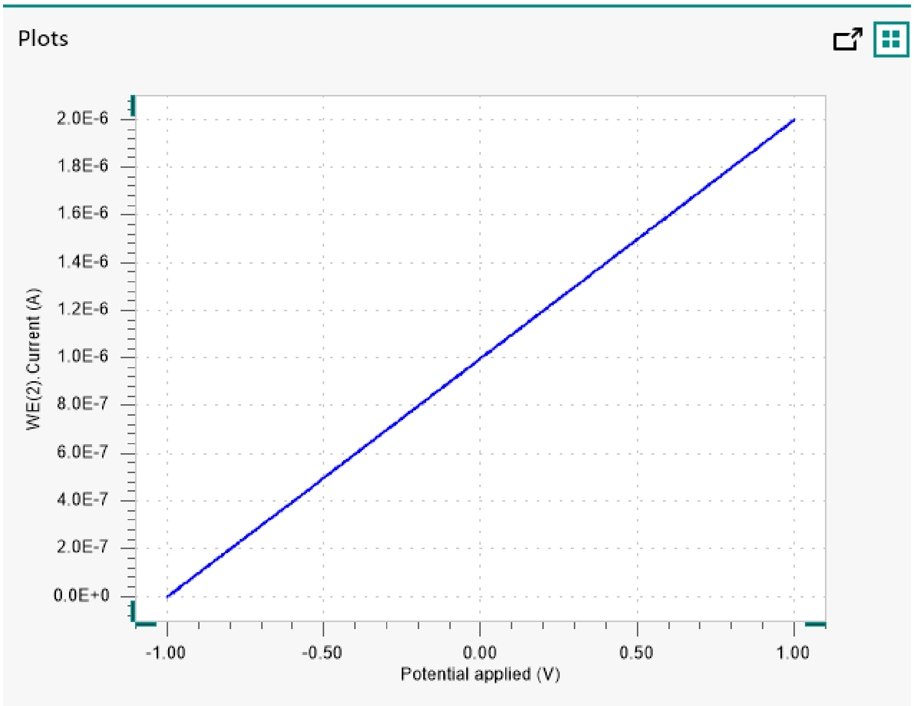
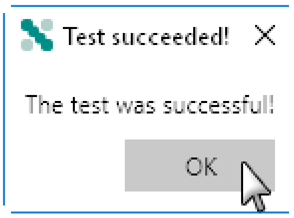


Figure 1154 The results of the TestARRAY procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestARRAY automatic evaluation of the data requires the following tests to succeed:

1. The inverted slope of the measured WE(2).Current versus the applied potential must be equal to 1000100 Ω± 5 %.
2. The intercept WE(2).Current measured in Scanning Bipotentiostat mode must be equal to 1 V ± 5 mV/1000100 Ω± 5 %.

Both conditions must be valid for the test to succeed.

16.3.2.4.8 BIPOT/ARRAY module specifications

The specifications of the BIPOT module are provided in *Table 37*.

Table 37 Specifications of the BIPOT module

Specification	Value
Operation mode	BIPOT (hardware defined)
Control DAC	DAC164
Maximum current	± 35 mA
Current ranges	100 nA to 10 mA (6 ranges)
Current accuracy	± 0.2 % of current range
Potential range	± 5 V
Potential accuracy	± 2 mV

The specifications of the ARRAY module are provided in *Table 38*.

Table 38 Specifications of the ARRAY module

Specification	Value
Operation mode	Scanning BIPOT (hardware defined)
Control DAC	DAC164
Maximum current	± 35 mA
Current ranges	100 nA to 10 mA (6 ranges)
Current accuracy	± 0.2 % of current range
Potential range	± 5 V
Potential accuracy	± 2 mV

16.3.2.5 Booster10A

The Booster10A is an **external** extension module for the Autolab potentiostat/galvanostat. This module extends the maximum current of the Autolab system to which they are connected to 10 A. The Booster10A can be used with all the measurement commands provided in NOVA.

16.3.2.5.1 Booster10A module compatibility

The Booster10A is compatible with the following instruments:

- PGSTAT302N, PGSTAT302 and PGSTAT30
- PGSTAT128N
- PGSTAT100N and PGSTAT100
- PGSTAT204 and M204
- PGSTAT20

NOTE

The Booster10A is **not** compatible with the Autolab instruments not listed above.

16.3.2.5.2 Booster10A scope of delivery

The Booster10A is supplied with the following items (when ordered for all compatible instrument, except the PGSTAT204 and M204):

- Booster10A instrument
- Digital connection cable
- 100 mΩ dummy cell *(see Figure 1155, page 1007)*

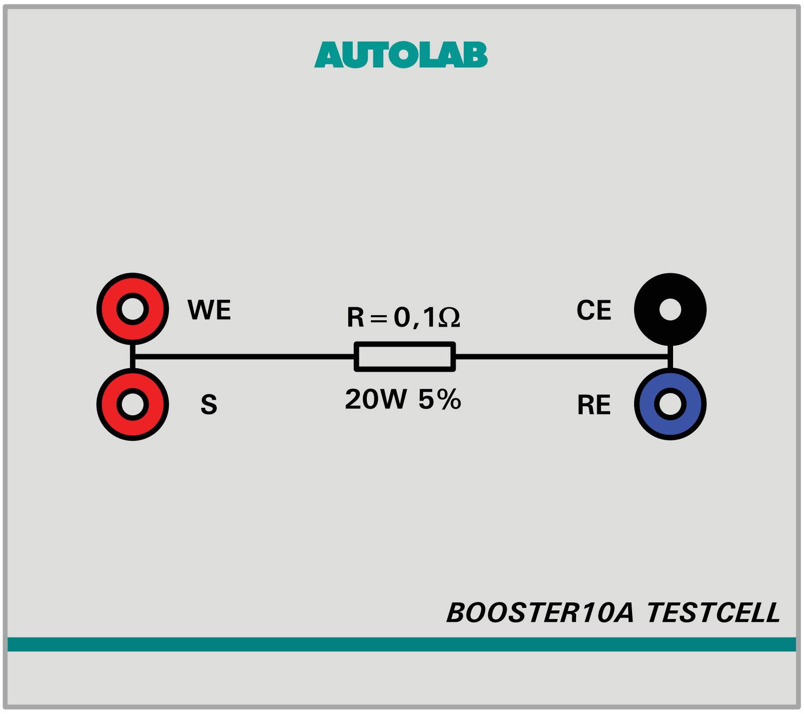


Figure 1155 The 100 mΩ dummy cell supplied with the Booster10A

The Booster10A is supplied with the following items (when ordered for the PGSTAT204 and M204):

- Booster10A instrument
- Digital adapter cable
- Cell adapter cable
- 100 mΩ dummy cell *(see Figure 1155, page 1007)*

16.3.2.5.3 Booster10A module settings

The Booster10A extends the available current ranges of the controlling instrument by adding a single 10 A current range. The module does not provide specific settings. The Booster10A can be used in two different modes, when connected to the Autolab:

- **Bypass mode:** in this mode, the Booster10A is connected to the Autolab but the extra current range provided by the Booster10A is not used. The Booster10A is bypassed and only provides connections to the electrochemical cell.
- **Operation mode:** in this mode, the extra current range provided by the Booster10A is used.

The mode of operation is controlled by the active current range. The user can specify which current range is used during a measurement or at any time by using the **Autolab control** command *(see Chapter 7.2.1, page 221)* or the **Autolab display** panel *(see Chapter 5.2.3, page 116)*.

Figure 1156 shows the additional current range provided in the Autolab control command.

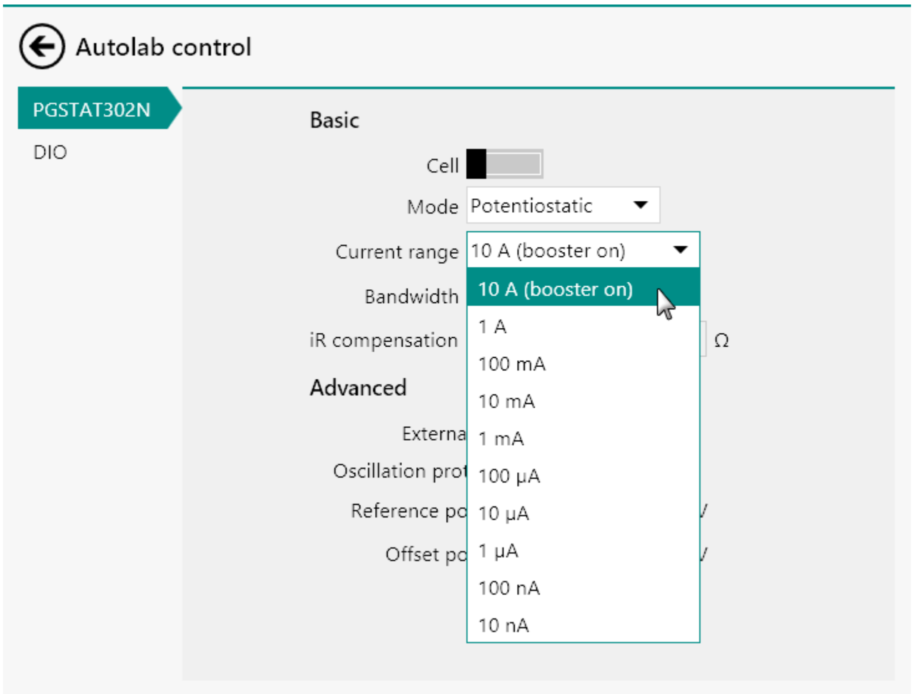


Figure 1156 The Booster10A is controlled by the current range

16.3.2.5.4 Booster10A module restrictions

Restrictions apply when using the Booster10A module:

- **Maximum current:** the maximum current provided by the Booster10A exceeds the maximum allowed current of the MUX module and the Autolab rotating disc and rotating ring disc electrodes.
- **Automatic current ranging:** the Automatic Current Ranging option cannot be used when the Booster10A is in operation, except during an impedance measurement (using the **FRA measurement** command *(see Chapter 7.6.1, page 288)* or the **FRA single frequency** command *(see Chapter 7.6.2, page 290)*) or when the Booster10A is in bypass mode. An **error** is shown in NOVA when conflicting settings are detected.
- **Instrument incompatibility:** when more than one instrument equipped with a Booster10A is connected to the same computer, then these instruments must be of the same type (either N Series Autolab or PGSTAT204/M204). It is not possible to control Booster10A using a N Series instrument and a PGSTAT204/M204 at the same time from the same computer.

16.3.2.5.5 Booster10 module front panel controls

The front panel of the Booster10A provides a number of controls and indicators, shown in *Figure 1157*.

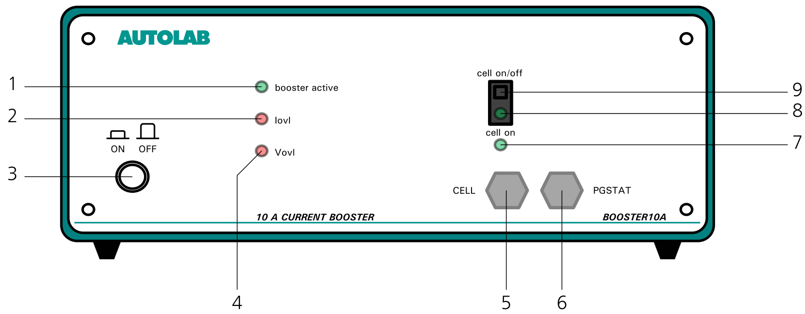


Figure 1157 Overview of the front panel of the Booster10A

1	**Booster active indicator LED** Indicates that the booster is active when lit.	**2**	**Current overload (Iovl) indicator LED** Indicates that a current overload is detected when lit.
3	**On/Off button** For switching the Booster on or off.	**4**	**Voltage overload (Vovl) indicator LED** Indicates that a voltage overload is detected when lit.
5	**Cell cable** Fixed cable providing connections to counter electrode (CE) and working electrode (WE).	**6**	**PGSTAT cable** Fixed cable providing analog connection to the Autolab PGSTAT.

7 Cell on indicator LED
Indicates that the cell is on when lit.

8 Manual cell On/Off indicator LED
The LED is lit when the cell is enabled.

9 Cell On/Off switch
For manually enabling or disabling the cell.

16.3.2.5.6 Booster10A module back plane connections

The back plane of the Booster10A provides a number of connections, shown in *Figure 1158*.

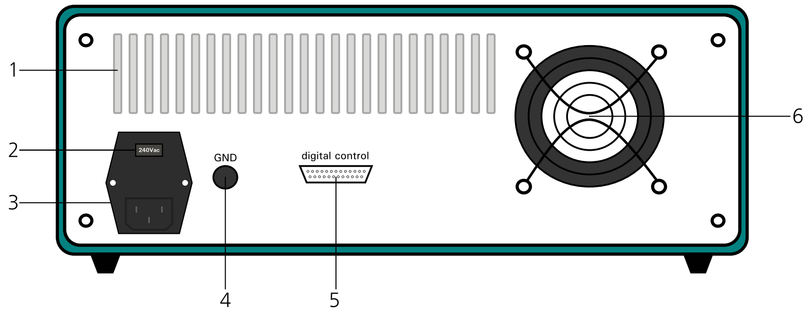


Figure 1158 The back plane of the Booster10A

1 Air flow holes
Required for cooling the Booster10A during operation.

2 Mains voltage indicator
Indicates the mains voltage settings of the Booster10A.

3 Mains connection socket
For connecting the Booster10A to the mains supply.

4 GND plug
For grounding purposes.

5 DIO connector (digital control)
For connecting the digital control cable to interface with the Autolab PGSTAT.

6 Fan
Required for cooling the Booster10A during operation.

CAUTION

Make sure that the mains voltage indicator is set properly before switching the Booster10A on.

16.3.2.5.7 Booster10A installation and configuration

The Booster10A can be used in combination with any compatible instrument. The installation and configuration can be carried out by the end-user at any time.

Depending on the type of instrument it is connected to, the Booster10A has to be installed and configured according to a specific procedure:

1. For all the compatible Autolab instruments, except the PGSTAT204 and the M204, please refer to *Chapter 16.3.2.5.7.1*.

2. For the PGSTAT204 and the M204, please refer to *Chapter 16.3.2.5.7.2*.

16.3.2.5.7.1 Booster10A installation and configuration

The following steps describe how to install and configure the Booster10A. These steps apply to all the compatible instruments, except the PGSTAT204 and the M204 module.

1 Remove the CE/WE cable

Unscrew and remove the CE/WE cable from the PGSTAT front panel panel.

NOTE

It is recommended to store this cable carefully for future use.

2 Connect the Booster10A PGSTAT cable to the PGSTAT

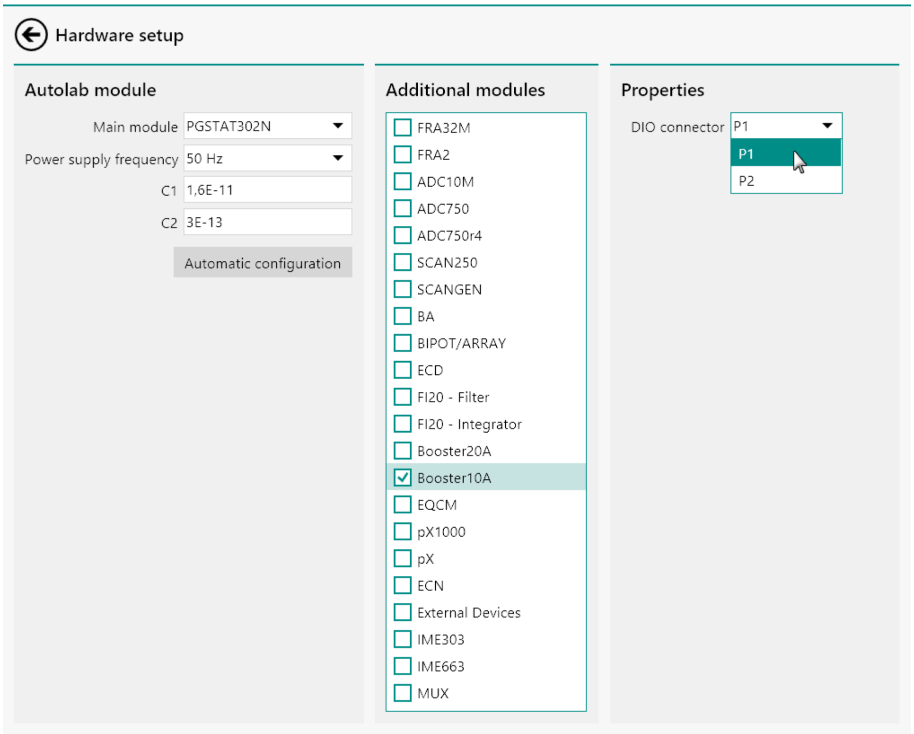
Connect the PGSTAT cable, located on the front panel of the Booster10A (item 6 in *Figure 1157*) to the CE/WE connector of the Autolab PGSTAT.

3 Connect the digital control cable

Connect the digital control cable, supplied with the Booster10A, to the digital control connector located on the backplane of the Booster10A (item 5 in *Figure 1158*). Connect the other end of the cable to one of the two DIO connectors (P1 or P2) located on the back plane of the Autolab PGSTAT.

4 Specify the hardware setup

Adjust the hardware setup and specify the DIO connector (P1 or P2) in the **Properties** panel.



5 **Connect the cell**

Use the CE and WE connectors provided by the Booster10A and the RE and S connectors provided by the Autolab to the electrochemical cell.

16.3.2.5.7.2 Booster10A installation and configuration (PGSTAT204 and M204 only)

For the Autolab PGSTAT204 and the M204 module, a special set of cables is required for connecting the Booster10A. The set of cables includes two cables:

1. **DIO adapter cable:** a female 25 pin SUB-D to male 15 pin SUB-D adapter cable.
2. **Cell adapter cable:** a dedicated cell cable assembly providing an interface between the PGSTAT204 or M204 module and the Booster10A *(see Figure 1159, page 1013)*.

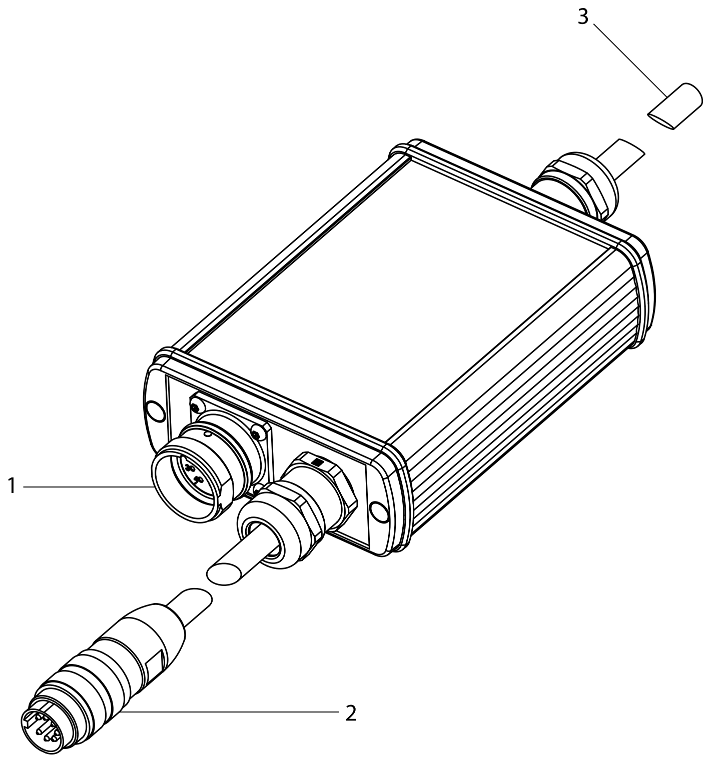


Figure 1159 The cell adapter cable assembly

1 Booster10A PGSTAT cable connector
For connecting the PGSTAT cable from the Booster10A (item 6 in *Figure 1157, page 1009*).

2 PGSTAT204/M204 cell cable connector
For connecting to the PGSTAT204 or M204 module cell connector (item 4 in *Figure 1083, page 909* or item 5 in *Figure 1092, page 923*).

3 RE/S cable
Cable providing reference electrode (RE) and sense electrode (S) to connect to the electrochemical cell.

The following steps describe how to install and configure the Booster10A for the PGSTAT204 and the M204 module.

1 Remove the cell cable

Unscrew and remove the cell cable from the PGSTAT204 front panel or the M204 module front panel.

NOTE

It is recommended to store this cable carefully for future use.

2 **Connect the Booster10A PGSTAT cable to the Booster10A PGSTAT cable connector**

Connect the PGSTAT cable, located on the front panel of the Booster10A (item 6 in *Figure 1157, page 1009*) to the Booster10A PGSTAT cable connector (item 1 in *Figure 1159)*.

3 **Connect the DIO adapter cable**

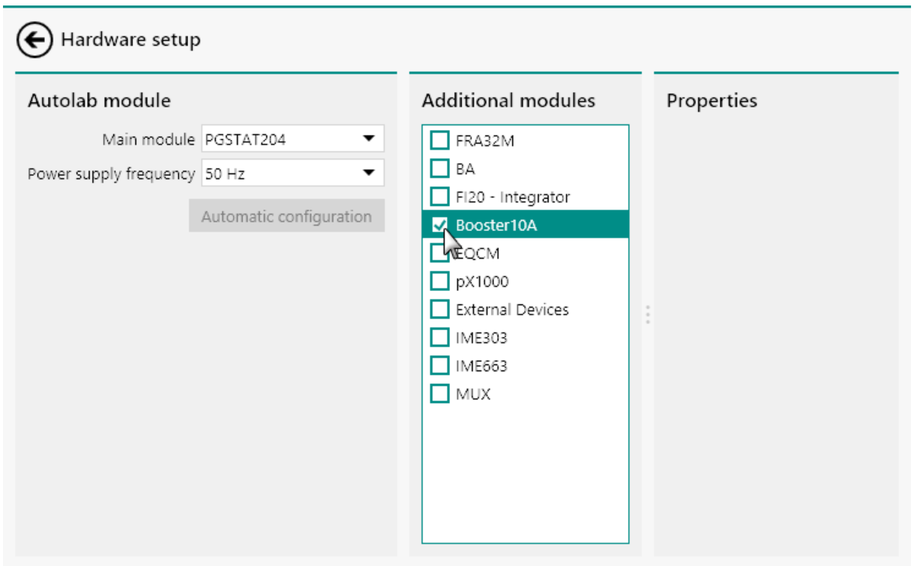
Connect the DIO adapter cable, supplied with the Booster10A, to the digital control connector located on the backplane of the Booster10A (item 5 in *Figure 1158, page 1010*). Connect the other end of the cable to the DIO connector located on the front panel of the Autolab PGSTAT204 or the M204 module (item 1 in *Figure 1083, page 909* or item 2 in *Figure 1092, page 923*).

4 **Connect the PGSTAT204/M204 cell cable connector to the PGSTAT204 or M204**

Connect the PGSTAT204/M204 cell cable connector (item 2 in *Figure 1159)* to the cell cable connector located on the front panel of the PGSTAT204 or M204 (item 4 in *Figure 1083, page 909* or item 5 in *Figure 1092, page 923*).

5 **Specify the hardware setup**

Adjust the hardware setup.



6 Connect the cell

Use the CE and WE connectors provided by the Booster10A and the RE and S connectors provided by the cell adapter cable (item 3 in *Figure 1159*) to connect to the cell.

16.3.2.5.8 Boosert10A testing

NOVA is shipped with a procedure which can be used to verify that the **Booster10A** is working as expected.

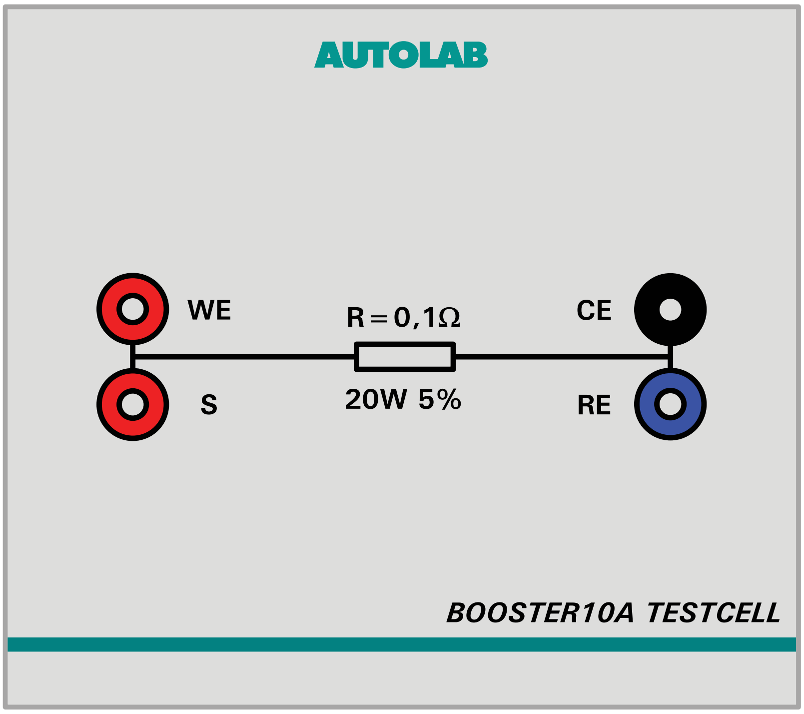
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestBooster10A** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases \Module test\TestBoosert10A.nox)

2 Connect the Autolab dummy cell

Connect the Autolab and Booster10A to the Booster10A test cell.



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1160*.

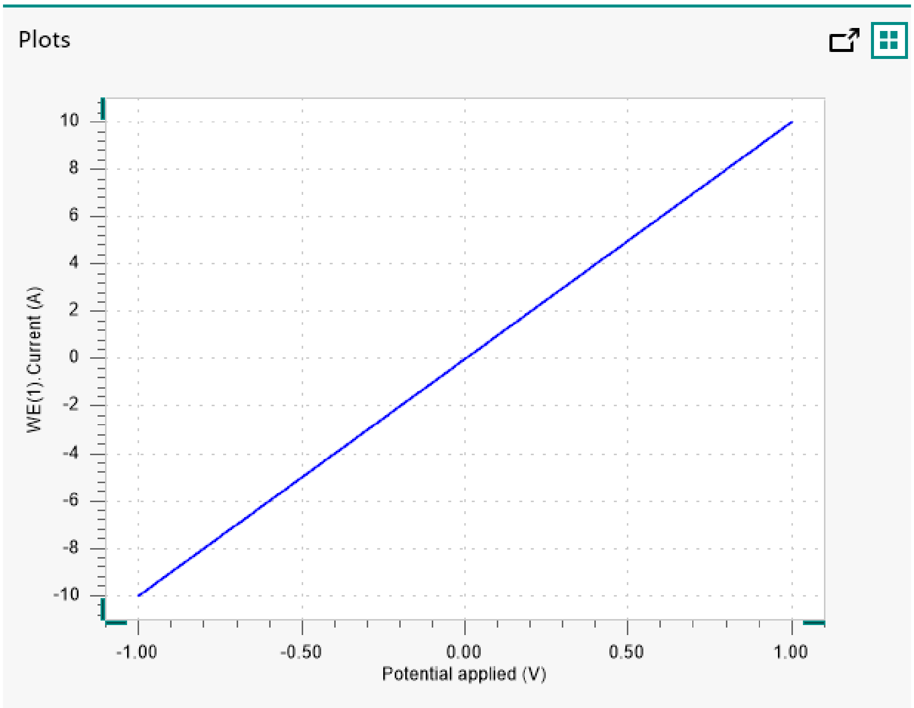
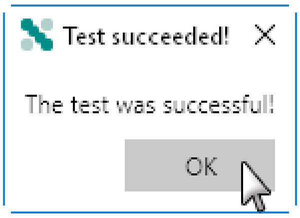


Figure 1160 The results of the TestBooster10A procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestBooster10A automatic evaluation of the data requires the following tests to succeed:

1. The inverted slope of the measured current versus the applied potential must be equal to 100 mΩ± 5 %.
2. The intercept of the measured current versus the applied potential must be equal to ± 4 mV divided by 100 mΩ± 5 %.

Both conditions must be valid for the test to succeed.

16.3.2.5.9 Booster10A module specifications

The specifications of the Booster10A module are provided in *Table 39*.

Table 39 Specifications of the Booster10A

Specification	Value
Maximum current	10 A
Compliance voltage	20 V
Maximum power	150 W
Current resolution	± 0.0003 %
Current accuracy	± 0.5 % of current range
PSTAT bandwidth	4 kHz
GSTAT bandwidth	2.5 kHz

16.3.2.6 Booster20A

The Booster20A is an **external** extension module for the Autolab potentiostat/galvanostat. This module extends the maximum current of the Autolab system to which they are connected to 20 A. The Booster20A can be used with all the measurement commands provided in NOVA.

16.3.2.6.1 Booster20A module compatibility

The Booster20A is compatible with the following instruments:

- PGSTAT302N, PGSTAT302 and PGSTAT30

NOTE

The Booster20A is **not** compatible with the Autolab instruments not listed above.

16.3.2.6.2 Booster20A scope of delivery

The Booster20A is supplied with the following items:

- Booster20A instrument
- Digital connection cable
- Analog connection cable
- Emergency stop button
- 50 mΩ dummy cell *(see Figure 1161, page 1018)*

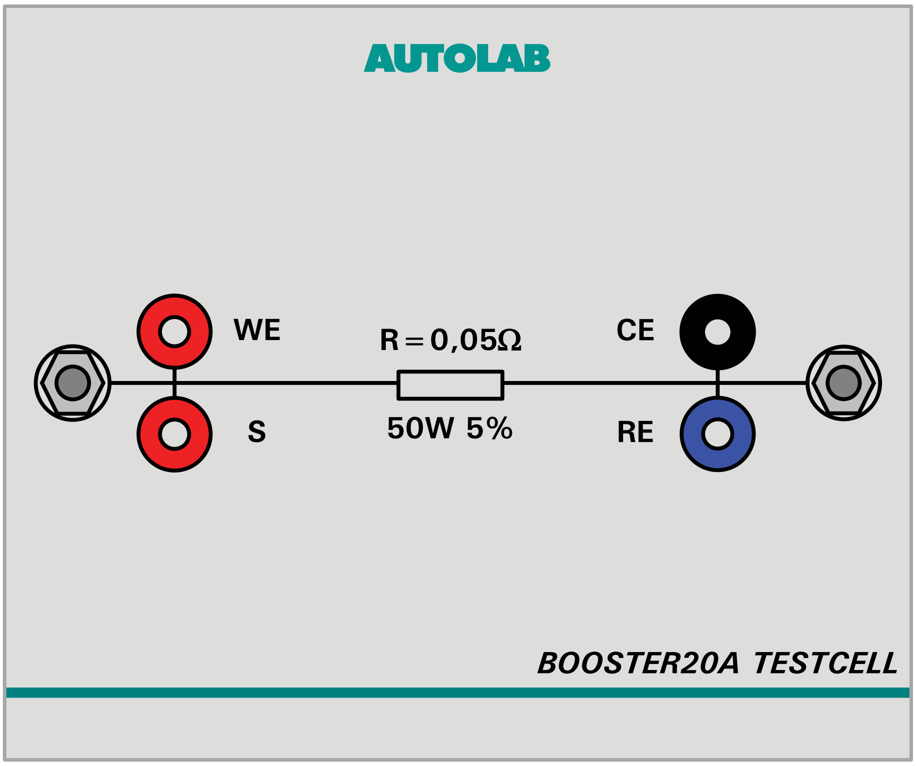


Figure 1161 The 50 mΩ dummy cell supplied with the Booster20A

16.3.2.6.3 Booster20A module settings

The Booster20A extends the available current ranges of the controlling instrument by adding a single 20 A current range. The module does not provide specific settings. The Booster20A can be used in two different modes, when connected to the Autolab:

- **Bypass mode:** in this mode, the Booster20A is connected to the Autolab but the extra current range provided by the Booster20A is not used. The Booster20A is bypassed and only provides connections to the electrochemical cell.
- **Operation mode:** in this mode, the extra current range provided by the Booster20A is used.

The mode of operation is controlled by the active current range. The user can specify which current range is used during a measurement or at any time by using the **Autolab control** command *(see Chapter 7.2.1, page 221)* or the **Autolab display** panel *(see Chapter 5.2.3, page 116)*.

Figure 1162 shows the additional current range provided in the Autolab control command.

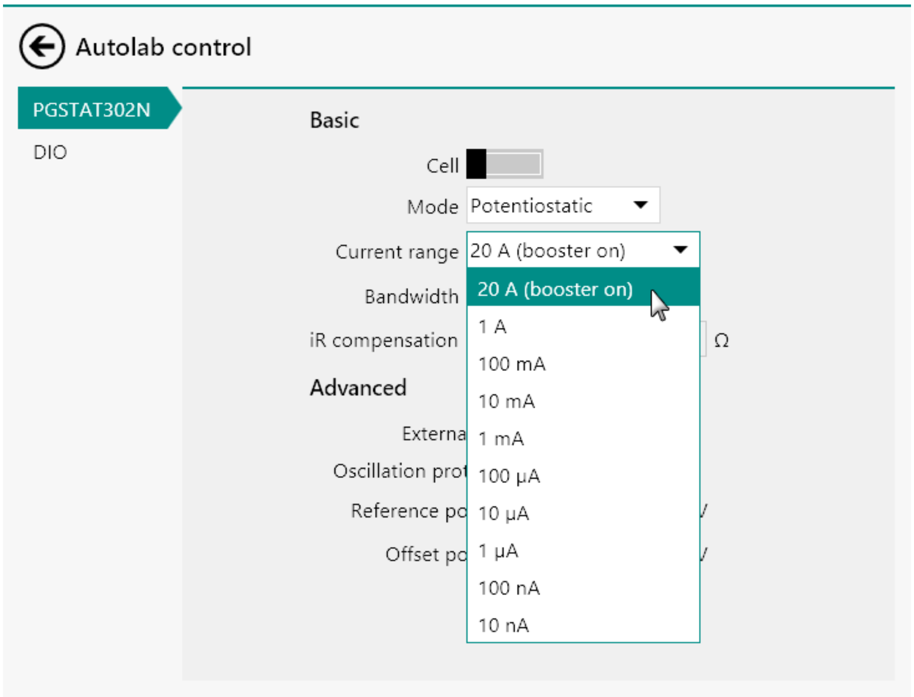


Figure 1162 The Booster20A is controlled by the current range

16.3.2.6.4 Booster20A module restrictions

Restrictions apply when using the Booster20A module:

- **Maximum current:** the maximum current provided by the Booster20A exceeds the maximum allowed current of the MUX module and the Autolab rotating disc and rotating ring disc electrodes.
- **Automatic current ranging:** the Automatic Current Ranging option cannot be used when the Booster20A is in operation, except during an impedance measurement (using the **FRA measurement** command *(see Chapter 7.6.1, page 288)* or the **FRA single frequency** command *(see Chapter 7.6.2, page 290)*) or when the Booster20A is in bypass mode. An **error** is shown in NOVA when conflicting settings are detected.

16.3.2.6.5 Booster20 module front panel controls

The front panel of the Booster20A provides a number of controls and indicators, shown in *Figure 1163*.

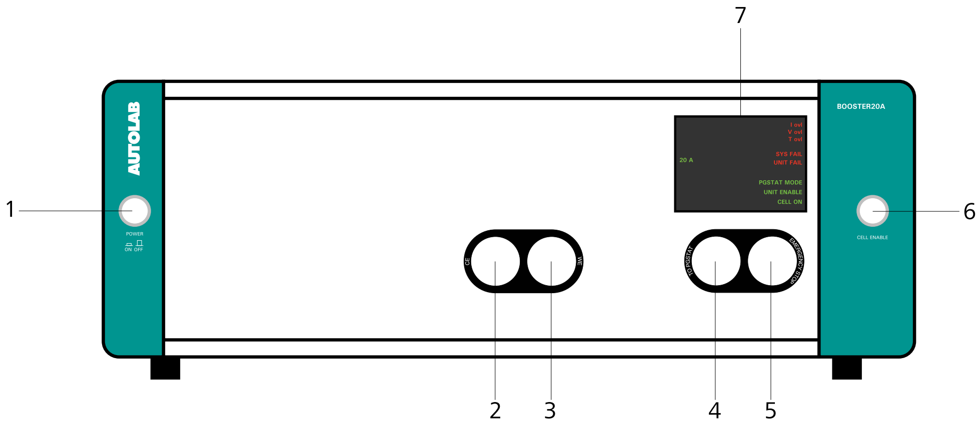


Figure 1163 Overview of the front panel of the Booster20A

1 On/Off button For switching the Booster on or off.	**2 CE (Counter electrode) cable connector** For connecting the CE cable.
3 WE (Working electrode) cable connector For connecting the WE cable.	**4 To PGSTAT connector** For connecting the analog control cable between the Booster and the Autolab PGSTAT.
5 Emergency stop connector For connecting the emergency stop button.	**6 Cell enable button** For enabling or disabling the cell.
7 Display For indications and warnings.	

The display (item 7 in *Figure 1163*) is used to provide information about the Booster20A to the user. *Figure 1164* shows a detail of this display.

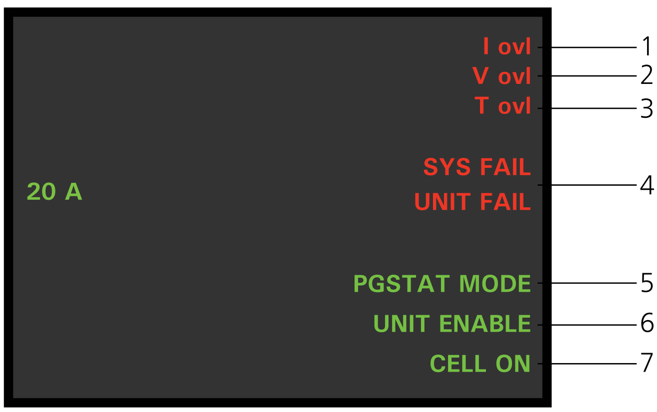


Figure 1164 Overview of the display of the Booster20A

1 I ovl LED Indicates that a current overload is detected when lit.	**2 V ovl LED** Indicates that a voltage overload is detected when lit.

3 **T ovl LED**
Indicates that a temperature overload is detected when lit.

4 **SYS FAIL and UNIT FAIL LEDs**
Indicates that the Booster20A is malfunctioning when lit.

5 **PGSTAT MODE LED**
Indicates that the Booster20A is in **Bypass** mode when lit.

6 **UNIT ENABLE**
Indicates that the Booster20A is in operation when lit.

7 **CELL ON**
Indicates that the cell is on when lit.

16.3.2.6.6 Emergency stop button

The Booster20A is supplied with an emergency stop button *(see Figure 1165, page 1021)*.

Figure 1165 The emergency switch of the Booster20A

The emergency stop button is connected to the front panel of the Booster20A (item 5 in *Figure 1163*).

The emergency stop button can be pressed at any time to immediately disconnect the Booster20A from the electrochemical cell. The stop button remains engaged until it is disengaged by the user. To disengage the emergency stop button, rotate the red knob counter-clockwise until it releases.

NOTE

Pressing the emergency stop button does not stop the NOVA measurement.

16.3.2.6.7 Booster20A module back plane connections

The back plane of the Booste20A provides a number of connections, shown in *Figure 1166*.

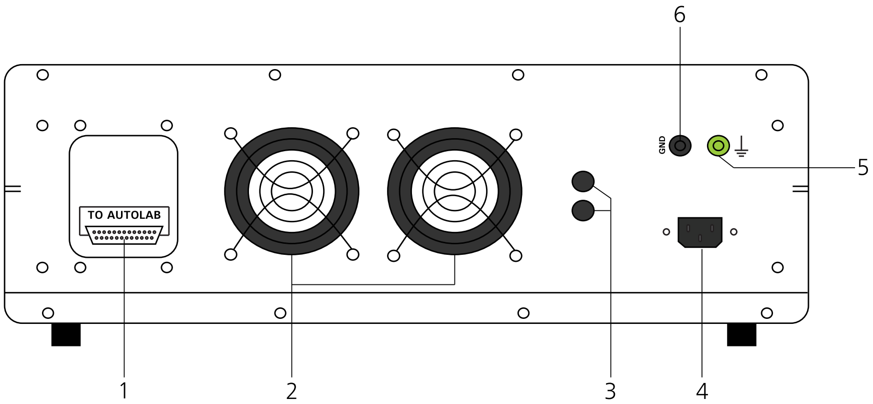


Figure 1166 The back plane of the Booster20A

1	**DIO connector (TO AUTOLAB)** For connecting the digital control cable to interface with the Autolab PGSTAT.	**2**	**Fans** Required for cooling the Booster20A during operation.
3	**Fuse holders** Fuse holders containing the fuses protecting the Booster20A.	**4**	**Mains connection socket** For connecting the Booster10A to the mains supply.
5	**Earth plug** For connections to the protective earth.	**6**	**GND plug** For connections to the ground.

16.3.2.6.8 Booster20A installation and configuration

The Booster20A can be used in combination with any compatible instrument. The installation and configuration can be carried out by the end-user at any time. The following steps describe how to install and configure the Booster20A.

1 Remove the CE/WE cable

Unscrew and remove the CE/WE cable from the PGSTAT front panel panel.

NOTE

It is recommended to store this cable carefully for future use.

2 **Connect the TO PGSTAT cable to the PGSTAT**

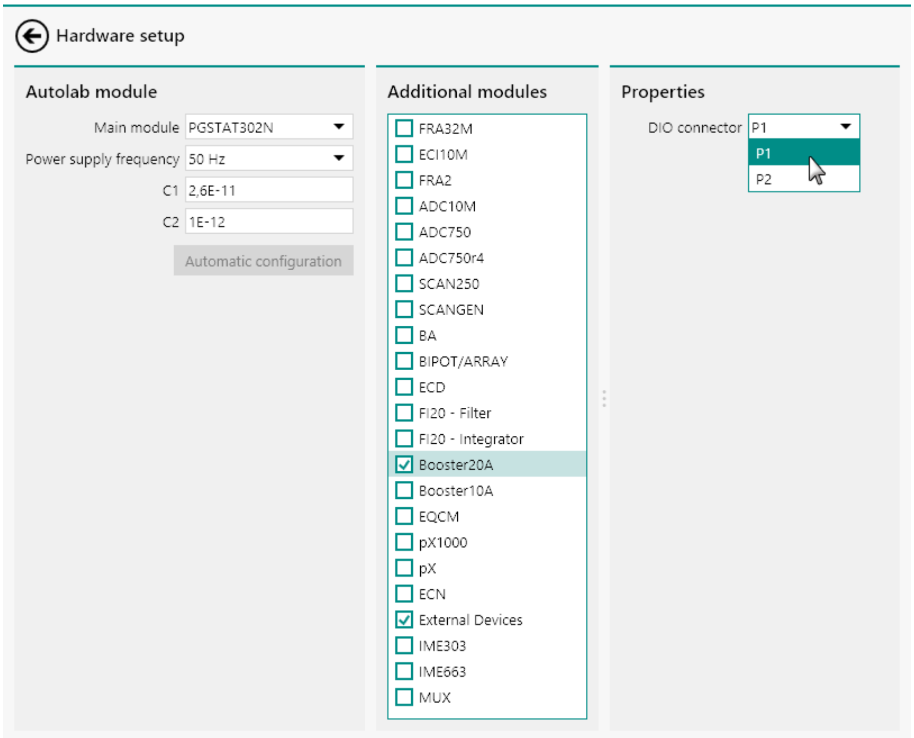
Connect the analog connection cable, supplied with the Booster20A, to the TO PGSTAT connector, located on the front panel of the Booster20A (item 4 in *Figure 1163*) and to the CE/WE connector of the Autolab PGSTAT.

3 **Connect the digital control cable**

Connect the digital control cable, supplied with the Booster20A, TO AUTOLAB connector located on the backplane of the Booster20A (item 1 in *Figure 1166*). Connect the other end of the cable to one of the two DIO connectors (P1 or P2) located on the back plane of the Autolab PGSTAT.

4 **Specify the hardware setup**

Adjust the hardware setup and specify the DIO connector (P1 or P2) in the **Properties** panel.



5 **Connect the emergency stop button**

Connect the emergency stop button to the dedicated connector on the front panel of the Booster20A (item 5 in *Figure 1163*).

6 Connect the WE cable

Connect the WE cable to the dedicated connector on the front panel of the Booster20A (item 3 in *Figure 1163*).

7 Connect the CE cable

Connect the CE cable to the dedicated connector on the front panel of the Booster20A (item 2 in *Figure 1163*).

8 Connect the cell

Use the CE and WE connectors provided by the Booster20A and the RE and S connectors provided by the Autolab to the electrochemical cell.

16.3.2.6.9 Booster20A testing

NOVA is shipped with a procedure which can be used to verify that the **Booster20A** is working as expected.

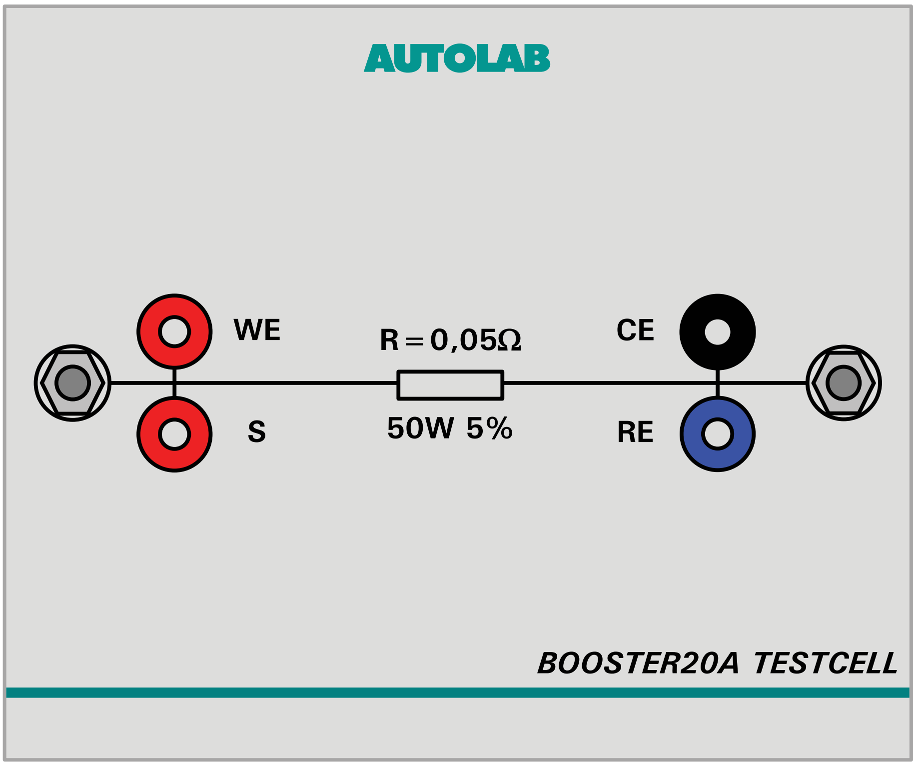
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestBooster20A** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases \Module test\TestBoosert20A.nox)

2 Connect the Autolab dummy cell

Connect the Autolab and Booster20A to the Booster20A test cell.



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1167*.

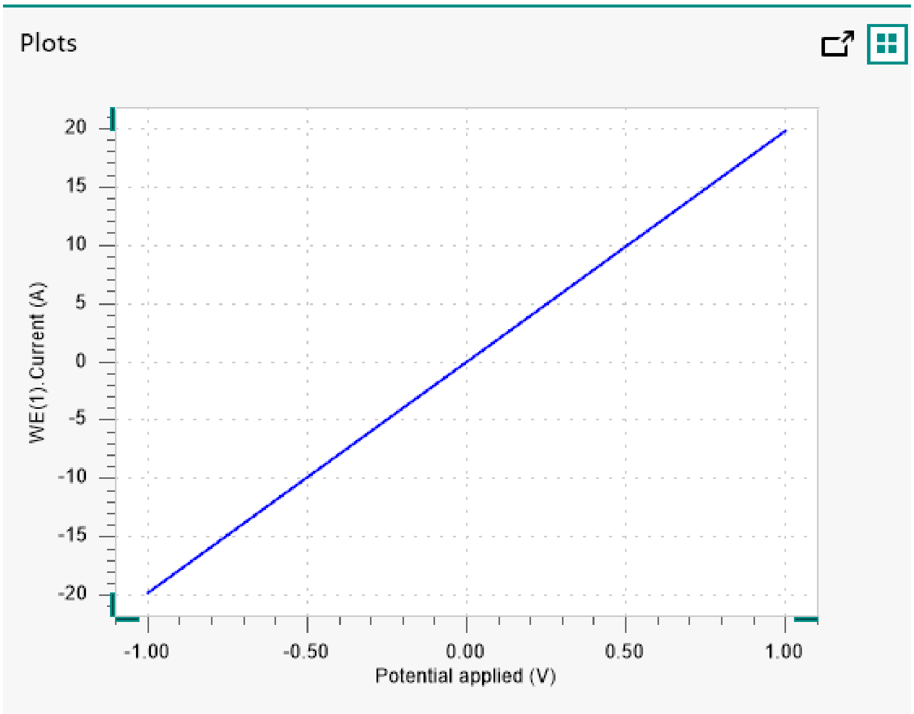
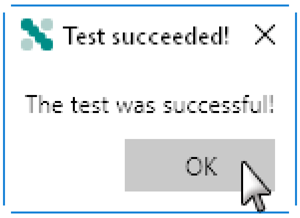


Figure 1167 The results of the TestBooster20A procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestBooster20A automatic evaluation of the data requires the following tests to succeed:

1. The inverted slope of the measured current versus the applied potential must be equal to 50 mΩ± 5 %.
2. The intercept of the measured current versus the applied potential must be equal to ± 4 mV divided by 50 mΩ± 5 %.

Both conditions must be valid for the test to succeed.

16.3.2.6.10 Booster20A module specifications

The specifications of the Booster20A module are provided in *Table 40*.

Table 40 Specifications of the Booster20A

Specification	Value
Maximum current	20 A
Compliance voltage	20 V
Maximum power	350 W
Current resolution	± 0.0003 %
Current accuracy	± 0.2 % of current range
PSTAT bandwidth	18 kHz
GSTAT bandwidth	40 kHz

16.3.2.7 ECD module

The ECD is an extension module for the Autolab PGSTAT. With the ECD module, it is possible to perform measurements at extremely low currents. The ECD module adds two extra current ranges (1 nA and 100 pA) and lowers the current resolution of the instrument by a factor of 100. An internal Sallen-Key filter is available as well, so that noise can be filtered out.

The ECD module is also fitted with an offset compensation circuit. This can be used to compensate the DC current, thus enabling current measurements at the highest possible resolution.

16.3.2.7.1 ECD module compatibility

The ECD module is available for the following instruments:

- PGSTAT302N, PGSTAT302 and PGSTAT30
- PGSTAT128N and PGSTAT12
- PGSTAT100N and PGSTAT100
- PGSTAT20
- PGSTAT10

NOTE

The ECD module is **not** compatible with the Autolab instruments not listed above.

16.3.2.7.2 ECD module scope of delivery

The ECD module is supplied with the following items:

- ECD module
- ECD module label

16.3.2.7.3 ECD hardware setup

To use the **ECD** module, the hardware setup needs to be adjusted. The checkbox for the module needs to be ticked *(see Figure 1168, page 1028)*.

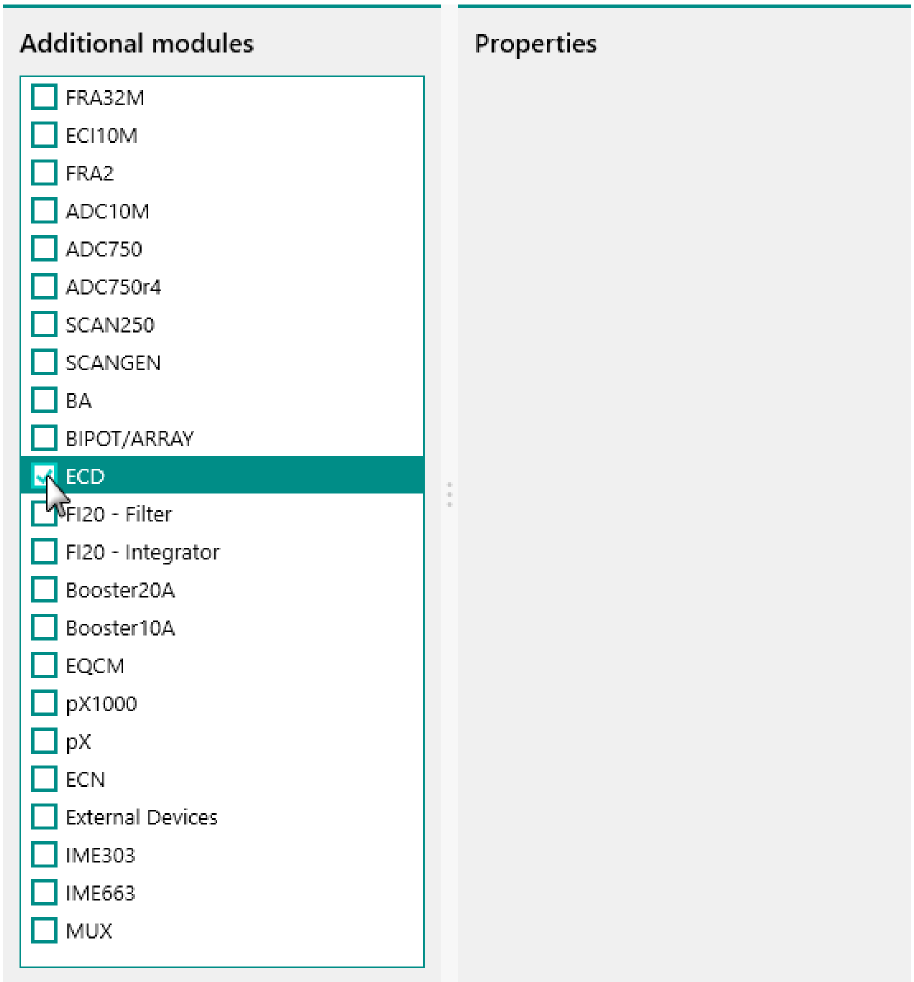


Figure 1168 The ECD module is selected in the hardware setup

16.3.2.7.4 ECD module settings

The ECD module settings are completely defined in the NOVA software. The following user-definable settings are available, through the **Autolab control** command *(see Figure 1169, page 1029)*:

- **ECD:** a toggle which can be used to switch the ECD module on or off.

- **Filter time:** a drop-down control which can be used to switch the filter the current measured through the ECD module on or off and to specify the filter time constant. The filter time can be set to off, 0.1 s, 1 s or 5 s. The higher the time constant, the heavier the filtering of the current.
- **Offset compensation (A):** defines the offset compensation current, in µA. This offset is subtracted from the current measured by the ECD module. The offset compensation can be specified in the range of ± 1 µA.

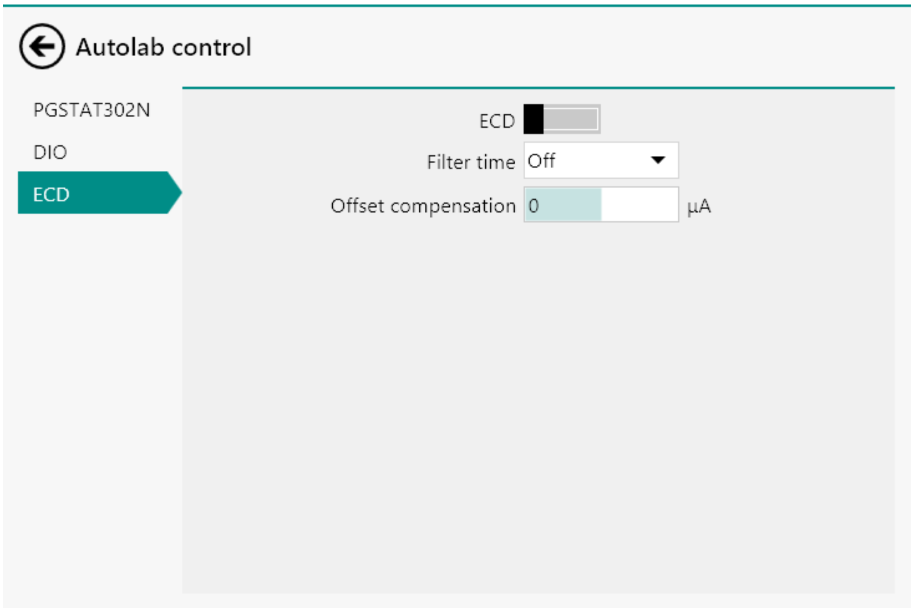


Figure 1169 The ECD module settings are defined in the Autolab control command

When the ECD module is switched on, the additional current ranges provided by the ECD module can be used. These current ranges can be selected using the drop-down control provided in the **Autolab control** command *(see Figure 1170, page 1030)*.

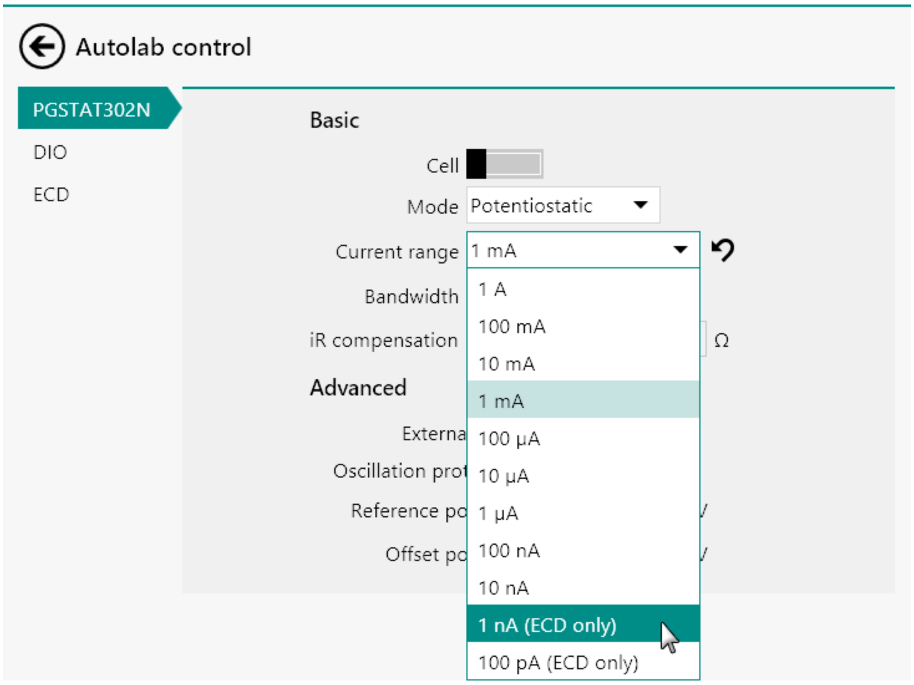


Figure 1170 The current ranges provided by the ECD module can be selected in the Autolab control command

16.3.2.7.5 ECD module restrictions

Restrictions apply when using the ECD module. The offset compensation of the ECD module is controlled directly from the DAC164 located in the instrument. This forces the following restrictions:

- **BIPOT/ARRAY module:** the BIPOT or ARRAY module cannot be used at the same time as the ECD module.
- **RDE or RRDE:** the remote control option of the rotating disc electrode (RDE) or rotating ring-disc electrode (RRDE) is not possible when the ECD module is used.

Additionally, the following restrictions apply to the available current ranges and the automatic current ranging option *(see Figure 1171, page 1031)*:

- **Current range restrictions:** when the ECD module is used, the 1 mA and higher current ranges are no longer usable by the Autolab. An **error** message is provided by NOVA when this situation is encountered.
- **Automatic current ranging restriction:** when the ECD on-board filter is used, the automatic current ranging option is not available. An **error** message is provided by NOVA when this situation is encountered.

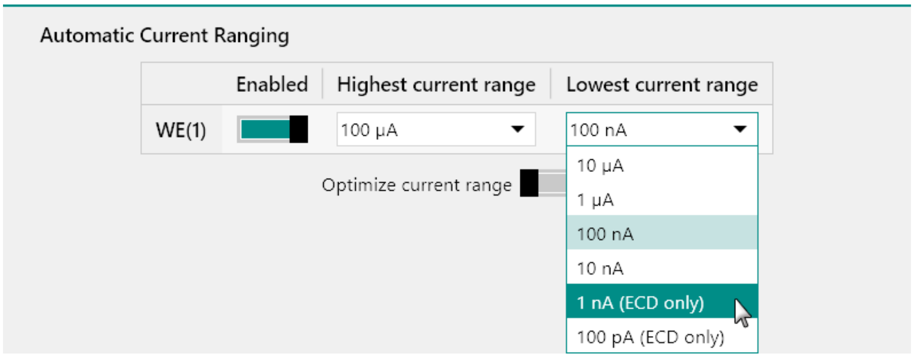


Figure 1171 The ECD current ranges are available in the Automatic current ranging option

The ECD module is also not available for use in combination with galvanostatic measurement and measurements involving the FRA module.

Finally, the current ranges provided by the ECD module have a limited bandwidth. When the interval time specified in the procedure is too small with respect to the bandwidth of the ECD current ranges, a warning is provided by the procedure validation. The bandwidth values of the ECD module are reported in *Table 41*.

Table 41 Overview of the bandwidth of the ECD module current ranges

Current range	Bandwidth
100 µ and 10 µA	2000 Hz
1 µA	1000 Hz
100 nA	250 Hz
10 nA	100 Hz
1 nA	50 Hz
100 pA	10 Hz

CAUTION

It is possible to force the procedure to continue despite the bandwidth warning. This is not recommended since the measured data could be affected by this limitation and be invalid.

16.3.2.7.6 ECD module front panel connections

The ECD module is fitted with a single female BNC connector, labeled ←I *(see Figure 1172, page 1032)*.

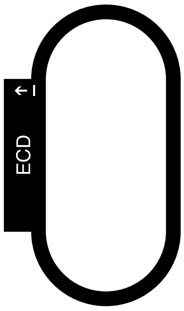


Figure 1172 The front panel label of the ECD module

The signal provided through the ←I connector on the front panel corresponds to the output of the current-to-voltage converter located on the ECD module. The output signal is a voltage, referred to the instrument ground, corresponding to the converted current according to:

$$E_{out}(\leftarrow I) = \frac{i_{(ECD)}}{[CR]}$$

Where $E_{out}(\leftarrow I)$ corresponds to the output voltage signal of the module, in V, $i_{(ECD)}$ corresponds to the current measured by the ECD module, in A and [CR] is the active current range of the ECD module.

NOTE

The front panel ←I BNC output is provided for information purposes only.

16.3.2.7.7 ECD module testing

NOVA is shipped with a procedure which can be used to verify that the **ECD** module is working as expected.

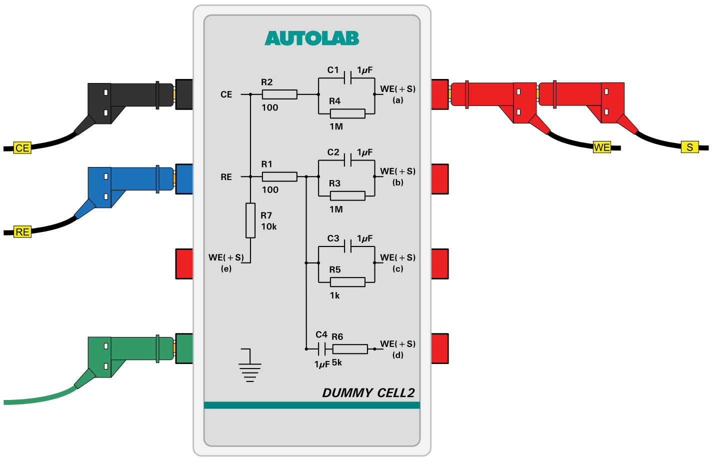
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestECD** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test \TestECD.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (a).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1173*.

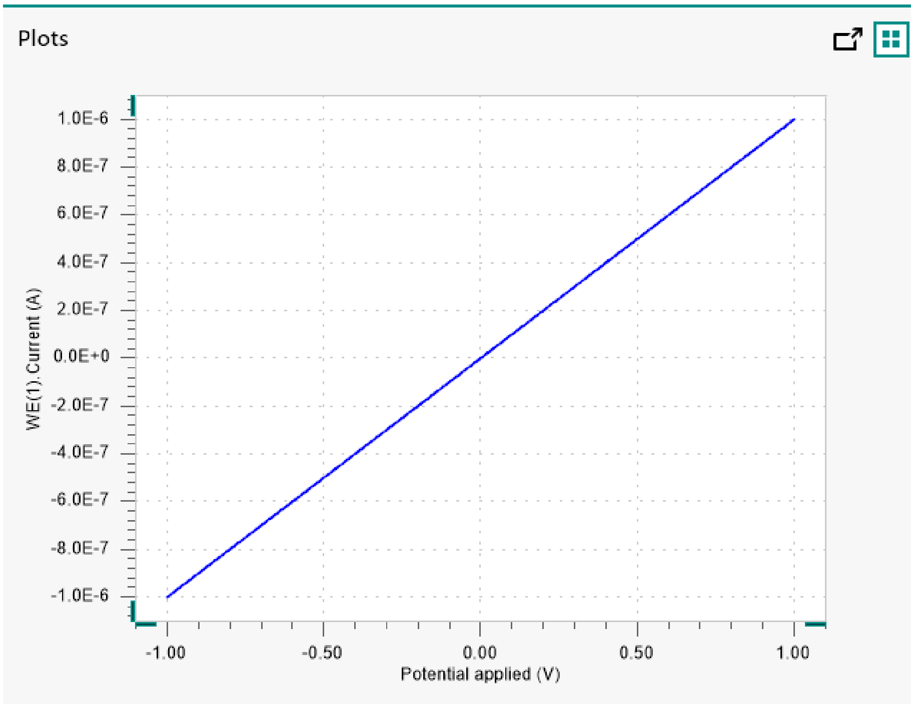
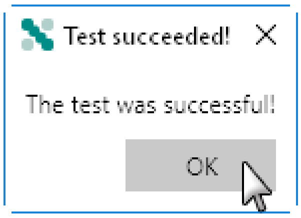


Figure 1173 The results of the TestECD procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestECD automatic evaluation of the data requires the following tests to succeed:

1. The residual current, determined by subtracting the expected current from the measured current, must be smaller or equal than ± 20 nA.
2. The inverted slope of the measured current versus the applied potential must be equal to 1000100 Ω± 5 %.
3. The intercept of the measured current versus the applied potential must be equal to ± 4 mV divided by 1000100 Ω± 5 %.

All three conditions must be valid for the test to succeed.

16.3.2.7.8 ECD module specifications

The specifications of the ECD module are provided in *Table 42*.

Table 42 Specifications of the ECD module

Specification	Value
Current ranges	100 µA to 100 pA, in 7 ranges
Currant accuracy	± 0.5 % of current range
Current offset	± 2 pA
On-board filter	3rd order Sallen-Key
Filter time constants	10 ms, 100 ms and 500 ms
Maximum offset compensation	± 1 µA

16.3.2.8 ECI10M module

The ECI10M is an extension module for the Autolab PGSTAT. With the ECI10M module, it is possible to perform electrochemical impedance spectroscopy measurements up to a frequency of 10 MHz.

The ECI10M module is an auxiliary external high bandwidth potentiostat/galvanostat that is used instead of the main Autolab PGSTAT during high frequency impedance measurements.

The ECI10M consist of a module installed in the Autolab and an external interface to be placed close to the cell.

CAUTION

For electrochemical impedance spectroscopy measurements, the **FRA32M** module must be installed in the same instrument as the **ECI10M** *(see Chapter 16.3.2.13, page 1091)*.

16.3.2.8.1 ECI10M module compatibility

The ECI10M module is available for the following instruments:

- PGSTAT302N with serial number **AUT83680** or higher.
- PGSTAT128N with serial number **AUT83680** or higher.

NOTE

The **ECI10M** module is **not** compatible with the Autolab instruments not listed above.

16.3.2.8.2 ECI10M module scope of delivery

The ECI10M module is supplied with the following items:

- ECI10M module
- ECI10M module label
- ECI10M external interface
- Five 25 cm long banana to banana cables (2 red, 1 blue, 1 black, 1 green)

16.3.2.8.3 ECI10M hardware setup

To use the **ECI10M** module, the hardware setup needs to be adjusted. The checkbox for the module needs to be ticked *(see Figure 1174, page 1036)*.

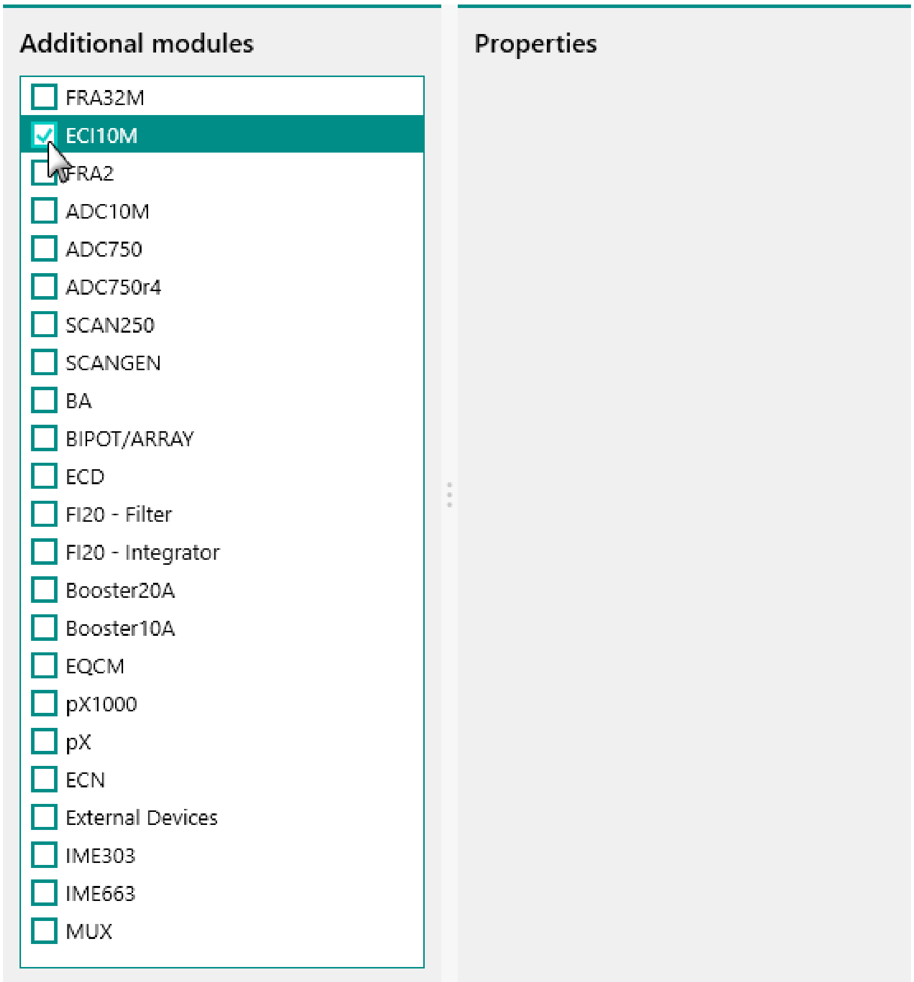


Figure 1174 The ECI10M module is selected in the hardware setup

16.3.2.8.4 ECI10M module settings

CAUTION

The ECI10M cannot be used at the same time as the Autolab PGSTAT. The ECI10M and the Autolab PGSTAT are both electrochemical interfaces to the electrochemical cell. To use the ECI10M module, it is necessary to switch the control of the electrochemical cell from the Autolab PGSTAT to the ECI10M.

CAUTION

The LCD display located on the front panel of the Autolab PGSTAT does not provide information when the ECI10M is used to control the electrochemical cell.

The EC10M module settings are completely defined in the NOVA software.

To use the ECI10M, it is first necessary to set the control of the cell to the ECI10M. This is done in the **Instrument** control panel *(see Chapter 5.2, page 85)*. When the ECI10M is installed in the Autolab, a drop-down list is provided, allowing the specification of the **Electrochemical interface**. The choice is provided between the Autolab main module and the ECI10M *(see Figure 1175, page 1037)*.

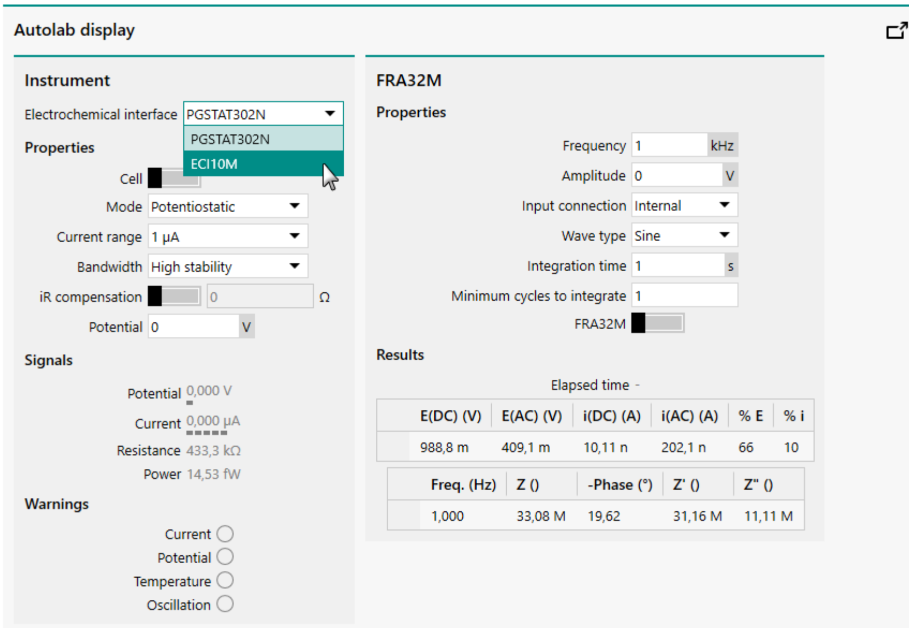


Figure 1175 The Electrochemical interface can be used to switch between Autolab PGSTAT and ECI10M control

This drop-down list can be used to specify the **Electrochemical interface**. When the ECI10M module is used, then the ECI10M will control the electrochemical cell. When the Autolab PGSTAT is used, the Autolab PGSTAT will control the electrochemical cell.

It is also possible to switch the **Electrochemical interface** directly in the **Instruments** panel of the **Dashboard**. Right-clicking an tile for an instrument fitted with the ECI10M module offers the possibility of toggling the **Electrochemical interface** to the ECI10M module or to the PGSTAT depending on the active interface *(see Figure 1176, page 1038)*.

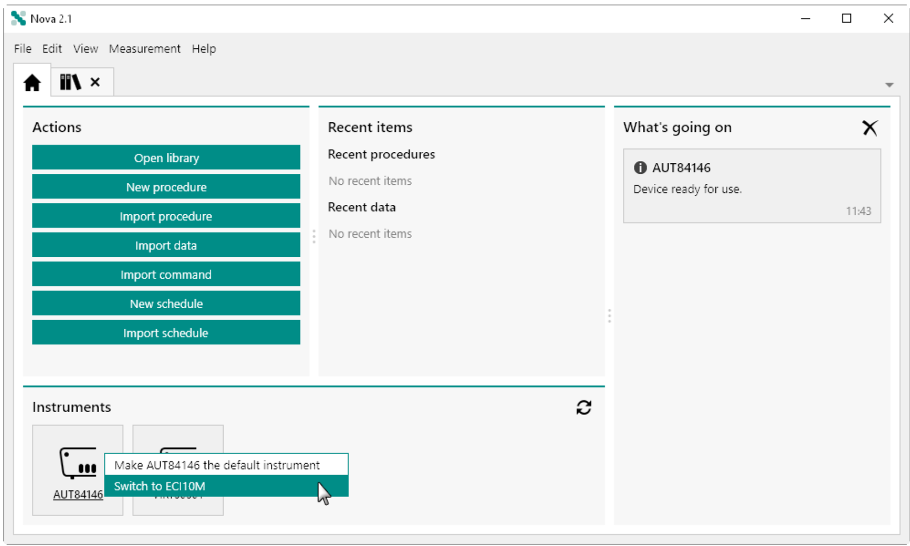


Figure 1176 The electrochemical interface can be directly selected through the Dashboard

CAUTION

Switching the **Electrochemical interface** from PGSTAT to ECI10M or the other way around using the provided drop-down list takes a couple of seconds. During this time, the Autolab system cannot be used.

At any time, a tooltip shows the active electrochemical interface, in bold *(see Figure 1177, page 1038)*.

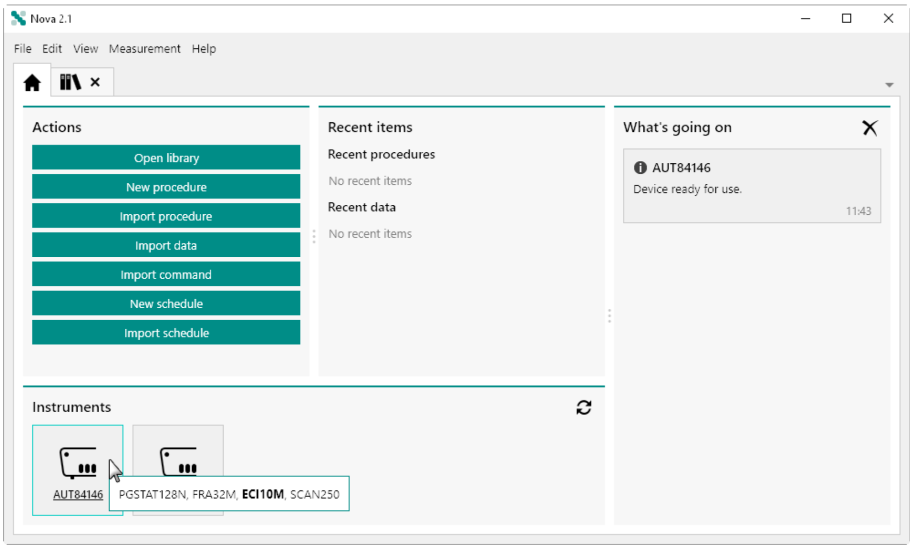


Figure 1177 A tooltip shows the active electrochemical interface in bold

Follow these steps to switch the control of the cell from the Autolab PGSTAT to the ECI10M module:

1. Make sure that the cell switch of the Autolab PGSTAT is in the **off** position.
2. Disconnect the Autolab PGSTAT cell connections (CE, RE, WE and S) from the electrochemical cell.
3. Switch the **Electrochemical interface** to **ECI10M**.
4. Connect the cables supplied with the ECI10M to the electrochemical cell.

Follow these steps to switch the control of the cell from the ECI10M module to the Autolab PGSTAT:

1. Make sure that the cell switch of the ECI10M module is in the **off** position.
2. Disconnect the cables supplied with the ECI10M from the electrochemical cell.
3. Switch the **Electrochemical interface** to **PGSTAT**.
4. Connect the Autolab PGSTAT cell connections (CE, RE, WE and S) to the electrochemical cell.

When the ECI10M module is used as the **Electrochemical interface**, the following user-definable settings are available, through the **Autolab control** command *(see Figure 1178, page 1040)*:

- **Cell:** a toggle that can be used to switch the cell of the ECI10M module on or off.
- **Mode:** specifies the mode of operation of the ECI10M module (potentiostatic, galvanostatic), using the provided drop-down list.
- **Current range:** specifies the active current range of the ECI10M module, using the provided drop-down list.
- **Bandwidth:** specifies the bandwidth of the ECI10M module (high stability, high speed, ultra high speed), using the provided drop-down list.
- **FRA32M input:** a toggle that can be used to switch the FRA32M input for the summation point of the ECI10M on or off.
- **Reference potential:** an input slider that can be used to set the reference potential of the ECI10M module. This value can be set in the range of ± 10 V.
- **Offset potential/current:** an input slider that can be used to set the offset potential or current of the ECI10M module. This value can be set in the range of ± 5 V (in potentiostatic mode) and in the range of ± 5 times the active current range (in galvanostatic mode).

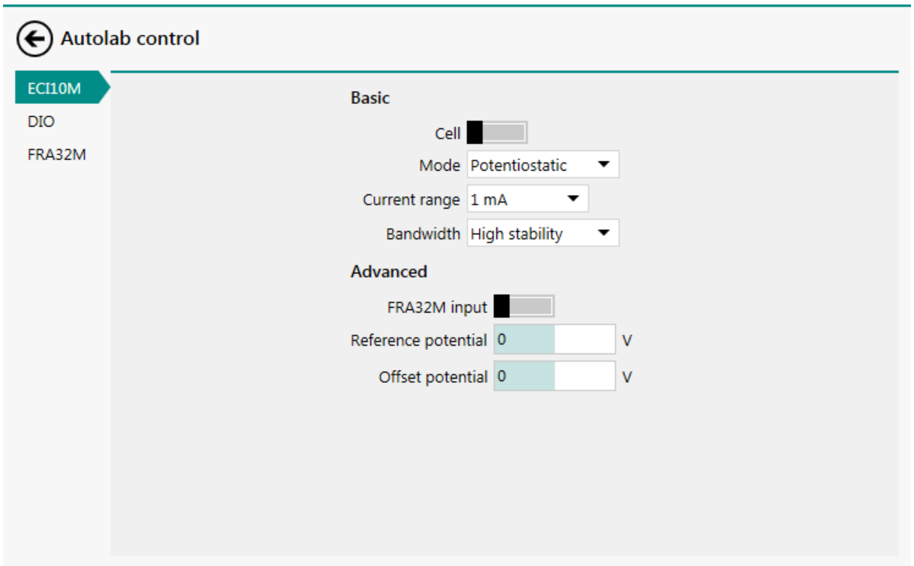


Figure 1178 The ECI10M module settings are defined in the Autolab control command

16.3.2.8.5 ECI10M bandwidth and contour map

The bandwidth for each current range of the **ECI10M** module are reported in *Table 43*.

Table 43 Bandwidth overview of the ECI10M module

Current range	Bandwidth
100 mA - 1 mA	10 MHz
100 µA	4 MHz
10 µA	150 kHz
1 µA	15 kHz
100 nA	1.5 kHz
10 nA	150 Hz

A typical contour map for the **ECI10M** module in combination with the **PGSTAT302N** potentiostat/galvanostat with **FRA32M** module is shown in *Figure 1179*.

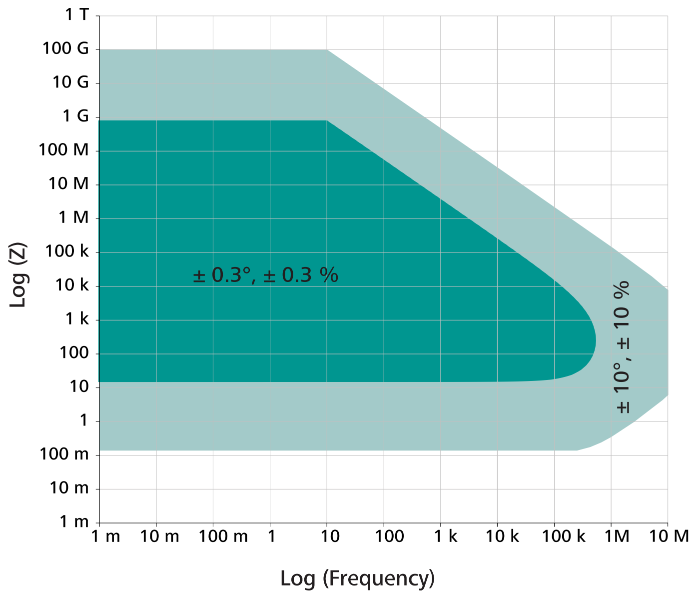


Figure 1179 Contour map of the ECI10M/PGSTAT302N/FRA32M combination

The map reported in *Figure 1179* shows a dark green area, which corresponds to the area of the map where an error of ± 0.3° on the measured phase angle and ± 0.3 % on the measured impedance value is expected. The light green area corresponds to the area of the map where an error of ± 10° on the measured phase angle and ± 10 % on the measured impedance value is expected.

NOTE

The contour map is determined empirically with an amplitude of 10 mV, in potentiostatic mode.

16.3.2.8.6 ECI10M module restrictions and precautions

Restrictions apply when using the ECI10M module:

- **FRA32M required:** in order to perform electrochemical impedance measurements up to 10 MHz using the ECI10M the **FRA32M** module must be installed in the instrument *(see Chapter 16.3.2.13, page 1091)*.
- **Concurrent use:** when the ECI10M is in use, the Autolab potentiostat/galvanostat is bypassed and cannot be used. The reverse applies to the ECI10M when the Autolab potentiostat/galvanostat is in use.
- **Module incompatibility:** while the ECI10M is in use, the additional internal **optional modules** installed in the Autolab cannot be used, with the exception of the FRA32M module.

Precautions apply when using the ECI10M module at high frequency:

- **Capacitive leakage:** for high frequency measurements with the ECI10M is it recommended to use unshielded cables to connect the ECI10M external interface to the electrochemical cell. Shielded cables have a capacitance of about 50 pF per meter. At very high frequency, the coaxial assembly of these cables will add a significant parallel capacitance to the impedance of the cell.
- **Cell proximity:** if possible, it is recommended to reduce the length of the cables between the ECI10M external interface and the electrochemical cell as much as possible. The default length of the cables supplied with the ECI10M is 25 cm and the device is optimized for this length of cable.

16.3.2.8.7 ECI10M front panel connection

The ECI10M module is fitted with a single female, DVI connector. A matching cable is used to connect to the external ECI10M external interface *(see Figure 1180, page 1042)*.

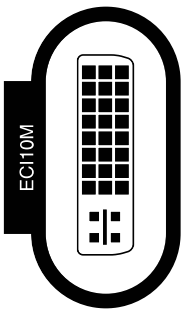


Figure 1180 The front panel labels of the ECI10M

The external interface of the ECI10M is fitted with a cable that directly connects to the front panel of the ECI10M module.

The front panel of the ECI10M external interface provides a number of connections and indicators, shown in *Figure 1181*.

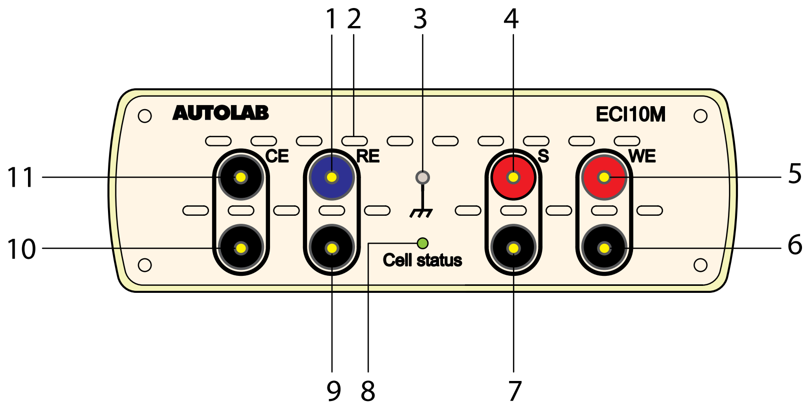


Figure 1181 Overview of the front panel of the ECI10M

1 RE connection (core)
4 mm banana connection for the Reference electrode (RE).

2 Air flow holes
For cooling the ECI10M during operation.

3 Ground plug
4 mm banana connector for grounding purposes.

4 S connection (core)
4 mm banana connection for the Sense electrode (S).

5 WE connection (core)
4 mm banana connection for the Working electrode (WE).

6 WE connection (shield)
4 mm banana connection for shield of the Working electrode (WE), if applicable.

7 S connection (shield)
4 mm banana connection for shield of the Sense electrode (S), if applicable.

8 Status LED
Used to indicate the status of the ECI10M (off, green or red).

9 RE connection (shield)
4 mm banana connection for shield of the Reference electrode (RE), if applicable.

10 CE connection (shield)
4 mm banana connection for shield of the Counter electrode (CE), if applicable.

11 CE connection (core)
4 mm banana connection for the Counter electrode (CE).

CAUTION

Never connect the electrode connectors from the PGSTAT instrument to the front panel of the ECI10M!

16.3.2.8.8 ECI10M module testing

NOVA is shipped with a procedure which can be used to verify that the **ECI10M** module is working as expected.

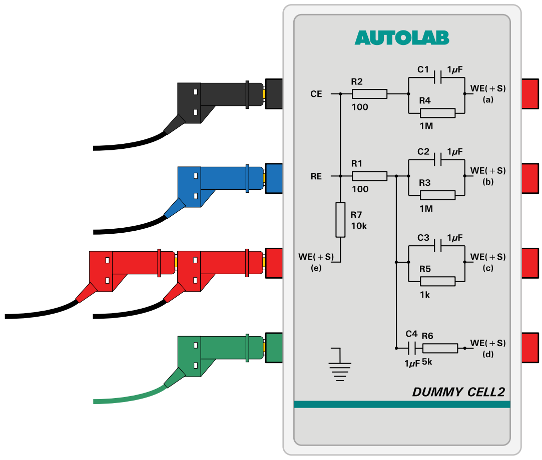
Follow the steps described below to run the test procedure.

1 **Load the procedure**

Load the **TestECI10M** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test\TestECI10M.nox)

2 **Connect the Autolab dummy cell**

Connect the **ECI10M** to the Autolab dummy cell circuit (e).



3 **Switch on the ECI10M**

Using the **Electrochemical interface** drop-down list provided in the **Instrument control** panel, switch the control of the electrochemical cell to the ECI10M *(see Figure 1182, page 1045)*.

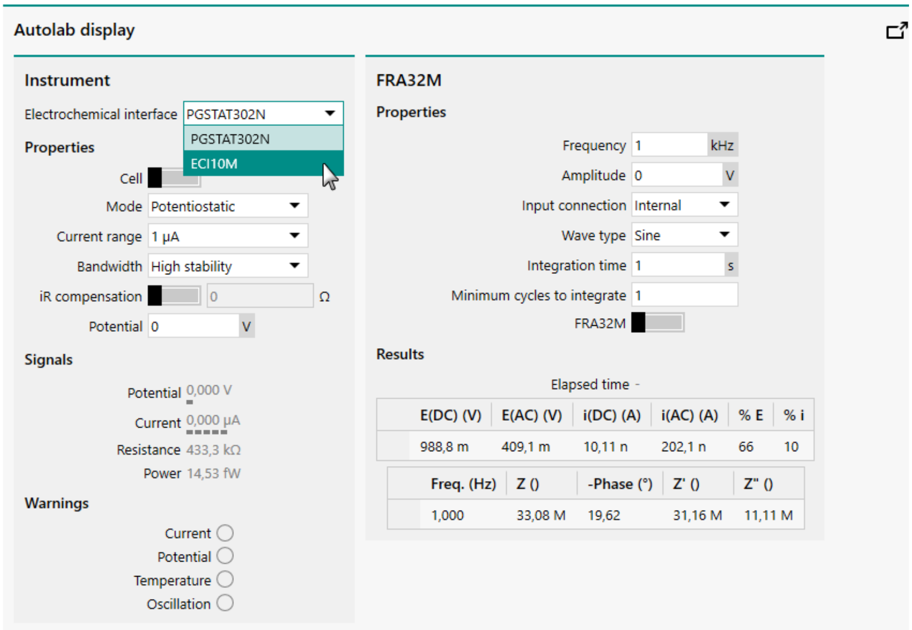


Figure 1182 Switch on the ECI10M module in the Instrument control panel

NOTE

Please allow the **ECI10M** module to warm up to 30 minutes.

4 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out an impedance spectroscopy measurement. During the measurement, the data is fitted using a (RC) equivalent circuit. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1183*.

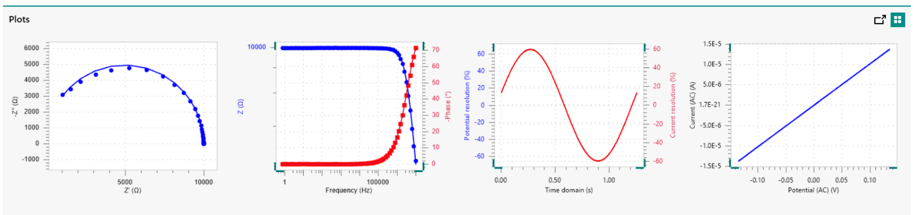
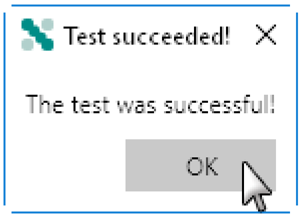


Figure 1183 The data measured by the TestECI10M procedure

5 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestECI10M automatic evaluation of the data requires the following tests to succeed:

1. The fitted resistance must be equal to 10000 Ω± 5 % .

This condition must be valid for the test to succeed.

16.3.2.8.9 ECI10M module specifications

The specifications of the ECI10M module are provided in *Table 44*.

Table 44 Specifications of the ECI10M

Specification	Value
Compliance voltage	± 10 V
Applied voltage range	± 10 V
Maximum current	± 100 mA
Current ranges	100 mA to 10 nA, in 8 decades
Electrode connections	2, 3 and 4
Frequency range	10 µHz - 10 MHz
Frequency resolution	0.003 %
Maximum AC amplitude	700 mV (RMS)
Bandwidth	15 MHz
Potential accuracy	± 0.2 %
Potential resolution	0.3 µV
Current accuracy	± 0.2 %
Current resolution	0.0003 % (of current range)
Input impedance	> 100 GΩ

16.3.2.9 ECN module

The ECN is an extension module for the Autolab PGSTAT. The ECN module provides the means to perform Electrochemical Noise (ECN) measurements. Electrochemical noise corresponds to seemingly random fluctuations in current and potential generated by stochastic phenomena occurring at the electrochemical interface.

The fluctuations of potential and current signals that arise directly from the electrochemical reactions, taking place on the electrode surface, can be measured using the optional ECN module. The ECN measurement is non-invasive because no external perturbation is applied and the current is monitored through a so-called Zero Resistance Ammeter (ZRA).

ECN can be used to monitor localized corrosion (pitting), uniform corrosion through measurement of the Noise Resistance, and the deterioration of paints on metal substrates. The same technique can also be used to monitor galvanic coupling in the presence of an electrolyte.

NOTE

Electrochemical noise measurements are also possible without the use of the dedicated ECN module. However, the resolution that can be achieved with the ECN is significantly better than without the use of this module.

The ECN module provides a dedicated differential amplifier, which can be used instead of the default differential amplifier provided by the Autolab. The ECN offers the possibility to carry out DC potential compensation and a four times additional amplification of the measured potential, leading to a maximum resolution of 760 nV.

The ECN module adds the following signal to the Sampler *(see Figure 1184, page 1047)*:

- **ECN(1).Potential (V):** this signal corresponds to the potential measured by the ECN module.

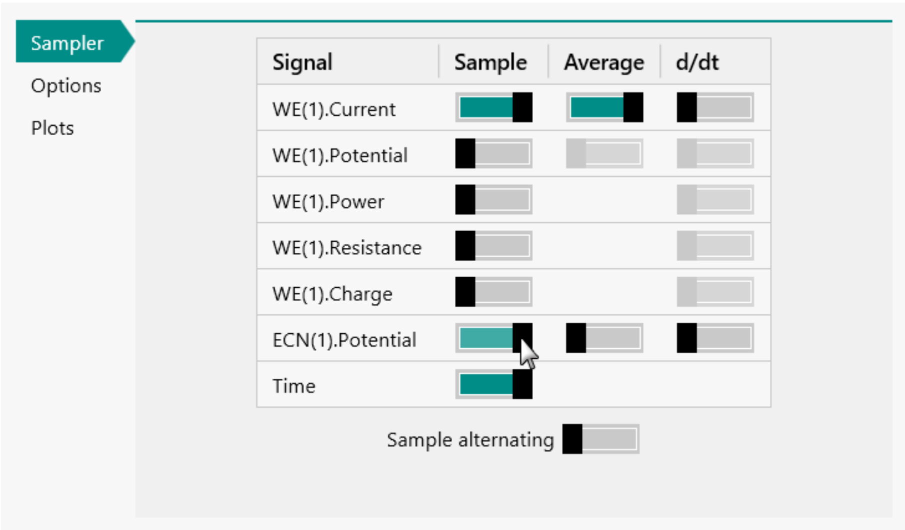


Figure 1184 The ECN module provides the ECN(1).Potential signal

16.3.2.9.1 ECN module compatibility

The ECN module is available for the following instruments:

- PGSTAT302N, PGSTAT302 and PGSTAT30
- PGSTAT128N and PGSTAT12
- PGSTAT10 and PGSTAT20

NOTE

The ECN module is **not** compatible with the Autolab instruments not listed above.

16.3.2.9.2 ECN module scope of delivery

The ECN module is supplied with the following items:

- ECN module
- ECN module label
- BNC to banana connection cable

16.3.2.9.3 ECN hardware setup

To use the **ECN** module, the hardware setup needs to be adjusted. The checkbox for the module needs to be ticked *(see Figure 1185, page 1049)*.

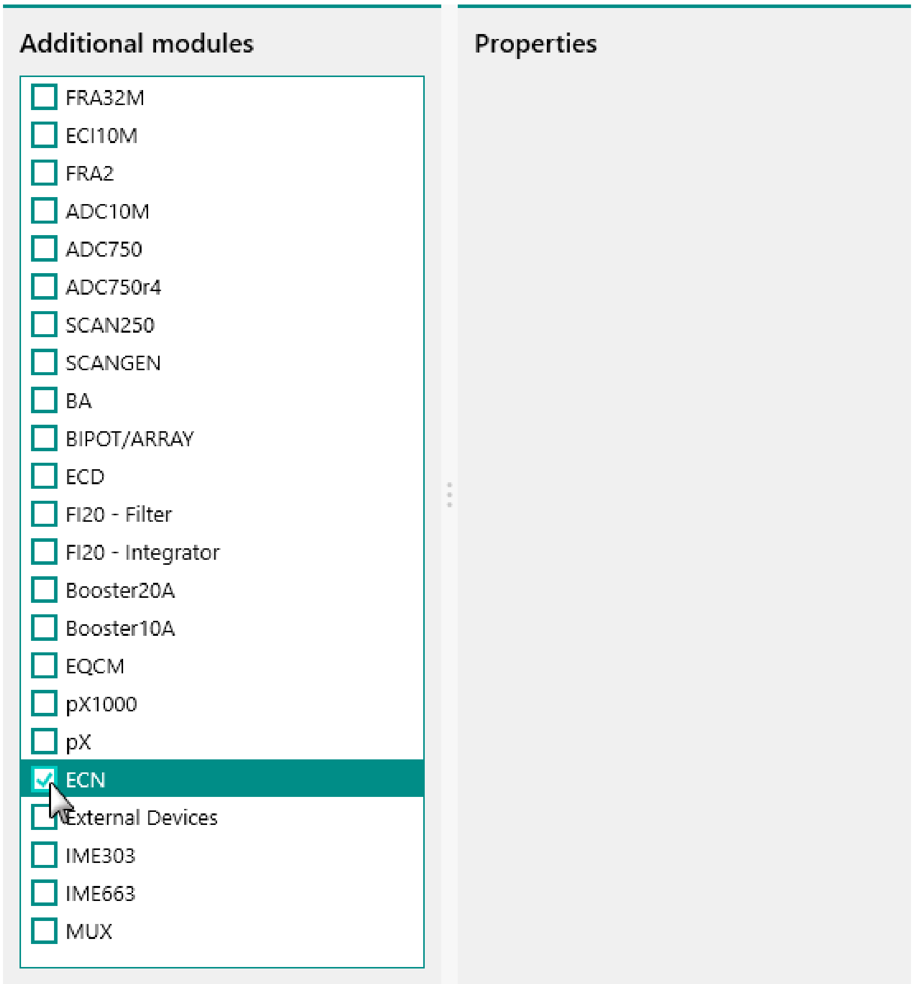


Figure 1185 The ECN module is selected in the hardware setup

16.3.2.9.4 ECN module settings

The ECN module settings are completely defined in the NOVA software. The following user-definable setting is available, through the **Autolab control** command *(see Figure 1186, page 1050)*:

- **Offset potential (V):** defines the offset compensation potential, in V. This offset is subtracted from the potential measured by the ECN module. The offset compensation can be specified in the range of ± 2.5 V.

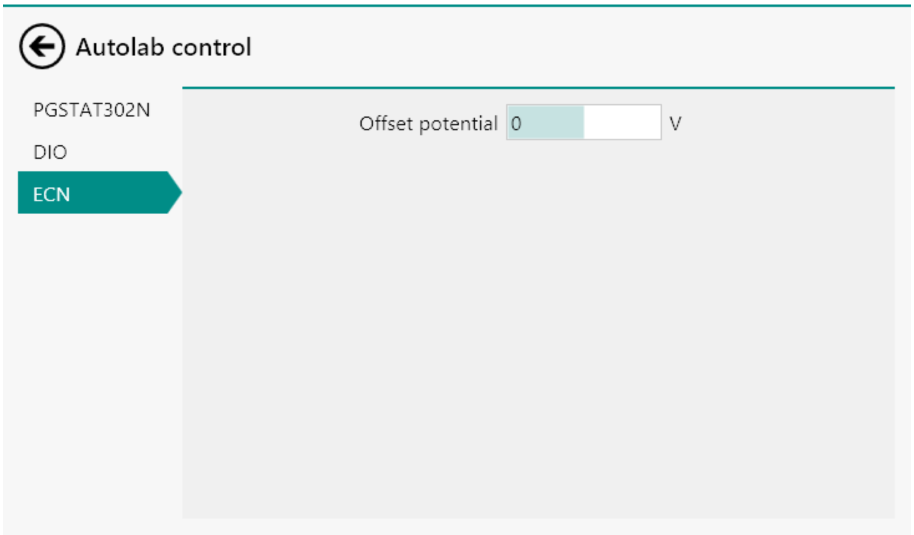


Figure 1186 The ECN module setting is defined in the Autolab control command

16.3.2.9.5 ECN module restrictions

Restrictions apply when using the ECN module during electrochemical noise measurements:

- **Current range restrictions:** during electrochemical noise measurements in combination with the ECN module, the current ranges of 1 mA and higher cannot be used. An **error** message is shown by NOVA when this situation is encountererd.
- **Cell switch:** electrochemical noise measurements are normally carried out at open circuit potential. Whenever the ECN(1).Potential signal is sampled, the cell should be switched off. NOVA provides a **warning** message when ECN measurements are carried out with the cell switched on.

16.3.2.9.6 ECN module front panel connections

The ECN module is fitted with two female BNC connector, labeled ⟵V and ⟶E, respectively *(see Figure 1187, page 1050)*.

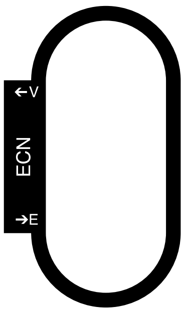


Figure 1187 The front panel label of the ECN module

The signal provided through the ⟵V connector on the front panel corresponds to the output of the voltage follower located on the ECN module.

The output signal is a voltage, referred to the instrument ground, corresponding to the converted potential according to:

$$E_{out}(\leftarrow V) = E_{ECN} \cdot 4$$

Where $E_{out}(\leftarrow V)$ corresponds to the output voltage of the module and E_{ECN} is the potential measured by the ECN amplified four times.

The →E input connector on the front panel is used to connect the ECN cable, supplied with the module.

NOTE

The front panel ←V BNC output is provided for information purposes only.

16.3.2.9.7 Connections for electrochemical noise measurements

In order to perform electrochemical noise measurements using the ECN module, the cable provided with the module must be connected to the →E input of the ECN module .

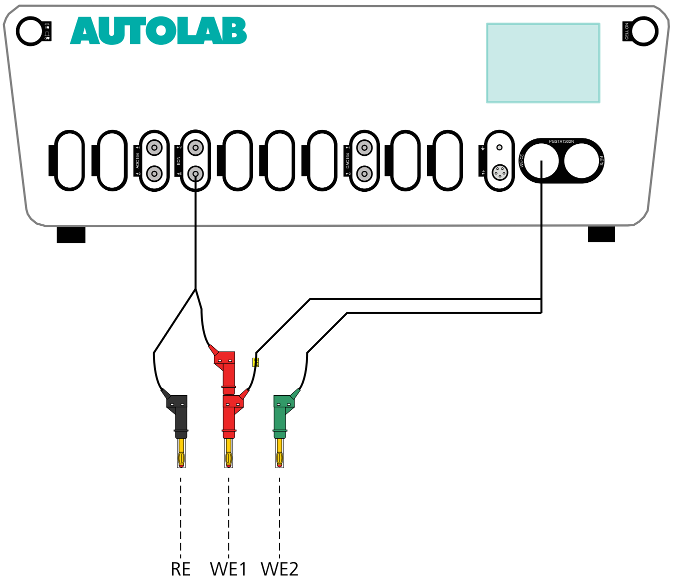


Figure 1188 Overview of the electrode connections for ECN measurements

The connections to the electrochemical cell should be carried out in the following way:

- The reference electrode (RE) and sense electrode (S) provided by the Autolab differential amplifier must be disconnected from the cell at all times.
- The counter electrode (CE) provided by the Autolab cell cable must be disconnected from the cell at all times.

- Connect the working electrode (WE) provided by the Autolab cell cable to electrode #1.
- Connect the green ground connector provided by the Autolab cell cable to electrode #2.
- Connect the red connector of the ECN cable to electrode #1.
- Connect the black connector of the ECN cable to the reference electrode (if present), or to electrode #2.

16.3.2.9.8 ECN module testing

NOVA is shipped with a procedure which can be used to verify that the **ECN** module is working as expected.

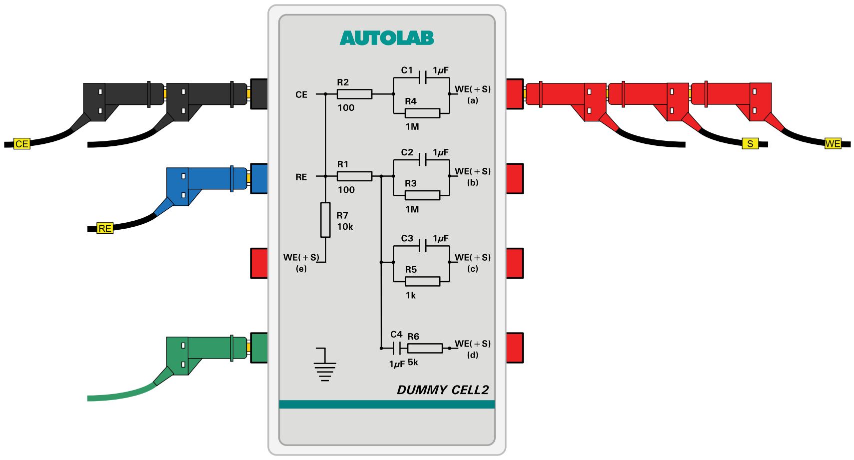
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestECN** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test \TestECN.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (a) and the ECN module to the Autolab dummy cell (a).



NOTE

Connect the cables from the ECN module to the dummy cell first.

3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1189*.

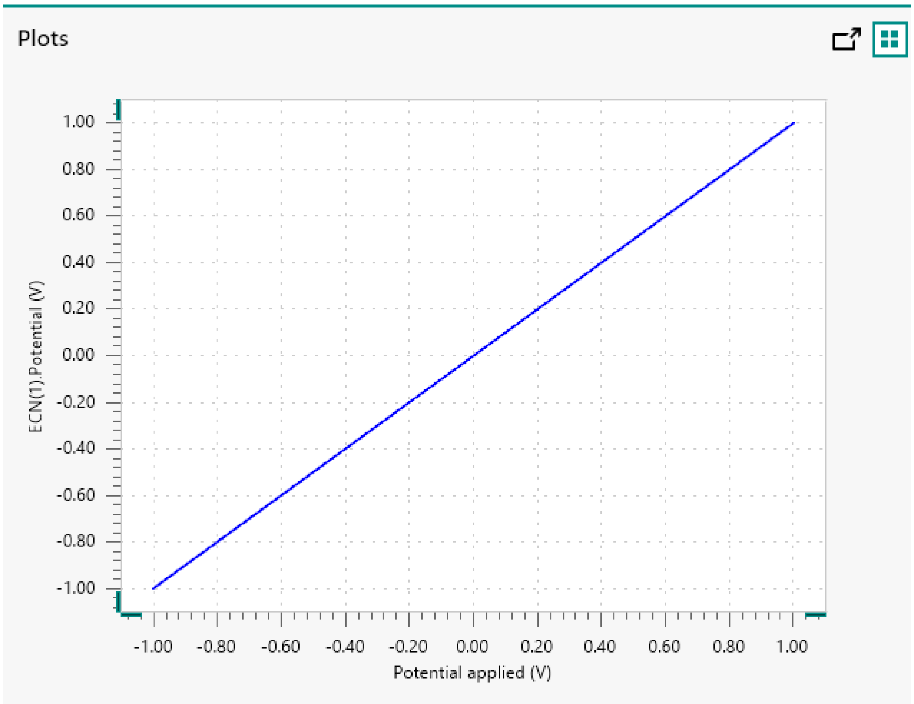
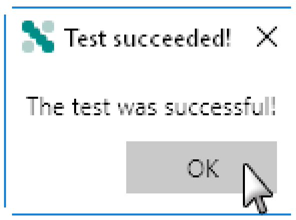


Figure 1189 The results of the TestECN procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestECN automatic evaluation of the data requires the following tests to succeed:

1. The inverted slope of the measured ECN(1).Potential versus the applied potential must be equal to 1 ± 5 %.
2. The intercept of the measured ECN(1).Potential versus the applied potential must be equal to ± 4.

Both conditions must be valid for the test to succeed.

16.3.2.9.9 ECN module specifications

The specifications of the ECN module are provided in *Table 45*.

Table 45 Specifications of the ECN module

Specification	Value
Input range	± 2.5 V
Maximum potential resolution	760 nV (Gain 100)
Potential offset compensation range	± 2.5 V
Potential accuracy	300 µV
Input impedance	100 GΩ
Input bias current	< 25 fA

16.3.2.10 EQCM module

The EQCM module is an extension module for the Autolab PGSTAT and the Multi Autolab. The EQCM module provides the means to perform Electrochemical Quartz Crystal Microbalance measurements.

The EQCM module measures a mass change per unit area by measuring the change in resonant frequency of a quartz crystal. Quartz crystals belong to a group of materials displaying the so-called piezoelectric effect. When a properly cut crystal (AT-cut) is exposed to an AC current, the crystal starts to oscillate at its resonant frequency and a standing shear wave is generated.

In first approximation, the resonant frequency depends on the thickness of the crystal. As mass is deposited on the surface of the crystal, the thickness increases; consequently the frequency of oscillation decreases from the initial value. With some simplifying assumptions, this frequency change can be quantified and correlated precisely to the mass change using Sauerbrey's equation:

$$\Delta f = -\frac{2f_0^2}{A\sqrt{\rho_q \mu_q}} \cdot \Delta m$$

Where Δf is the change in oscillating frequency, f_0 is the nominal resonant frequency of the crystal (6 MHz), Δm is the change in mass, in g/cm^2, A is the area of the crystal in cm^2, ρ_q is the density of quartz, in g/cm^3 and μ_q is the is the shear modulus of quartz, in g/cm·s^2.

For a 6 MHz crystal, the same equation can be reduced to:

$$-\Delta f = \Delta m \cdot C_f$$

Where Where C_f is 0.0815 Hz/ng/cm^2.

NOTE

For more information on the EQCM module please refer to the EQCM User Manual. More information on the validity and the application of the Sauerbrey equation can be found in the peer-reviewed literature.

The EQCM module adds the following signal to the Sampler *(see Figure 1190, page 1055)*:

- **EQCM(1).Temperature (°C):** this signal corresponds to the temperature recorded by the sensor located at the bottom of the EQCM cell.
- **EQCM(1).Driving force (V):** this value represents the amount of energy required to sustain the oscillation of the crystal. When the loading of the crystal increases, the driving force also increases. In air, the typical driving force is close to 0 V. In water, the driving force is about 0.85 V.
- **EQCM(1).ΔFrequency (Hz):** this signal corresponds to the relative change in oscillation frequency of the quartz crystal. This variation is expressed with respect to an arbitrary, user-defined reference frequency (zero Hz).

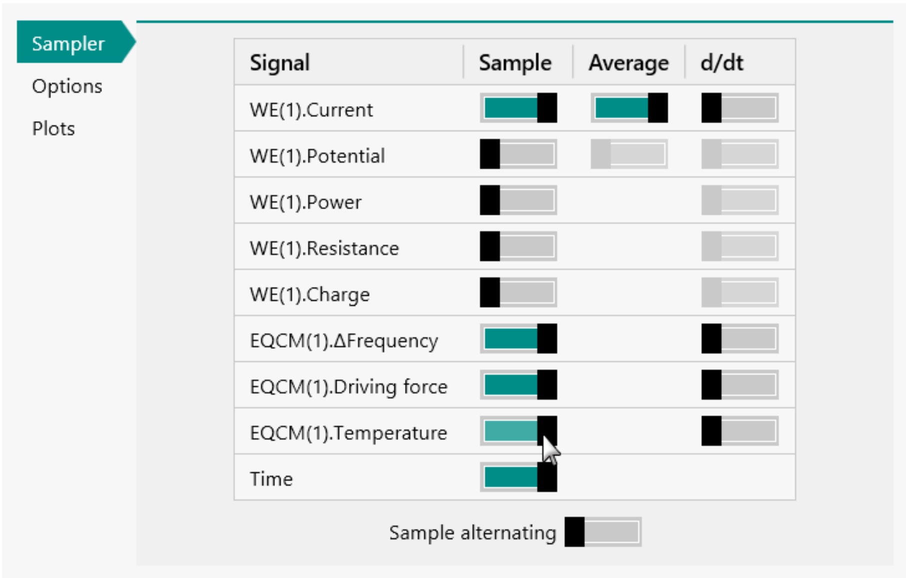


Figure 1190 The EQCM module provides the EQCM(1).ΔFrequency, EQCM(1).Driving force and EQCM(1).Temperature signals

16.3.2.10.1 EQCM module compatibility

The EQCM module is available for the following instruments:

- PGSTAT302N, PGSTAT302 and PGSTAT30
- PGSTAT128N and PGSTAT12
- PGSTAT100N and PGSTAT100
- M101
- PGSTAT204/M204

NOTE

The EQCM module is **not** compatible with the Autolab instruments not listed above.

16.3.2.10.2 EQCM module scope of delivery

The EQCM module is supplied with the following items:

- EQCM module
- EQCM module label
- EQCM oscillator
- EQCM connection cable
- EQCM cell
- Ag/AgCl reference electrode
- Au counter electrode
- 6 MHz, double-sided, Au coated quartz crystal (2)
- EQCM trimmer
- EQCM User Manual

16.3.2.10.3 EQCM hardware setup

To use the **EQCM** module, the hardware setup needs to be adjusted. The checkbox for the module needs to be ticked *(see Figure 1191, page 1057)*.

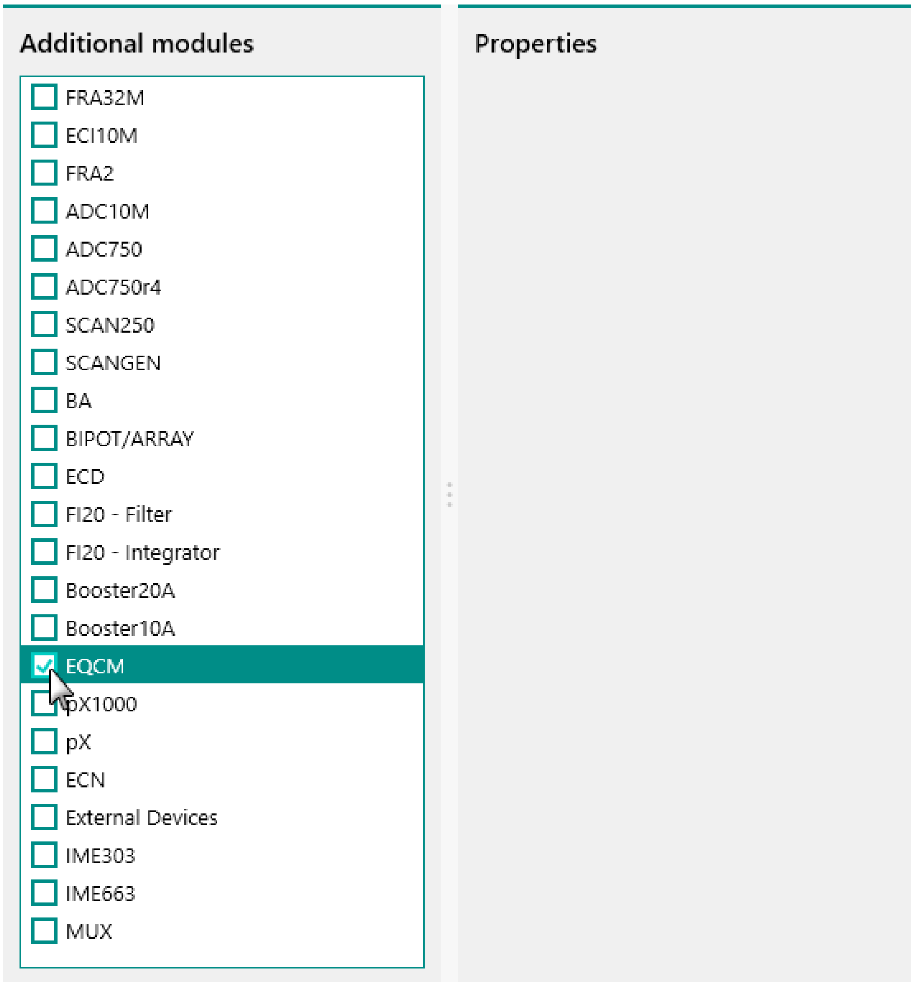


Figure 1191 The EQCM module is selected in the hardware setup

16.3.2.10.4 EQCM module settings

The EQCM module settings are completely defined in the NOVA software. The following user-definable setting is available, through the **Autolab control** command *(see Figure 1192, page 1058)*:

- **EQCM:** a toggle control that can be used to switch the EQCM module on or off.

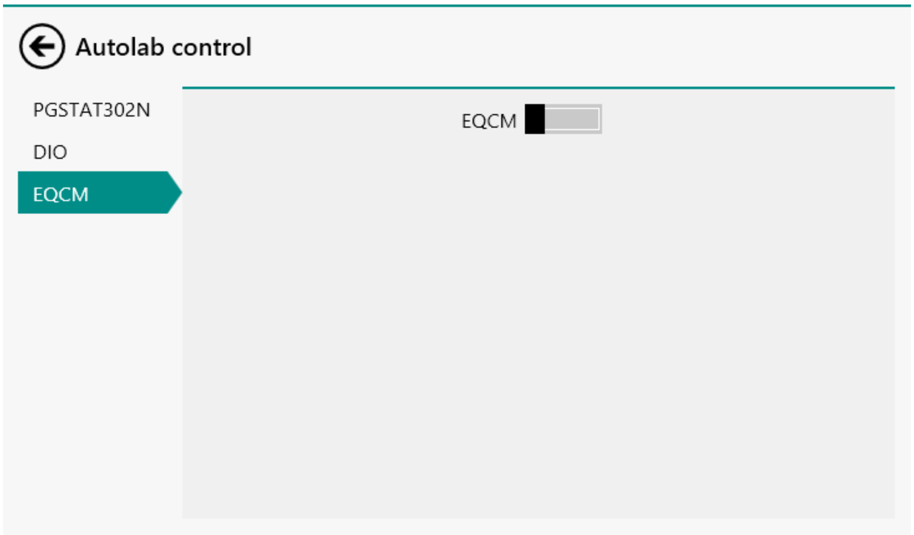


Figure 1192 The EQCM module setting is defined in the Autolab control command

NOTE

The EQCM oscillator will be powered as long as the EQCM module is on. It is recommended to allow the EQCM oscillator to warm up before starting electrochemical measurements and to keep it on when needed to avoid temperature changes.

16.3.2.10.5 EQCM module restrictions

Restrictions apply when using the EQCM module:

- **Sampling rate:** The EQCM module is capable of providing one new set of values for the measured signals (ΔFrequency, Driving force, Temperature) with an interval time of 20 ms (50 samples/s). When the sampling rate specified in the procedure is smaller than 20 ms, the EQCM module is not able to provide new data points quickly enough. In practice this means that last available data point provided by the EQCM module will be measured several times, until a new data point is available.
- **Dynamic frequency range:** The EQCM module uses a dynamic window of frequency of 1000 Hz. This ensures that the frequency is measured with the highest possible resolution. When changes in frequency larger than 1000 Hz are measured, the measurement window is adjusted downwards or upwards depending on the direction of the frequency change. This software rewindowing takes 100 ms. During this adjustment, the EQCM is not able to supply new data points and this can be seen in the measured data

NOTE

More information on the EQCM module is provided in the EQCM user manual.

16.3.2.10.6 EQCM front panel connection

The EQCM module is fitted with a single female, 9 pin SUB-D connector. A matching cable is used to connect to the external EQCM oscillator box *(see Figure 1193, page 1059)*.

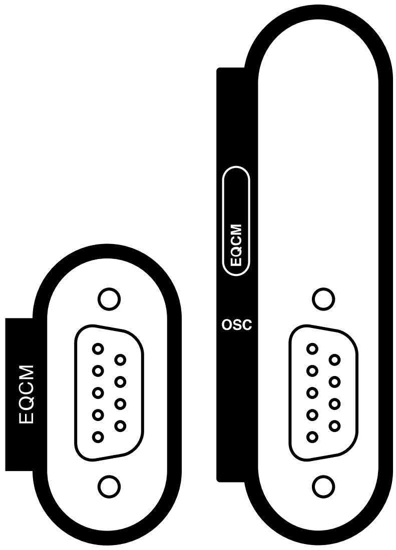


Figure 1193 The front panel labels of the EQCM module (left: EQCM module in PGSTAT, right: EQCM module in Multi Autolab)

16.3.2.10.7 EQCM module testing

NOVA is shipped with a procedure which can be used to verify that the **EQCM** module is working as expected.

CAUTION

This test is carried out using 2 ml of water.

Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestEQCM** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test\TestEQCM.nox)

2 Insert an EQCM crystal in the EQCM cell

insert a 6 MHz EQCM crystal in the EQCM cell. Fill the cell with 2 ml of water and check for leakage. Connect the cell to the EQCM oscillator and the oscillator to the Autolab PGSTAT using the provided cable. Leave the cell connectors from the PGSTAT disconnected.

> **NOTE**
>
> For more information on the EQCM hardware, please refer to the EQCM User Manual.

3 Start the procedure

Start the procedure and follow the instructions on-screen.

> **NOTE**
>
> Please allow the **EQCM** module to warm up to 15 minutes.

After 15 minutes, the test can be continued. The Determine EQCM zero frequency window will be displayed. Using the provided adjustment tool, rotate the trimmer on the EQCM oscillator in order to minimize the driving force and zero the ΔFrequency signal (as explained in the EQCM User Manual).

The test carries out a time resolved measurement measurement. The measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1194*.

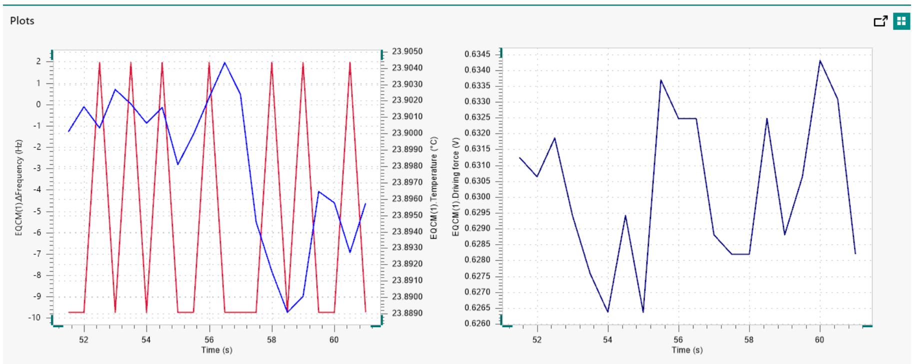
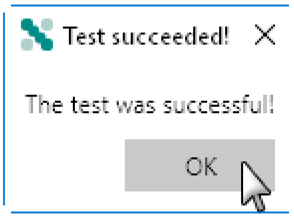


Figure 1194 The results of the TestEQCM procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestEQCM automatic evaluation of the data requires the following tests to succeed:

1. The EQCM(1).Driving force must stable within ± 0.025 V.
2. The average value of the EQCM(1).ΔFrequency must be larger or equal to -5 Hz and must be smaller or equal to 5 Hz.

Both condition must be valid for the test to succeed.

16.3.2.10.8 EQCM module specifications

The specifications of the EQCM module are provided in *Table 46*.

Table 46 Specifications of the EQCM module

Specification	Value
Oscillation frequency	6 MHz
Frequency resolution	0.07 Hz
Relative accuracy	1 Hz
Absolute accuracy	10 Hz
Frequency range	80 kHz
Temperature sensor accuracy	1 °C
Temperature sensor resolution	0.1 °C

16.3.2.11 FI20 module

The FI20 module is an extension module for the Autolab potentiostat/galvanostat. This module provides two electronic circuits that can be used to process the current measured by the instrument. Each of these circuits fulfills a specific role:

- **Filter circuit:** the filter circuit is designed to filter the current during electrochemical measurements. The filter can be used to remove noise on the measurements in cases where it is impossible to remove the noise by the use of proper shielding of the cell and electrodes or by using a Faraday cage. The module uses a third order Sallen-Key filter, with three different filter time constants (0.1 s, 1 s and 5 s).
- **Integrator circuit:** the integrator circuit provides the means to integrate the measured current. The integrator can be used to perform chronocoulometric experiments and the so-called cyclic or linear sweep voltammetry current integration. The module consists of an analog integrator fitted with four different integration time constants (0.01 s, 0.1 s, 1 s, 10 s).

NOTE

The **integrator** circuit is present by default (as on-board integrator) on the following instruments: PGSTAT10, PGSTAT20, µAutolab II, µAutolab III, PGSTAT101, M101, PGSTAT204 and M204.

CAUTION

The FI20 and the on-board integrator can only be used to process the current measured by the Autolab potentiostat/galvanostat (WE(1).Current).

The FI20 module and the on-board integrator add the following signal to the Sampler *(see Figure 1195, page 1063)*:

- **Integrator(1).Charge (C):** this signal corresponds to the measured charge.
- **Integrator(1).Integrated current (A):** this signal corresponds to the converted equivalent current obtained by deriving the measured charge over the interval time used in the measurement. This signal can be used in order to perform so-called current integration cyclic and linear sweep voltammetry measurements. In first approximation, at low scan rates, the results obtained with the current integration method can be compared to the results obtained with a linear scan generator.

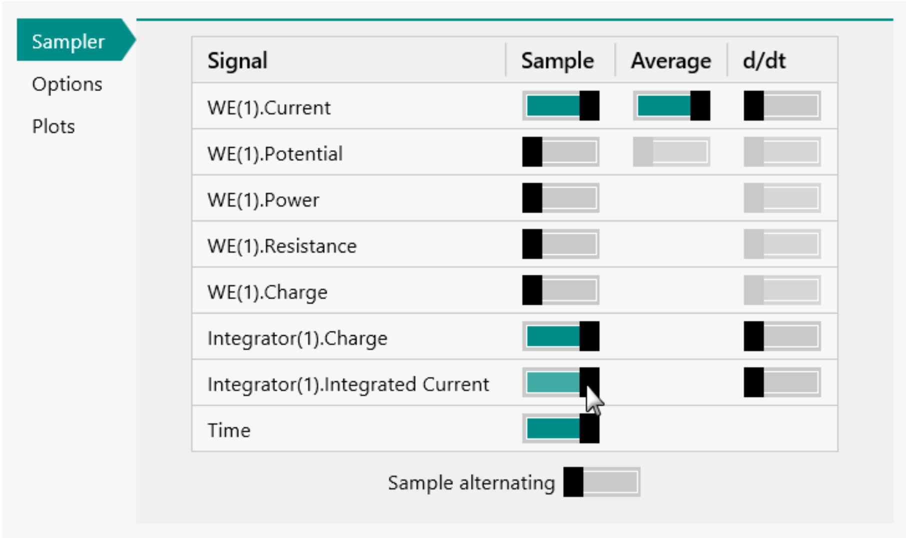


Figure 1195 The FI20 module and on-board integrator provide the Integrator(1).Charge and Integrator(1).Integrated Current signals

NOTE

The difference between the Integrator(1).Charge and the WE(1).Charge signal available in the Sampler is that the latter is obtained from the mathematical integration of the WE(1).Current while the Integrator(1).Charge is determined by analog integration of the current.

NOTE

The Integrator(1).Integrated current (A) signal is only available for the **CV staircase** and **LSV staircase** commands.

16.3.2.11.1 FI20 module compatibility

The FI20 module is available for the following instruments:

- PGSTAT302N, PGSTAT302 and PGSTAT30
- PGSTAT128N and PGSTAT12
- PGSTAT100N and PGSTAT100
- PGSTAT20
- PGSTAT10

NOTE

The FI20 module is **not** compatible with the Autolab instruments not listed above.

The on-board integrator is fitted in the following instruments:

- µAutolab II and µAutolab III
- PGSTAT101 and M101
- PGSTAT204 and M204

16.3.2.11.2 FI20 module scope of delivery

- FI20 module
- FI20 module label

16.3.2.11.3 FI20 hardware setup

To use the **FI20** module, the hardware setup needs to be adjusted. The checkbox for the module needs to be ticked *(see Figure 1196, page 1064)*.

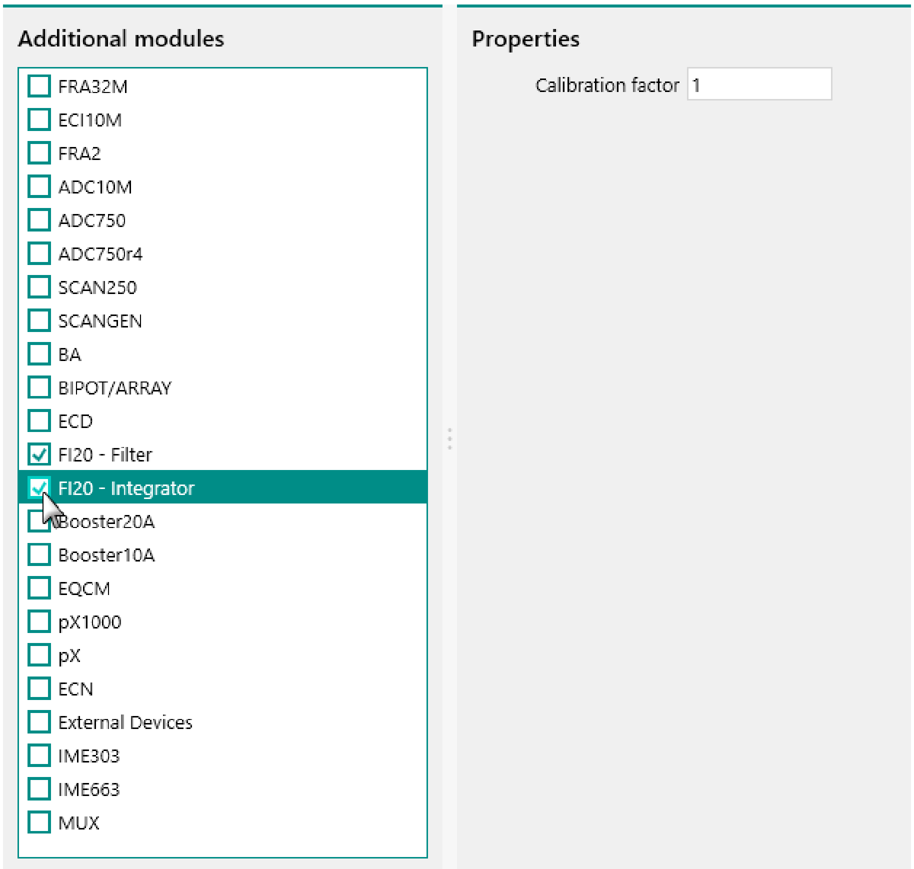


Figure 1196 The FI20 module is selected in the hardware setup

Two checkboxes are provided for instruments that can accommodate the **FI20** module. The **Properties** panel shows the *Calibration factor* value for the **FI20 - Integrator** module when it is selected in the **Additional modules** panel, as shown in *Figure 1196*. This value can be adjusted, if necessary.

For intruments that are fitted with the on-board integrator module, the hardware setup is adjusted correctly by default *(see Figure 1197, page 1065)*.

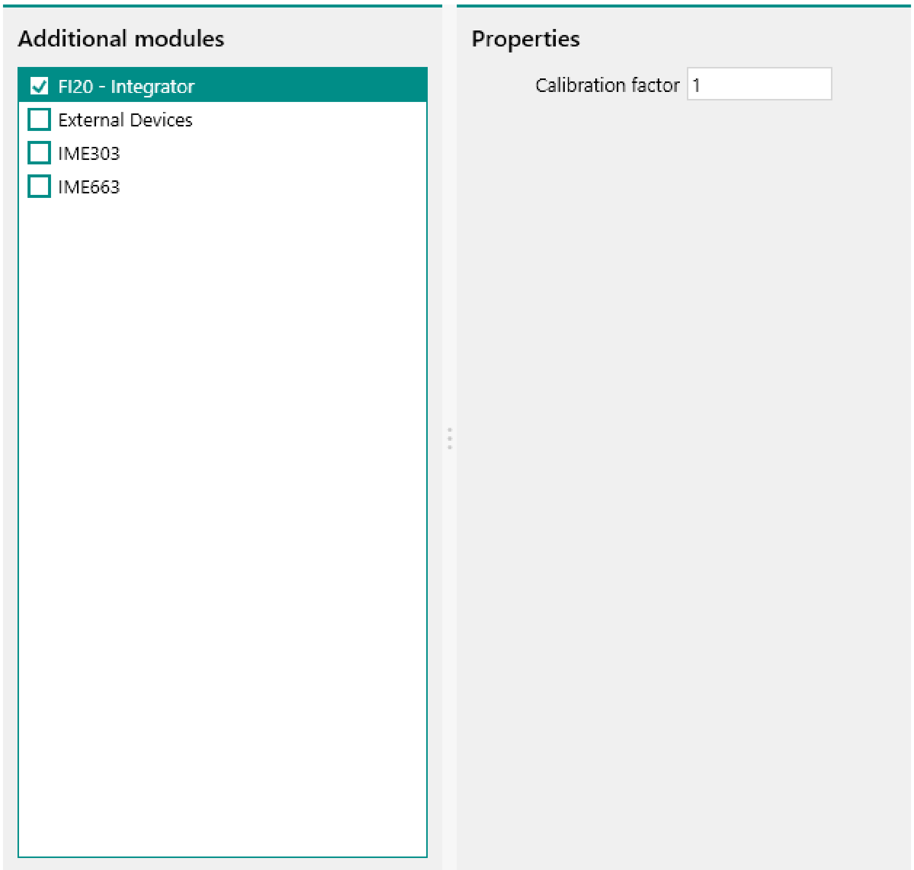


Figure 1197 The on-board integrator is automatically selected for applicable instruments

Similarly to the properties of the **FI20** module, the *Calibration factor* value can be adjusted if necessary in the **Properties** panel *(see Figure 1197, page 1065)*.

NOTE

The *Calibration factor* is determined during the **Integrator test** of the **Diagnostics** application. Please refer to *Chapter 17.3* for more information.

16.3.2.11.4 FI20 module and on-board integrator settings

The settings of the FI20 module and the on-board integrator are completely defined in the NOVA software.

The following user-definable settings are available for the **filter** circuit, the **integrator** circuit of the FI20 module and the on-board integrator, through the **Autolab control** command *(see Figure 1198, page 1066)*:

- In the **Filter** sub-panel:
  - **Filter time:** a drop-down control that can be used to select the time constant for the filter circuit. The filter time can be set to off, 100 ms, 1 s and 5 s.
- In the **Integrator** sub-panel:
  - **Integration time:** a drop-down control that can be used to select the time constant for the integrator circuit. The integration time can be set to 10 ms, 100 ms, 1 s, 10 s and **hold**.
  - **Discharge integrator:** a toggle that can be used to discharge the integrator.
  - **Integrator drift:** a value field which can be used to manually set the integrator drift, in C/s.

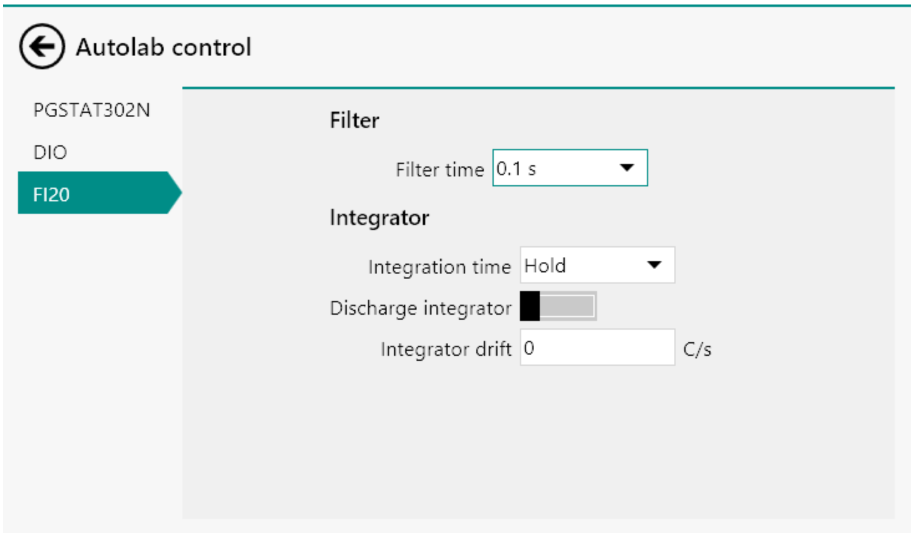


Figure 1198 The settings of the filter circuit and the integrator circuit of the FI20 module and the on-board integrator are defined in the Autolab control command

The **Filter time** setting defines the time constant of the filter. The effect of a filter constant of x s is that x seconds after a potential perturbation has been applied, the current response can be measured correctly. A high time constant results in heavy filtering.

NOTE

The effective filter time constant corresponds to ten times the time constants provided in the Autolab control command, resulting in 99.995 % attenuation

The **hold** setting, available from the Integration time drop down list is a special mode that can be used to isolate the integrator at any time during a measurement. When this mode is selected, the integrator is disconnected from the working electrode and the accumulated charge is stored. The integrator is not discharged when the Hold mode is active. The integrator can be reconnected to the working electrode at any time by selecting any of the available integration time values from the drop down list.

The integration time constant should be selected carefully, taking the current range into account. If the current range or the integration time constant is too low, the integrator can be saturated, leading to a wrong charge value. The saturation is reached when:

$$Q_{sat} = 10([CR] \cdot \tau_{int})$$

Where Q_{sat} is the saturation charge, [CR] is the active current range and τ_{int} is the active integrator time constant.

NOTE

When the measured charge exceeds the maximum saturation capacity of the integrator, it is possible to use the mathematical integration of the current instead, by sampling the WE(1).Charge signal using the Sampler.

The **Discharge integrator** setting can be used to fully discharge the integrator. This can be done at any time during a measurement, independently of the activity of the working electrode. If the integrator is not discharged during long measurements, there is a saturation risk.

NOTE

It is highly recommended to set the Discharge integrator property to On at the end of each measurement involving the **integrator** circuit of the FI20 module or the on-board integrator. Leaving the integrator charged could introduce extra noise during consequent measurements.

The **Integrator drift** can be manually specified. The drift is defined as the charge accumulation due to leakage current, in C/s. Setting the drift allows to compensate for the non-ideality of the Autolab. This value can be set manually or it can be determined automatically using the provided drift determination tool, available in the instrument control panel *(see Chapter 5.2.2.6, page 115)*.

NOTE

The drift depends on the active current range. It is recommended to determine the drift whenever the current range is modified.

16.3.2.11.5 FI20 module restrictions

Restrictions apply when using the FI20 module or the on-board integrator.

The following restrictions apply when using the **filter** circuit of the FI20 module:

- **Automatic current ranging:** the Automatic Current Ranging option cannot be used with the **filter** circuit of the FI20 module is used. An **error** message is provided by NOVA when this situation is encountered.
- **Bandwidth limit:** the **filter** circuit of the FI20 module has a limited bandwidth. Depending on the interval time, a **warning** can be provided by NOVA indicating that the selected filter time constant is too slow to measure properly.

The following restrictions apply when using the **integrator** circuit of the FI20 module or the on-board integrator:

- **Automatic current ranging:** the Automatic Current Ranging option cannot be used with the **integrator** circuit of the FI20 module or the on-board integrator is used. An **error** message is provided by NOVA when this situation is encountered.

- **Charge saturation:** the **integrator** circuit of the FI20 module or the on-board integrator has a limited charge capacity. The integration time constant should be selected carefully, taking the current range into account. If the current range or the integration time constant is too low, the integrator circuit can be saturated, leading to a wrong charge value.

16.3.2.11.6 FI20 module front panel connections

The FI20 module is fitted with two female BNC connectors, labeled ←I and ←Q *(see Figure 1199, page 1069)*.

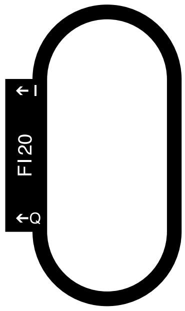


Figure 1199 The front panel labels of the FI20 module

The signal provided through the ←I connector on the front panel corresponds to the filtered current provided by the filter circuit of the FI20 module. This corresponds to the current measured by the Autolab potentiostat/galvanostat after processing by the filter. The output signal is a voltage, referred to the instrument ground, corresponding to the converted current according to:

$$E_{out}(\leftarrow I) = \frac{i_{(FI20)}}{[CR]}$$

Where $E_{out}(\leftarrow I)$ corresponds to the output voltage signal of the module, in V, $i_{(FI20)}$ corresponds to the filtered current, in A and [CR] is the active current range of the Autolab potentiostat/galvanostat module.

The signal provided through the ←Q connector on the front panel corresponds to the integrated current provided by the integrator circuit of the FI20 module. The output signal is a voltage, referred to the instrument ground, corresponding to the converted current according to:

$$E_{out}(\leftarrow Q) = \frac{Q_{(FI20)}}{[CR]}$$

Where $E_{out}(\leftarrow Q)$ corresponds to the output voltage signal of the module, in V, $Q_{(FI20)}$ corresponds to the charge, in C and [CR] is the active current range of the Autolab potentiostat/galvanostat module.

16.3.2.11.7 FI20 module and on-board integrator module testing

Three test procedures are provided for testing the FI20 module and the on-board integrator:

- For the Filter circuit of the FI20 module, please refer to *Chapter 16.3.2.11.7.1*.
- For the Integrator circuit of the FI20 module and the on-board integrator of all instrument equipped with an on-board integrator except the PGSTAT101, please refer to *Chapter 16.3.2.11.7.2*.
- For the on-board integrator of the PGSTAT101, please refer to *Chapter 16.3.2.11.7.3*.

16.3.2.11.7.1 FI20 module filter test

NOVA is shipped with a procedure which can be used to verify that the filter circuit of the **FI20** module is working as expected.

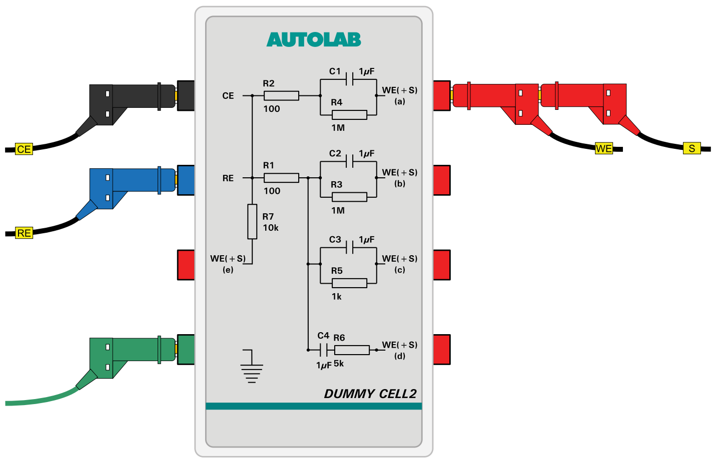
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestFI20-Filter** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases \Module test\TestFI20-Filter.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (a).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1200*.

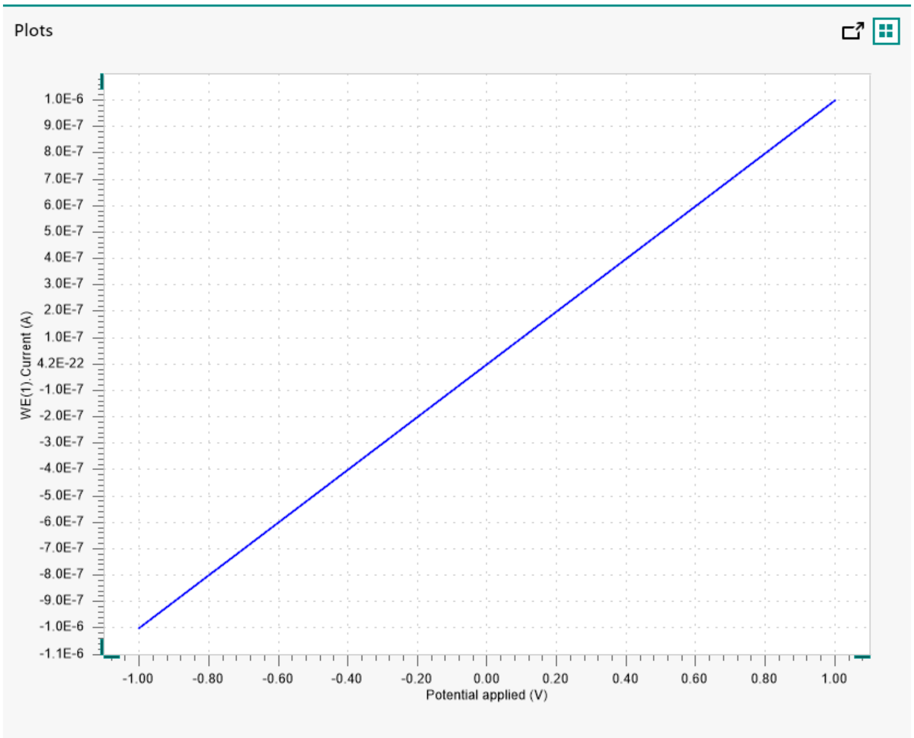
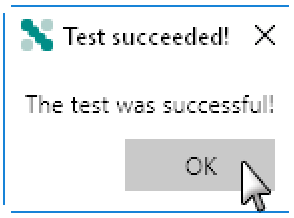


Figure 1200 The results of the TestFI20-Filter procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestFI20-Filter automatic evaluation of the data requires the following tests to succeed:

1. The inverted slope of the measured current versus the applied potential must be equal to 1000100 Ω± 5 %.
2. The intercept of the measured current versus the applied potential must be equal to ± 4 mV divided by 1000100 Ω± 5 %.

Both conditions must be valid for the test to succeed.

16.3.2.11.7.2 FI20 module integrator test

NOVA is shipped with a procedure which can be used to verify that the integrator circuit of the **FI20** module and the on-board integrator of all equipped instruments, except the Autolab PGSTAT101, is working as expected.

NOTE

To test the on-board integrator of the PGSTAT101, a dedicated test is provided. Please refer to *Chapter 16.3.2.11.7.3* for more information.

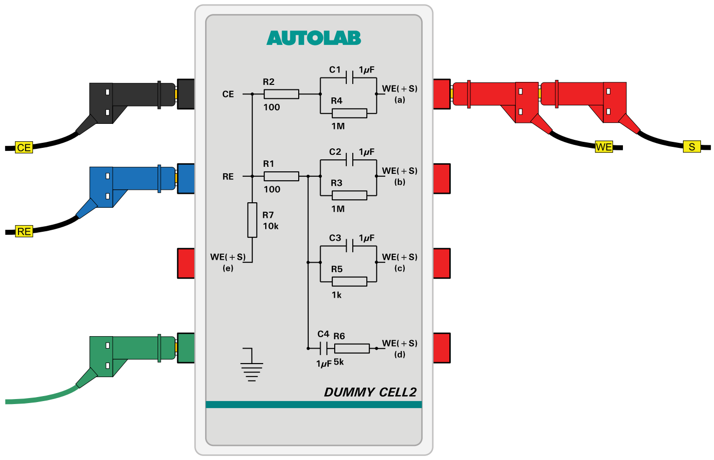
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestFI20-Integrator** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test\TestFI20-Integrator.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (a).



3 Reset the integrator drift

Open the instrument control panel, set the current range to 10 µA and reset the integrator drift using the provided button *(see Figure 1201, page 1073)*.

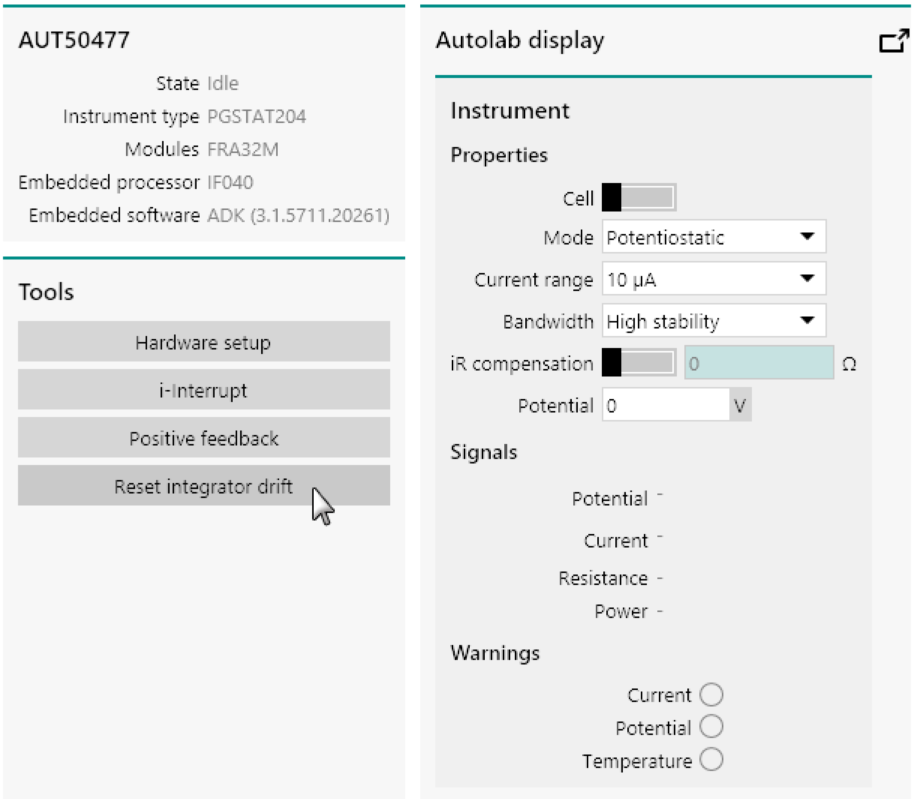


Figure 1201 Resetting the integrator drift

4 **Start the procedure**

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *(see Figure 1202, page 1073)*.

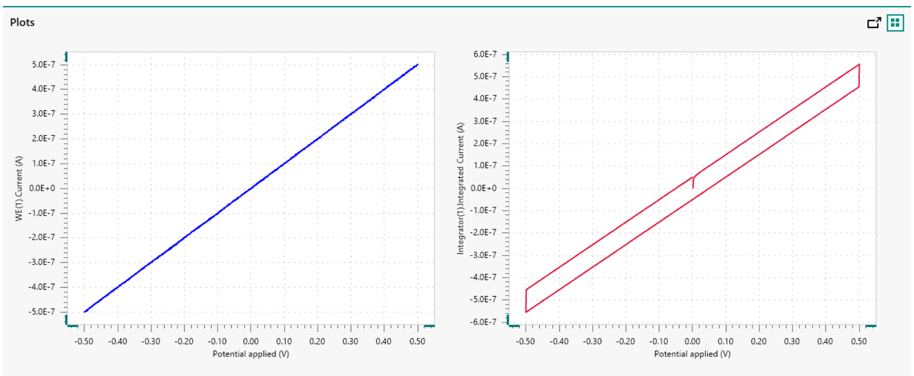
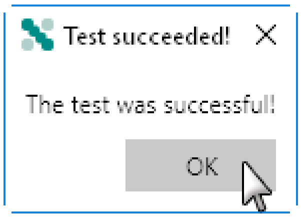


Figure 1202 The results of the TestFI20-Integrator procedure

5 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestFI20-Integrator automatic evaluation of the data requires the following tests to succeed:

1. The inverted slope of the measured current versus the applied potential must be equal to 1000100 Ω± 5 %.
2. The calculated capacitance, determined from the integrated current signal, must be equal to 1 µF ± 10 %.

Both conditions must be valid for the test to succeed.

16.3.2.11.7.3 On-board integrator test (PGSTAT101 only)

NOVA is shipped with a procedure which can be used to verify that the the on-board integrator of the Autolab PGSTAT101 is working as expected.

CAUTION

This test is only suitable for the Autolab PGSTAT101! For the other instruments, please refer to *Chapter 16.3.2.11.7.2*.

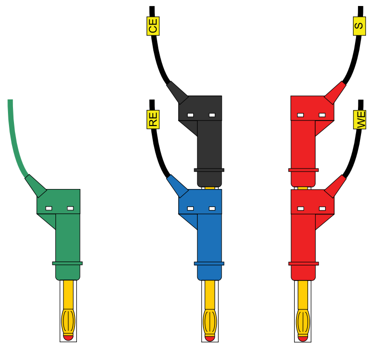
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestFI20-Integrator-PGSTAT101** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X \SharedDatabases\Module test\TestFI20-Integrator-PGSTAT101.nox)

2 Connect the electrode cables

Connect the counter electrode (CE) and reference electrode (RE) together and the working electrode (WE) and sense electrode (S) together.



3 Reset the integrator drift

Open the instrument control panel, set the current range to 10 µA and reset the integrator drift using the provided button *(see Figure 1203, page 1075)*.

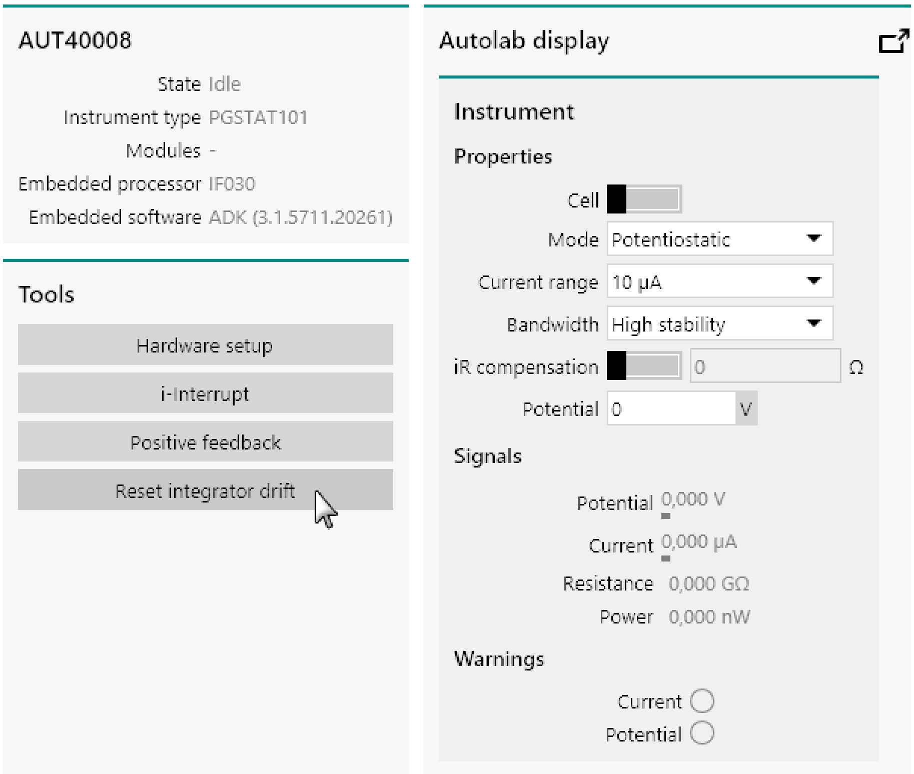
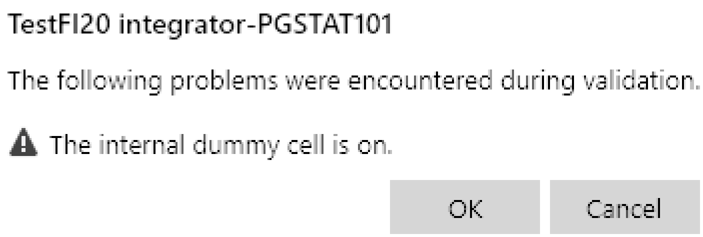


Figure 1203 Resetting the integrator drift

4 Ignore the warning

Ignore the warning message shown when the procedure is loaded and when the procedure is started.



5 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1204*.

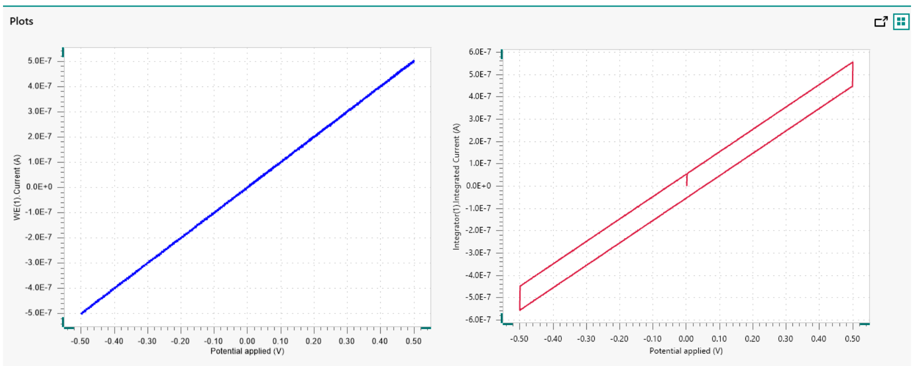
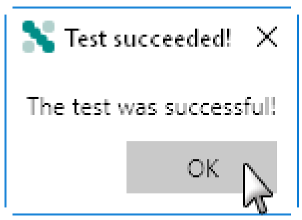


Figure 1204 The results of the TestFI20-Integrator-PGSTAT101 procedure

6 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestFI20-Integrator-PGSTAT101 automatic evaluation of the data requires the following tests to succeed:

1. The inverted slope of the measured current versus the applied potential must be equal to 1000100 Ω± 5 %.
2. The calculated capacitance, determined from the integrated current signal, must be equal to 1 µF ± 10 %.

Both conditions must be valid for the test to succeed.

16.3.2.11.8 FI20 module specifications

The specifications of the **filter** circuit of the FI20 module are provided in *Table 47*.

Table 47 Specifications of the filter circuit of the FI20 module

Specification	Value
Type of filter	3rd order Sallen-Key
Filter time constants	10 ms, 100 ms, 500 ms
Output offset	± 2 mV

The specifications of the **integrator** circuit of the FI20 module and the on-board integrator are provided in *Table 48*.

Table 48 Specifications of the integrator circuit of the FI20 module and the on-board integrator

Specification	Value
Integration time constants	10 ms, 100 ms, 1 s, 10 s
Charge measurement accuracy	0.2 %
Temperature dependence	< 0.04 %/K

16.3.2.12 FRA2 module

The FRA2 module is an extension module for the Autolab PGSTAT and the µAutolabIII. This module consists of function generator and a transfer function analyzer. The function generator can be used to generate a sine wave based signal and analyze the transfer function between two sine wave based signals. Instruments fitted with this module can perform electrochemical impedance spectroscopy (EIS) measurements.

NOTE

The FRA2 module is no longer available and it is now replaced by its successor module, the FRA32M.

16.3.2.12.1 FRA2 module compatibility

The FRA2 module is available for the following instruments:

- PGSTAT302N, PGSTAT302 and PGSTAT30
- PGSTAT128N and PGSTAT12
- PGSTAT100N and PGSTAT100
- PGSTAT20
- PGSTAT10

- μAutolab III

> **NOTE**
>
> The FRA2 module is **not** compatible with the Autolab instruments not listed above.

16.3.2.12.2 FRA2 module scope of delivery

The FRA2 module is supplied with the following items:

- FRA2 module
- FRA2 module labels

16.3.2.12.3 FRA2 hardware setup

To use the **FRA2** module, the hardware setup needs to be adjusted. The checkbox for the module needs to be ticked. The **FRA2** module also has a number of additional properties that need to be specified correctly in the **Properties** panel *(see Figure 1205, page 1079)*.

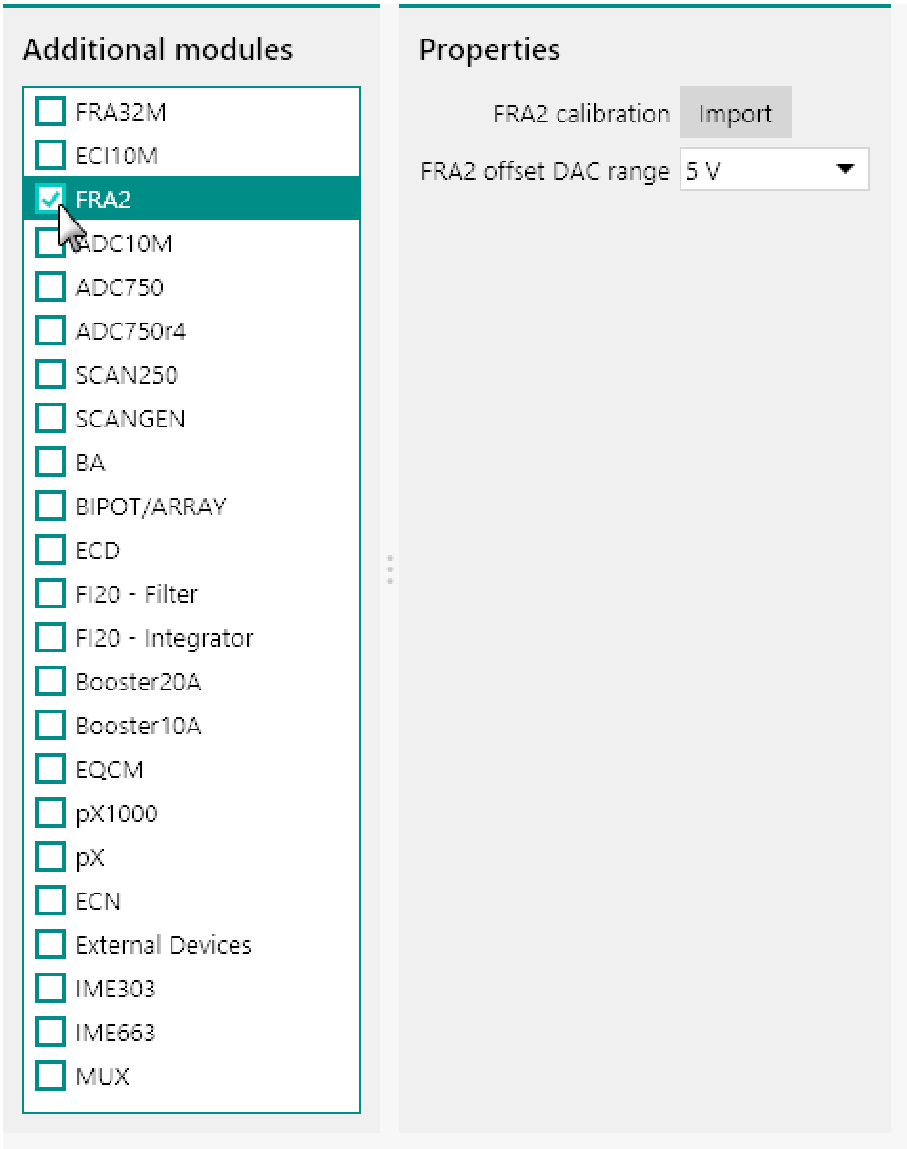


Figure 1205 The FRA2 module is selected in the hardware setup

The following additional properties must be defined:

- **FRA2 calibration:** the calibration file (*fra2cal.ini*) for the FRA2 module. The Import button is provided to specify the location of this file.
- **FRA2 offset DAC range:** specifies the offset DAC range of the FRA2 module using the provided drop-down list. This property can be set to either 5 V or 10 V.

CAUTION

The serial number specified in the **FRA2 calibration file** must match the serial number of the connected instrument.

CAUTION

The **FRA2 offset DAC range** property must be specified carefully. Failure to set this value properly may result in faulty data at frequencies of 25 Hz and lower (refer to front panel labels of the FRA2 module on the instrument *(see Chapter 16.3.2.12.9, page 1087)*)

16.3.2.12.4 C1 and C2 calibration factors

When the **FRA2** module is used in combination with the Autolab, the **C1** and **C2** calibration factors need to be determined.

NOTE

The **C1** and **C2** calibration factors are predetermined when the **FRA2** module is preinstalled. These factors must be determined experimentally when a **FRA2** module is installed into an existing instrument. This determination must only be carried out upon installation of the module.

Two procedures are supplied with NOVA to determine these calibration factors:

- PGSTAT C1 calibration
- PGSTAT C2 calibration

The determination of **C1** and **C2** requires the following items:

- Autolab Dummy cell
- Faraday cage

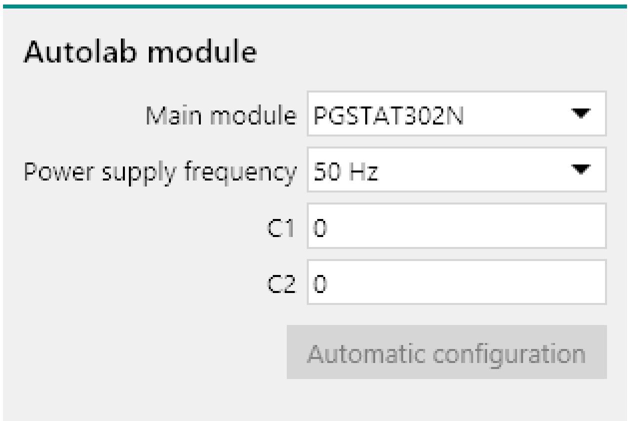
Typical values are indicated in *Table 49*.

Table 49 Typical values for C1 and C2

Instrument type	C1	C2
PGSTAT302N	1.6 E-11	3.0 E-13
PGSTAT302F	1.6 E-11	1.0 E-12
PGSTAT128N (serial number ≤ AUT84179)	2.6 E-11	1.0 E-12
PGSTAT128N (serial number > AUT84179)	1.6 E-11	1.0 E-12
PGSTAT100N	1.6 E-11	5.0 E-13
PGSTAT30	1.6 E-11	5.0 E-13

Instrument type	C1	C2
PGSTAT302	1.6 E-11	3.0 E-13
PGSTAT12	2.6 E-11	1.0 E-12
PGSTAT100	1.6 E-11	5.0 E-13
μAutolab type III	3.2 E-11	1.5 E-13

Before starting the determination of **C1** and **C2**, verify that the starting values are set to 0. In the Hardware setup panel, make sure that the value of C1 and C2 are set to 0 .



The calibration factors must be determined in sequence:

1. For the determination of C1, please refer to *Chapter 16.3.2.12.4.1*.
2. For the determination of C2, please refer to *Chapter 16.3.2.12.4.2*.

16.3.2.12.4.1 Determination of C1

Follow these steps to determine the value of the **C1** calibration factor.

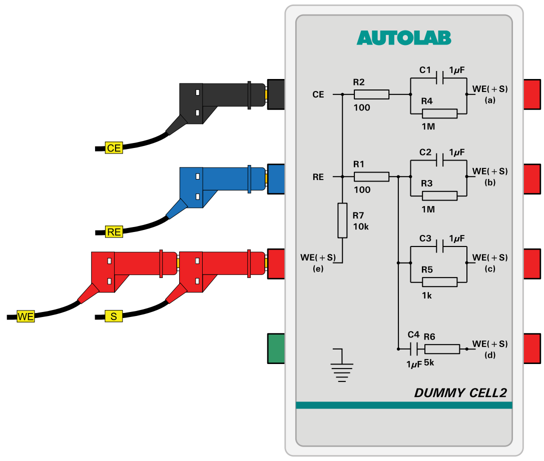
CAUTION

Do **not** connect the ground connector from the PGSTAT to the Autolab Dummy cell. Place the dummy cell in the Faraday cage.

1 Start the NOVA software and allow the instrument to warm up for at least 30 minutes.

2 Open the **PGSTAT C1 calibration** procedure.

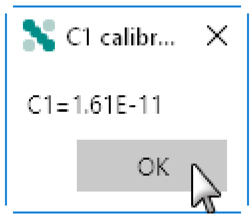
3 Connect the Autolab Dummy cell as shown. Connect the ground lead from the PGSTAT to the Faraday cage.



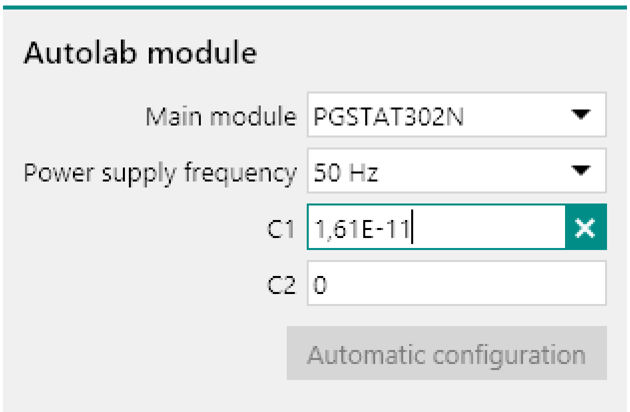
4 Start the measurement and wait until it finishes. Ignore the warning message displayed at the beginning of the measurement.

5 During the measurement, the data will be plotted as a Bode plot and should be similar to the example shown.

6 The measured data is automatically fitted and a message is shown at the end, displaying the measured **C1** value.



7 Open the instrument hardware setup and type the measured value in the **C1** field.



8 Close the hardware setup, wait for the Autolab to be reinitialized using the updated Hardware setup and continue with the determination of the **C2** calibration factor.

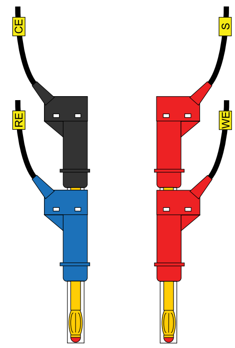
16.3.2.12.4.2 Determination of C2

> **CAUTION**
>
> Do **not** connect the ground connector from the PGSTAT to the Autolab Dummy cell. Place the dummy cell in the Faraday cage.

1 Make sure that the value of **C1** has already been determined, as specified in *Chapter 16.3.2.12.4.1*.

2 Open the **PGSTAT C2 calibration** procedure.

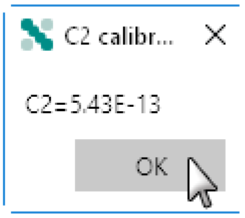
3 Disconnect the Autolab Dummy cell and leave the leads open in the Faraday cage. CE and RE must be connected together as well as WE and S. Make sure RE/CE and WE/S are not connected together. Connect the ground lead from the PGSTAT to the Faraday cage.



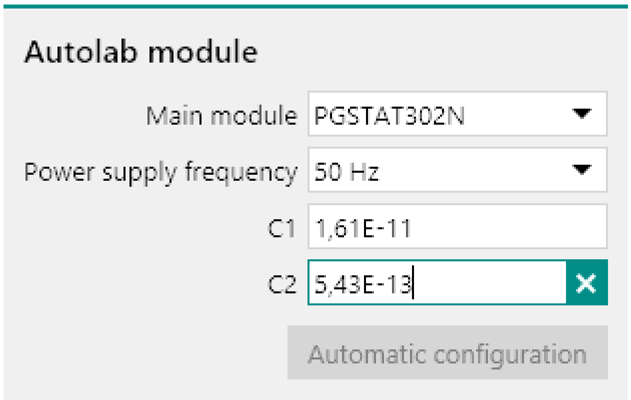
4 Start the measurement and wait until it finishes. Ignore the warning message displayed at the beginning of the measurement.

5 During the measurement, the data will be plotted as a Bode plot and should be similar to the example shown.

6 The measured data is automatically fitted and a message is shown at the end, displaying the measured **C2** value.



7 Open the instrument hardware setup and type the measured value in the **C2** field.



8 Close the hardware setup and wait for the Autolab to be reinitialized using the updated Hardware setup.

16.3.2.12.5 FRA2 module settings

The FRA2 module settings are completely defined in the NOVA software. The following user-definable settings are available, through the **Autolab control** command *(see Figure 1206, page 1085)*:

- **FRA2:** a  toggle that can be used to switch the output of the FRA2 on or off.
- **Frequency:** the output frequency of the FRA2 module, in Hz.
- **Amplitude:** the amplitude of the FRA2 module output, in V, specified as a TOP value.

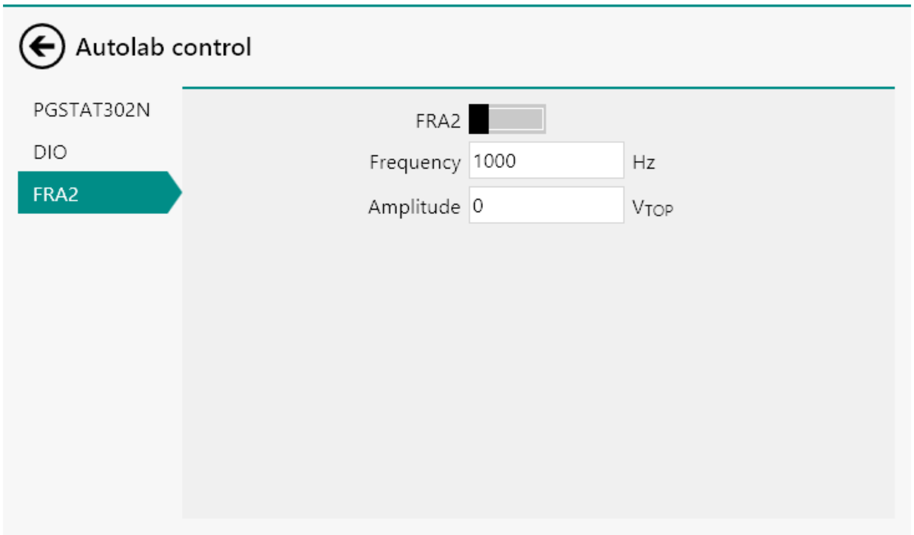


Figure 1206 The FRA2 module settings defined in the Autolab control command

16.3.2.12.6 FRA2 module manual control

The FRA2 can be manually controlled, using the Autolab display provided in the instrument control panel. This manual control can be used to perform impedance measurement, through the internal connection to the Autolab potentiostat/galvanostat or through the external connections provided by the FRA2 module.

The following properties can be adjusted in the manual control panel *(see Figure 1207, page 1086)*:

- **Frequency:** specifies the output frequency of the FRA2 module, in Hz.
- **Amplitude:** specifies the amplitude to output amplitude of the FRA2. The units depend on the mode of the potentiostat/galvanostat and on the *Input connection* property. The amplitude is specified as a top amplitude.
- **Input connection:** specifies if the measurement should be carried out internally (through the PGSTAT) or externally, using the external inputs provided on the front panel of the FRA2 module.
- **Wave type (Single sine, 5 sines or 15 sines):** specifies the type of signal used during the measurement. The choice is provided between the default single sine or the multi sine wave types.
- **Integration time:** specifies the time during which the signal is measured, in s.
- **Minimum number of cycles to integrate:** specifies the minimum number of cycles to integrate during the measurement.
- **FRA2:** a  toggle that can be used to switch the FRA2 module on or off.

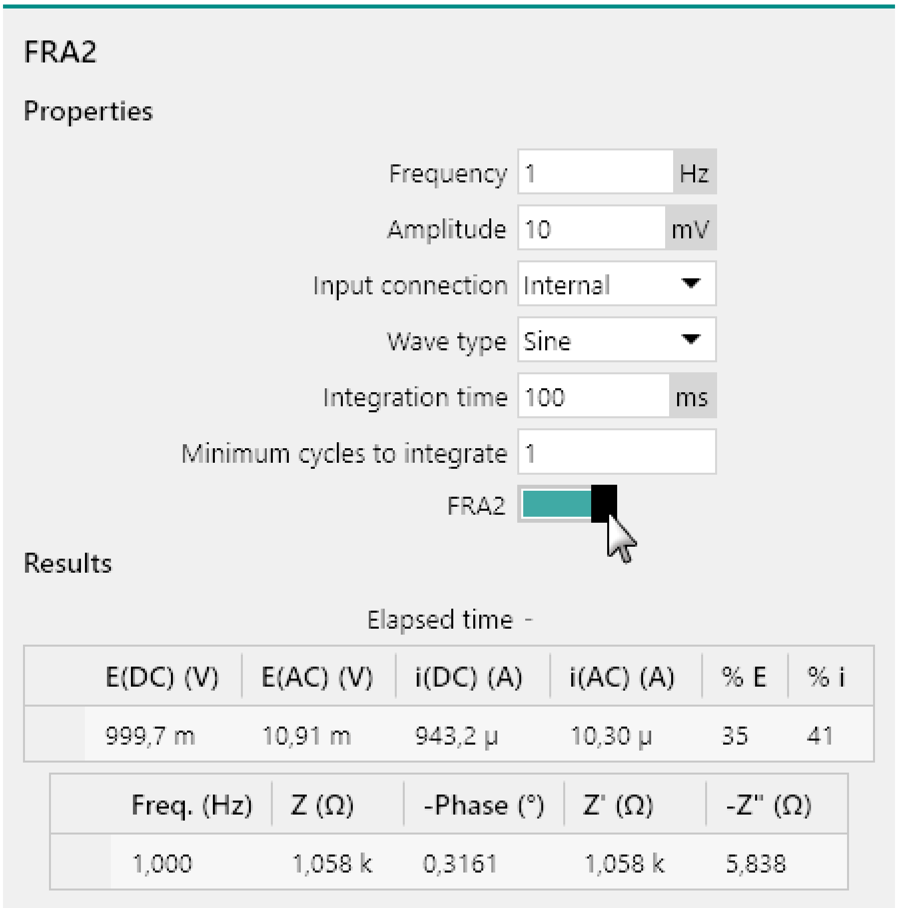


Figure 1207 The FRA2 manual control panel

As soon as the FRA2 is switched on using the provided toggle, the module will start a manual measurement using the properties specified in the panel. The measured values will be displayed after the measurement in the Results sub-panel.

NOTE

The measurement will continue until the module is switched off using the provided toggle.

NOTE

For impedance measurements using the potentiostat/galvanostat, it is necessary to set the potential or current, specify the DC potential or current and select the appropriate current range using the instrument manual control panel *(see Chapter 5.2.3, page 116)*.

16.3.2.12.7 FRA2 contour map

A typical contour map for the **FRA2** module in combination with the **PGSTAT302N** potentiostat/galvanostat is shown in *Figure 1208*.

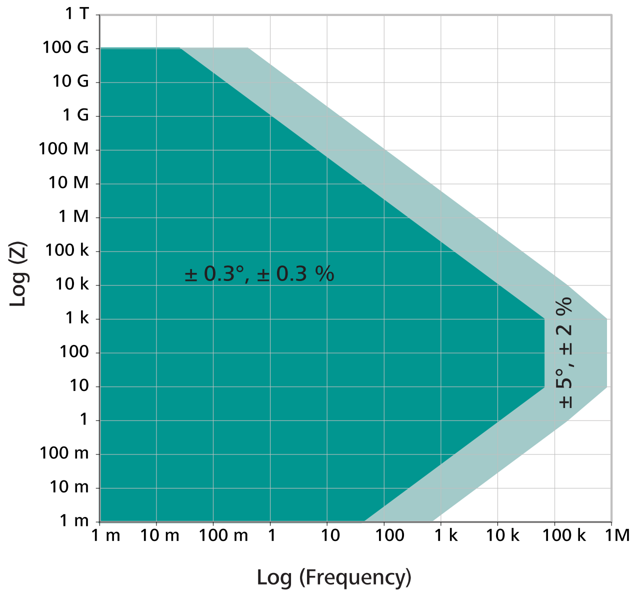


Figure 1208 Contour map of the PGSTAT302N/FRA2 combination

The map reported in *Figure 1208* shows a dark green area, which corresponds to the area of the map where an error of ± 0.3° on the measured phase angle and ± 0.3 % on the measured impedance value is expected. The light green area corresponds to the area of the map where an error of ± 5° on the measured phase angle and ± 2 % on the measured impedance value is expected.

NOTE

The contour map is determined empirically with the maximum possible amplitude, in potentiostatic mode.

16.3.2.12.8 FRA2 module restrictions

No restrictions apply when using the FRA2 module.

16.3.2.12.9 FRA2 module front panel connections

The FRA2 module has twice the size of a normal Autolab module. The module consists of a function generator module and transfer function analyzer. The FRA2 module is fitted with three female BNC connectors, labeled →X, →Y (on the transfer function analyzer front panel) and ←V (on the function generator front panel).

Two versions of the FRA2 module exist:

- **FRA2 module:** the default FRA2 module with a 5 V input range. This module is identified with the module labels shown in *Figure 1209*.
- **FRA2V10 module:** a modified version of the FRA2 module with a 10 V input range. This module is identified with the module labels shown in *Figure 1210*.

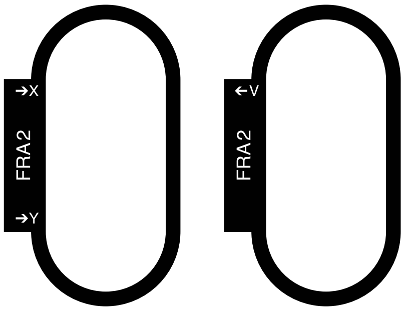


Figure 1209 The front panel labels of the FRA2 module (5 V input range version)

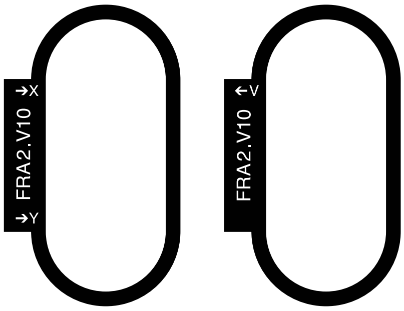


Figure 1210 The front panel labels of the FRA2 module (10 V input range version)

The two connectors, labeled →X and →Y are input connectors that can be used to analyze external transfer functions. They have an input range of ± 5 V or ± 10 V and an input impedance of 50 Ω.

The signal provided through the ←V connector on the front panel corresponds to the output of the sinewave generator of the FRA2. Whenever the FRA2 module is used, the voltage provided at this output corresponds to the signal generated by the module (either single sine or multi sine).

The output signal is a voltage, referred to the instrument ground, corresponding to the applied amplitude, multiplied by 10 (when the instrument is working in potentiostatic mode), or the converted amplitude, multiplied by 10 (when the instrument is working in galvanostatic mode):

$$E_{out}(\leftarrow V) = E_{(FRA2)} \cdot 10$$

$$E_{out}(\leftarrow V) = \frac{i_{(FRA2)}}{[CR]} \cdot 10$$

Where E_{out}(←V) corresponds to the output voltage signal of the module, in V, $E_{(FRA2)}$ and $i_{(FRA2)}$ corresponds to the specified amplitude, in V or A, respectively and and [CR] is the active current range of the FRA2 module.

NOTE

The front panel ←V BNC output is provided for information purposes only except for impedance measurement involving external transfer functions.

16.3.2.12.10 FRA2 module testing

NOVA is shipped with a procedure which can be used to verify that the **FRA2** module is working as expected.

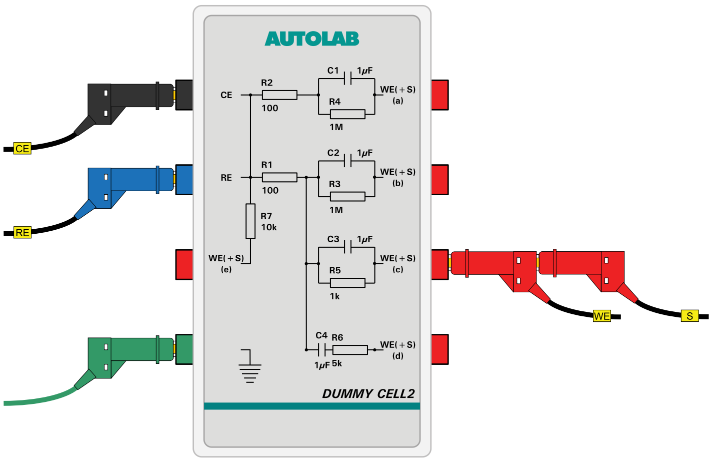
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestFRA** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test \TestFRA.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (c).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out an impedance spectroscopy measurement. During the measurement, the data is fitted using a R(RC) equivalent circuit. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1211*.

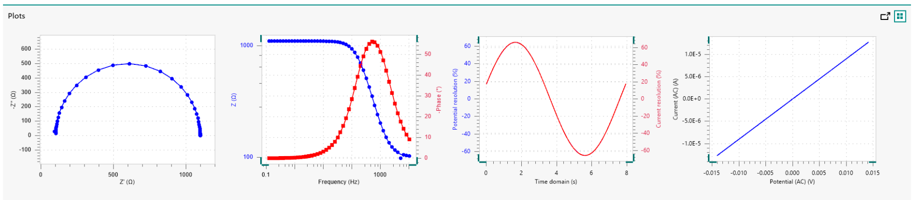
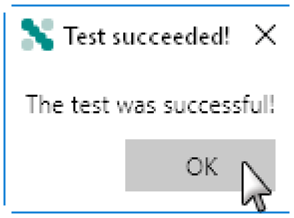


Figure 1211 The data measured by the TestFRA procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestFRA automatic evaluation of the data requires the following tests to succeed:

1. The fitted series resistance must be equal to 100 Ω± 5 % .
2. The fitted parallel resistance must be equal to 1000 Ω± 5 % .
3. The fitted parallel capacitance must be equal to 1 µF ± 10 % .
4. The calculated χ^2 must be smaller or equal to 0.01.

All four conditions must be valid for the test to succeed.

16.3.2.12.11 FRA2 module specifications

The specifications of the FRA2 module are provided in *Table 50*.

Table 50 Specifications of the FRA2 module

Specification	Value
Frequency range	10 µHz - 4 MHz
Frequency range in combination with Autolab PGSTAT	10 µHz - 1 MHz
Frequency resolution	0.003 %
Input range	± 10 V
Output amplitude, potentiostatic mode	0.2 mV to 0.350 mV (RMS)

Specification	Value
Output amplitude, galvanostatic mode	0.0002 - 0.35 times current range (RMS)
Output amplitude, external mode	2 mV - 3.5 V (RMS)
Input resolution	12 bit

16.3.2.13 FRA32M module

The FRA32M module is an extension module for the Autolab PGSTAT and the Multi Autolab. This module consists of function generator and a transfer function analyzer. The function generator can be used to generate a sine wave based signal and analyze the transfer function between two sine wave based signals. Instruments fitted with this module can perform electrochemical impedance spectroscopy (EIS) measurements.

16.3.2.13.1 FRA32M module compatibility

The FRA32M module is available for the following instruments:

- PGSTAT302N, PGSTAT302 and PGSTAT30
- PGSTAT128N and PGSTAT12
- PGSTAT100N and PGSTAT100
- M101
- PGSTAT204/M204

NOTE

The FRA32M module is **not** compatible with the Autolab instruments not listed above.

16.3.2.13.2 FRA32M module scope of delivery

The FRA32M module is supplied with the following items:

- FRA32M module
- FRA32M module label

16.3.2.13.3 C1 and C2 calibration factors

When the **FRA32M** module is used in combination with the Autolab, the **C1** and **C2** calibration factors need to be determined.

NOTE

The **C1** and **C2** calibration factors are predetermined when the **FRA32M** module is preinstalled. These factors must be determined experimentally when a **FRA32M** module is installed into an existing instrument. This determination must only be carried out upon installation of the module.

NOTE

On some instruments, the value of **C1** and **C2** is already determined and stored in the on-board processor of the instrument. In this case, the values reported in the Hardware setup are not 0. For these instruments, it is not necessary to determine **C1** and **C2**.

Two procedures are supplied with NOVA to determine these calibration factors:

- PGSTAT C1 calibration
- PGSTAT C2 calibration

The determination of **C1** and **C2** requires the following items:

- Autolab Dummy cell
- Faraday cage

CAUTION

The determination of the **C1** and **C2** calibration factors is not required for the PGSTAT204 and for the M101 and M204 modules used in combination with the **FRA32M** module in the Multi Autolab instrument.

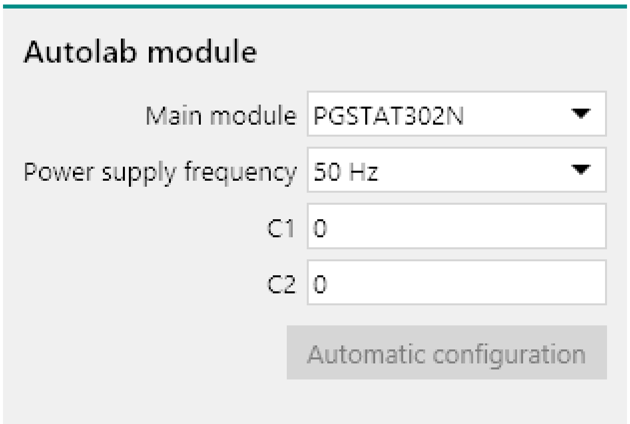
Typical values are indicated in *Table 51*.

Table 51 Typical values for C1 and C2

Instrument type	**C1**	**C2**
PGSTAT302N	1.6 E-11	3.0 E-13
PGSTAT302F	1.6 E-11	1.0 E-12
PGSTAT128N (serial number ≤ AUT84179)	2.6 E-11	1.0 E-12

Instrument type	C1	C2
PGSTAT128N (serial number > AUT84179)	1.6 E-11	1.0 E-12
PGSTAT100N	1.6 E-11	5.0 E-13
PGSTAT30	1.6 E-11	5.0 E-13
PGSTAT302	1.6 E-11	3.0 E-13
PGSTAT12	2.6 E-11	1.0 E-12
PGSTAT100	1.6 E-11	5.0 E-13

Before starting the determination of **C1** and **C2**, verify that the starting values are set to 0. In the Hardware setup panel, make sure that the value of C1 and C2 are set to 0 .



The calibration factors must be determined in sequence:

1. For the determination of C1, please refer to *Chapter 16.3.2.13.3.1*.
2. For the determination of C2, please refer to *Chapter 16.3.2.13.3.2*.

16.3.2.13.3.1 Determination of C1

Follow these steps to determine the value of the **C1** calibration factor.

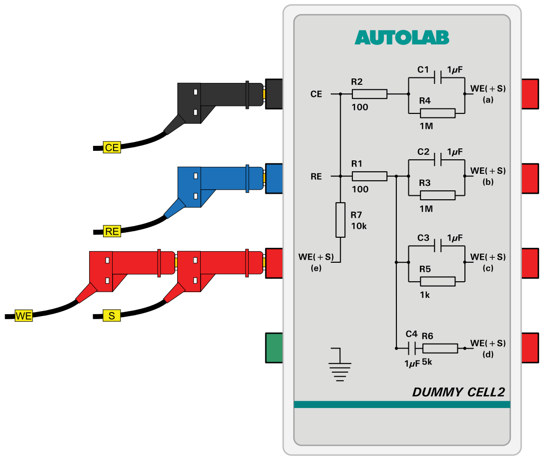
CAUTION

Do **not** connect the ground connector from the PGSTAT to the Autolab Dummy cell. Place the dummy cell in the Faraday cage.

1 Start the NOVA software and allow the instrument to warm up for at least 30 minutes.

2 Open the **PGSTAT C1 calibration** procedure.

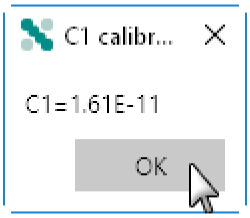
3 Connect the Autolab Dummy cell as shown. Connect the ground lead from the PGSTAT to the Faraday cage.



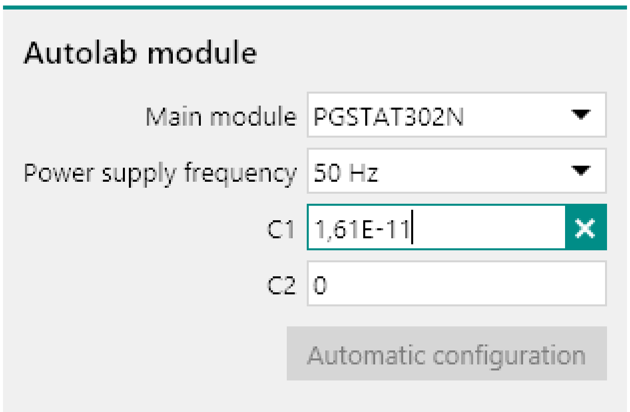
4 Start the measurement and wait until it finishes. Ignore the warning message displayed at the beginning of the measurement.

5 During the measurement, the data will be plotted as a Bode plot and should be similar to the example shown.

6 The measured data is automatically fitted and a message is shown at the end, displaying the measured **C1** value.



7 Open the instrument hardware setup and type the measured value in the **C1** field.



8 Close the hardware setup, wait for the Autolab to be reinitialized using the updated Hardware setup and continue with the determination of the **C2** calibration factor.

16.3.2.13.3.2 Determination of C2

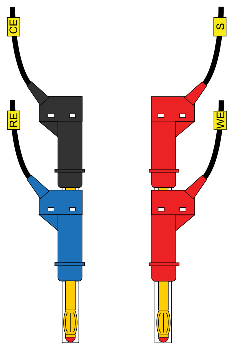
CAUTION

Do **not** connect the ground connector from the PGSTAT to the Autolab Dummy cell. Place the dummy cell in the Faraday cage.

1 Make sure that the value of **C1** has already been determined, as specified in *Chapter 16.3.2.12.4.1*.

2 Open the **PGSTAT C2 calibration** procedure.

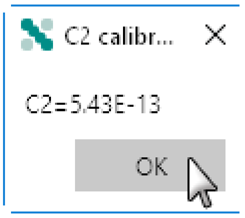
3 Disconnect the Autolab Dummy cell and leave the leads open in the Faraday cage. CE and RE must be connected together as well as WE and S. Make sure RE/CE and WE/S are not connected together. Connect the ground lead from the PGSTAT to the Faraday cage.



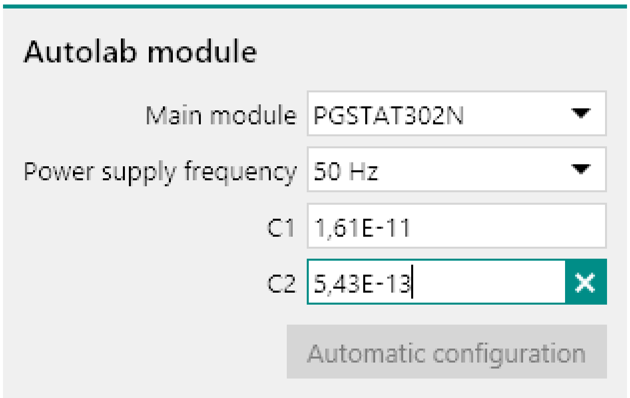
4 Start the measurement and wait until it finishes. Ignore the warning message displayed at the beginning of the measurement.

5 During the measurement, the data will be plotted as a Bode plot and should be similar to the example shown.

6 The measured data is automatically fitted and a message is shown at the end, displaying the measured **C2** value.



7 Open the instrument hardware setup and type the measured value in the **C2** field.



8 Close the hardware setup and wait for the Autolab to be reinitialized using the updated Hardware setup.

16.3.2.13.4 FRA32M module settings

The FRA32M module settings are completely defined in the NOVA software. The following user-definable settings are available, through the **Autolab control** command *(see Figure 1212, page 1097)*:

- **FRA32M:** a toggle that can be used to switch the output of the FRA32M on or off.
- **Frequency:** the output frequency of the FRA32M module, in Hz.
- **Amplitude:** the amplitude of the FRA32M module output, in V, specified as a TOP value.
- **Input impedance 50 Ω:** a toggle that can be used to set the input impedance of the FRA32M module to 50 Ω.

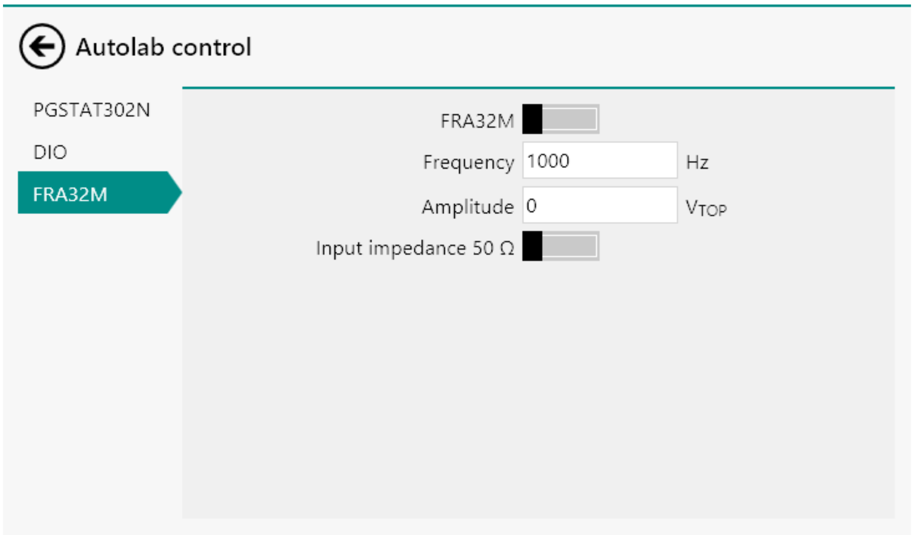


Figure 1212 The FRA32M module settings defined in the Autolab control command

16.3.2.13.5 FRA32M module manual control

The FRA32M can be manually controlled, using the Autolab display provided in the instrument control panel. This manual control can be used to perform impedance measurement, through the internal connection to the Autolab potentiostat/galvanostat or through the external connections provided by the FRA32M module.

The following properties can be adjusted in the manual control panel *(see Figure 1213, page 1098)*:

- **Frequency:** specifies the output frequency of the FRA32M module, in Hz.
- **Amplitude:** specifies the amplitude to output amplitude of the FRA32M. The units depend on the mode of the potentiostat/galvanostat and on the *Input connection* property. The amplitude is specified as a top amplitude.
- **Input connection:** specifies if the measurement should be carried out internally (through the PGSTAT) or externally, using the external inputs provided on the front panel of the FRA32M module.
- **Wave type (Single sine, 5 sines or 15 sines):** specifies the type of signal used during the measurement. The choice is provided between the default single sine or the multi sine wave types.
- **Integration time:** specifies the time during which the signal is measured, in s.
- **Minimum number of cycles to integrate:** specifies the minimum number of cycles to integrate during the measurement.
- **FRA32M:** a toggle that can be used to switch the FRA32M module on or off.

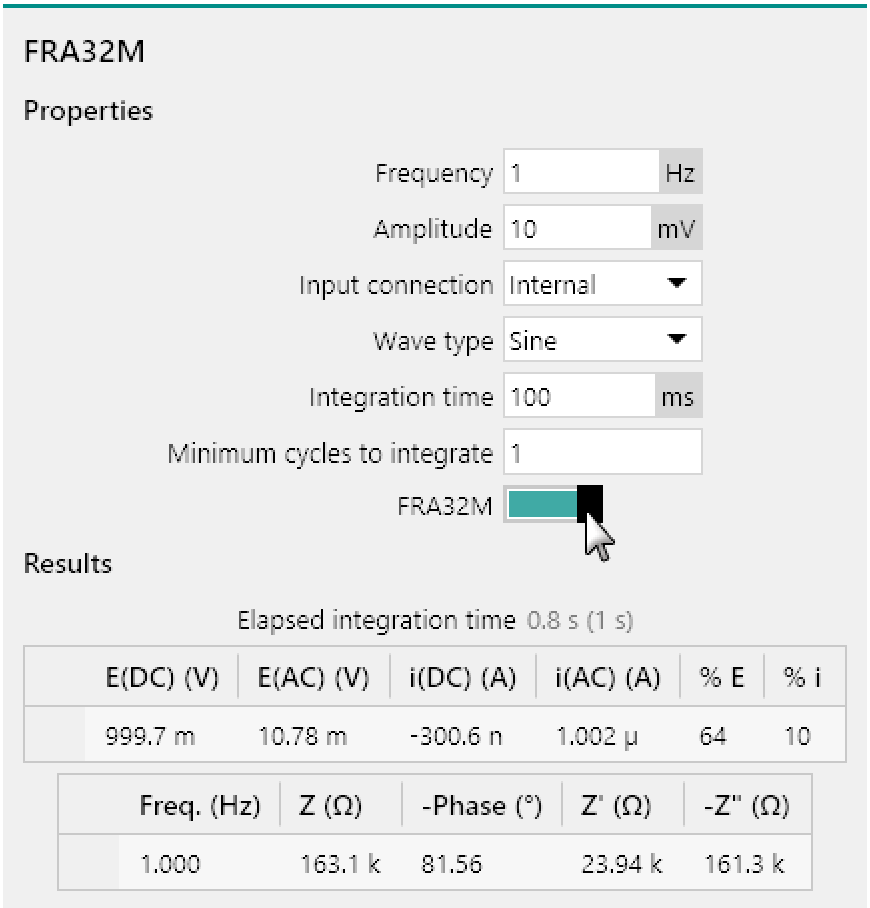


Figure 1213 The FRA32M manual control panel

As soon as the FRA32M is switched on using the provided toggle, the module will start a manual measurement using the properties specified in the panel. The measured values will be displayed after the measurement in the **Results** sub-panel.

NOTE

The measurement will continue until the module is switched off using the provided toggle.

NOTE

For impedance measurements using the potentiostat/galvanostat, it is necessary to set the potential or current, specify the DC potential or current and select the appropriate current range using the instrument manual control panel *(see Chapter 5.2.3, page 116)*.

16.3.2.13.6 FRA32M contour map

A typical contour map for the **FRA32M** module in combination with the **PGSTAT302N** potentiostat/galvanostat is shown in *Figure 1214*.

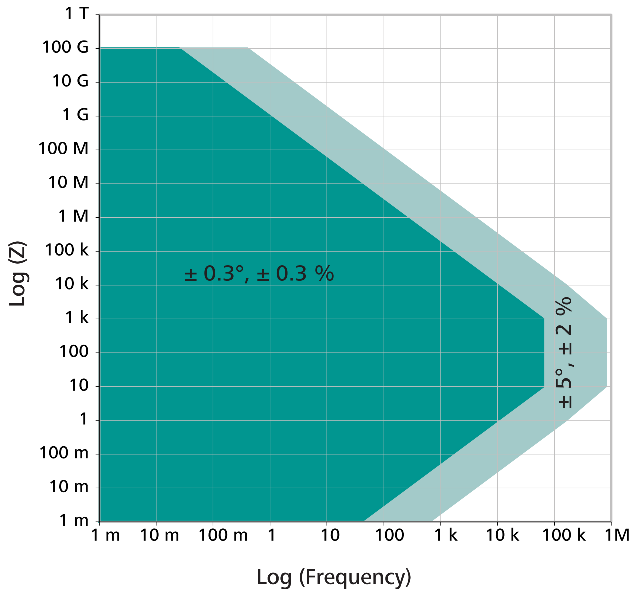


Figure 1214 Contour map of the PGSTAT302N/FRA32M combination

The map reported in *Figure 1214* shows a dark green area, which corresponds to the area of the map where an error of ± 0.3° on the measured phase angle and ± 0.3 % on the measured impedance value is expected. The light green area corresponds to the area of the map where an error of ± 5° on the measured phase angle and ± 2 % on the measured impedance value is expected.

NOTE

The contour map is determined empirically with the maximum possible amplitude, in potentiostatic mode.

16.3.2.13.7 FRA32M module restrictions

No restrictions apply when using the FRA32M module.

16.3.2.13.8 FRA32M module front panel connections

The FRA32M module is fitted with three female SMB connectors, labeled →X, →Y and ←V, from top to bottom *(see Figure 1215, page 1100)*.

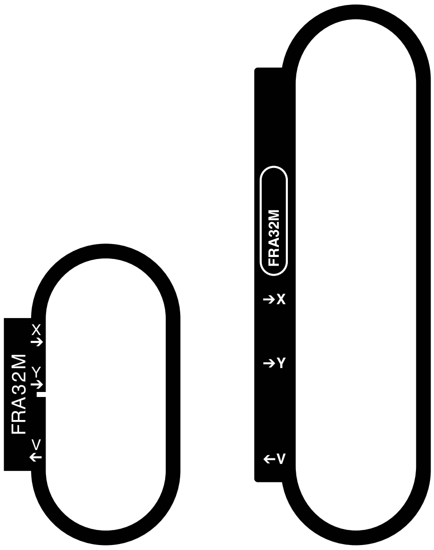


Figure 1215 The front panel labels of the FRA32M module (left: FRA32M module in PGSTAT, right: FRA32M module in Multi Autolab)

The two connectors, labeled →X and →Y are input connectors that can be used to analyze external transfer functions. They have an input range of ± 10 V and an input impedance of 50 **Ω**.

The signal provided through the ←V connector on the front panel corresponds to the output of the sinewave generator of the FRA32M. Whenever the FRA32M module is used, the voltage provided at this output corresponds to the signal generated by the module (either single sine or multi sine).

The output signal is a voltage, referred to the instrument ground, corresponding to the applied amplitude, multiplied by 10 (when the instrument is working in potentiostatic mode), or the converted amplitude, multiplied by 10 (when the instrument is working in galvanostatic mode):

$$E_{out}(\leftarrow V) = E_{(FRA32M)} \cdot 10$$

$$E_{out}(\leftarrow V) = \frac{i_{(FRA32M)}}{[CR]} \cdot 10$$

Where $E_{out}(\leftarrow V)$ corresponds to the output voltage signal of the module, in V, $E_{(FRA32M)}$ and $i_{(FRA32M)}$ corresponds to the specified amplitude, in V or A, respectively and and [CR] is the active current range of the FRA32M module.

NOTE

The front panel ←V SMB output is provided for information purposes only except for impedance measurement involving external transfer functions.

16.3.2.13.9 FRA32M module testing

NOVA is shipped with a procedure which can be used to verify that the **FRA32M** module is working as expected.

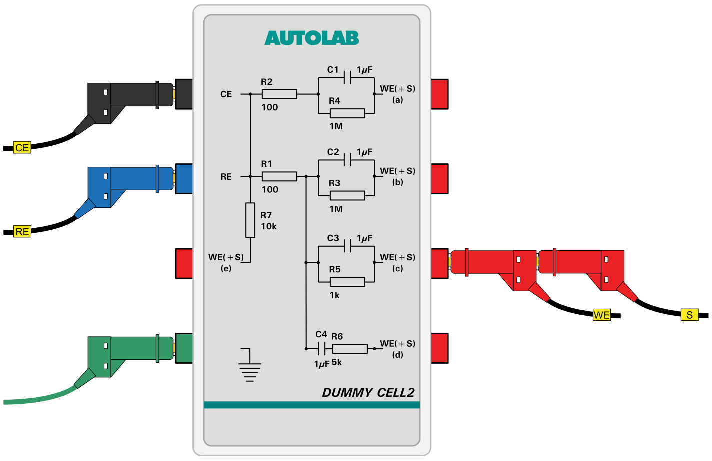
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestFRA** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test \TestFRA.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (c).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out an impedance spectroscopy measurement. During the measurement, the data is fitted using a R(RC) equivalent circuit. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1216*.

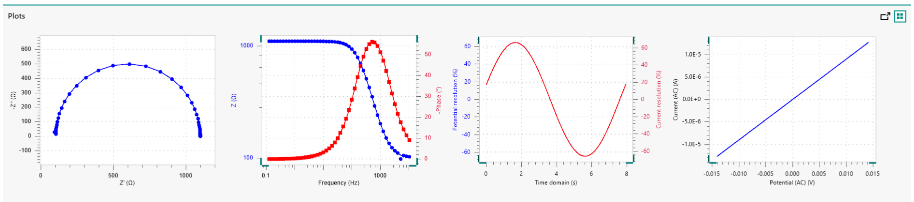
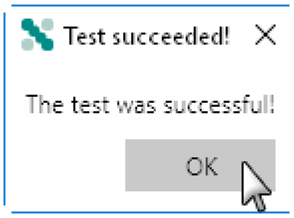


Figure 1216 The data measured by the TestFRA procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestFRA automatic evaluation of the data requires the following tests to succeed:

1. The fitted series resistance must be equal to 100 Ω± 5 %.
2. The fitted parallel resistance must be equal to 1000 Ω± 5 %.
3. The fitted parallel capacitance must be equal to 1 µF ± 10 %.
4. The calculated χ^2 must be smaller or equal to 0.01.

All four conditions must be valid for the test to succeed.

16.3.2.13.10 FRA32M module specifications

The specifications of the FRA32M module are provided in *Table 52*.

Table 52 Specifications of the FRA32M module

Specification	Value
Frequency range	10 µHz - 32 MHz
Frequency range in combination with Autolab PGSTAT	10 µHz - 1 MHz
Frequency range in combination with ECI10M module	40 Hz - 10 MHz
Frequency resolution	0.003 %
Input range	± 10 V
Output amplitude, potentiostatic mode	0.2 mV to 0.350 mV (RMS)

Specification	Value
Output amplitude, galvanostatic mode	0.0002 - 0.35 times current range (RMS)
Output amplitude, external mode	2 mV - 3.5 V (RMS)
Input resolution	14 bit

16.3.2.14 IME303 module

The IME303 is an external interface to the Princeton Applied Research PAR303(A) Stand.

The Princeton Applied Research PAR303(A) Stand is a polarographic stand which provides the means to perform electrochemical measurements using a mercury drop electrode. The mercury drops are formed at the very end of a narrow glass capillary. The Princeton Applied Research PAR303(A) Stand can operate in three different modes:

- **Dropping Mercury Electrode (DME):** in this mode mercury drops form at the end of the capillary. The drop grows until the weight of the drop exceeds the surface tension and the drop falls into the solution, leading to a new drop at end of the capillary.
- **Static Drop Mercury Electrode (SDME):** in this mode mercury drops are formed at the end of the capillary. The drop grows until a hardware-controlled size and it remains at the end of the capillary until the tapper, built into the Princeton Applied Research PAR303(A) Stand, is activated. This dislodges the mercury drop, which falls into the solution, leading to a new, identical drop at the end of the capillary.
- **Hanging Drop Mercury Electrode (HDME):** in this mode a single mercury drop is formed at the end of the capillary. The drop grows until a hardware-controlled size and it remains at the end of the capillary. The tapper, can be activated if needed to dislodge this drop and create a new drop at the end of the capillary.

The IME303 provides remote controls for the Princeton Applied Research PAR303(A) Stand. The following actions can be controlled through the IME303:

- **Stirrer on/off:** the optional stirrer connected to the Princeton Applied Research PAR303(A) Stand can be remotely switched on or off.
- **Purge on/off:** the purge function of the Princeton Applied Research PAR303(A) Stand can be remotely switched on or off.
- **Create new drop:** the tapper of the Princeton Applied Research PAR303(A) Stand can be remotely activated. This knocks the current mercury drop from the electrode and created a new drop.

NOTE

For more information on the Princeton Applied Research PAR303(A) Stand, please consult the corresponding User Manual.

CAUTION

Take all necessary precautions when working with mercury. It is highly recommended to consult the Material Safety Data Sheet (MSDS) before operating the Princeton Applied Research PAR303(A) Stand. It is also recommended to dispose of the mercury waste properly.

16.3.2.14.1 IME303 module compatibility

The IME303 module is available for the following instruments:

- PGSTAT302N, PGSTAT302 and PGSTAT30
- PGSTAT128N and PGSTAT12
- PGSTAT100N and PGSTAT100
- PGSTAT101, M101, PGSTAT204 and M204
- PGSTAT20 and PGSTAT10
- µAutolab II and µAutolab III

NOTE

The IME303 module is **not** compatible with the Autolab instruments not listed above.

16.3.2.14.2 IME303 module scope of delivery

Depending on the type of instrument it is connected to, the IME303 is the following items:

- **For PGSTAT302N, 302, 30, 128N, 12, 100N and 100, µAutolab II and µAutolab III, PGSTAT10 and PGSTAT20:**
  - IME303 interface.
  - Power cable.
  - Autolab to IME303 DIO cable: a cable fitted with a female 25 pin SUB-D connector (labeled DIO) and a female 15 pin SUB-D connector (labeled IME).
  - IME303 to Princeton Applied Research PAR303(A) Stand cable: a cable fitted with a male 9 pin SUB-D connector and a female 25 SUB-D connector.
  - 4 mm to 2 mm banana plug adapters (3).
- **For PGSTAT101, M101, PGSTAT204, M204:**
  - IME303.
  - Power cable.
  - Autolab to IME303 DIO cable: a cable fitted with a female 15 pin SUB-D connector (labeled DIO) and a female 15 pin SUB-D connector (labeled IME).
  - IME303 to Princeton Applied Research PAR303(A) Stand cable: a cable fitted with a male 9 pin SUB-D connector and a female 25 SUB-D connector.
  - 4 mm to 2 mm banana plug adapters (3).

16.3.2.14.3 IME303 module settings

The IME303 module settings are defined in the NOVA software. The following user-definable setting is available, through the **Autolab control** command *(see Figure 1217, page 1106)*:

- **Purge:** this control can be used to switch the nitrogen purge of the Princeton Applied Research PAR303(A) Stand on or off.
- **Stirrer:** this control can be used to switch the stirrer of the Princeton Applied Research PAR303(A) Stand on or off if the stirrer is installed.
- **Number of new drops:** this control can be used to create the specified number of new drops by activating the tapper of the Princeton Applied Research PAR303(A) Stand as many times.

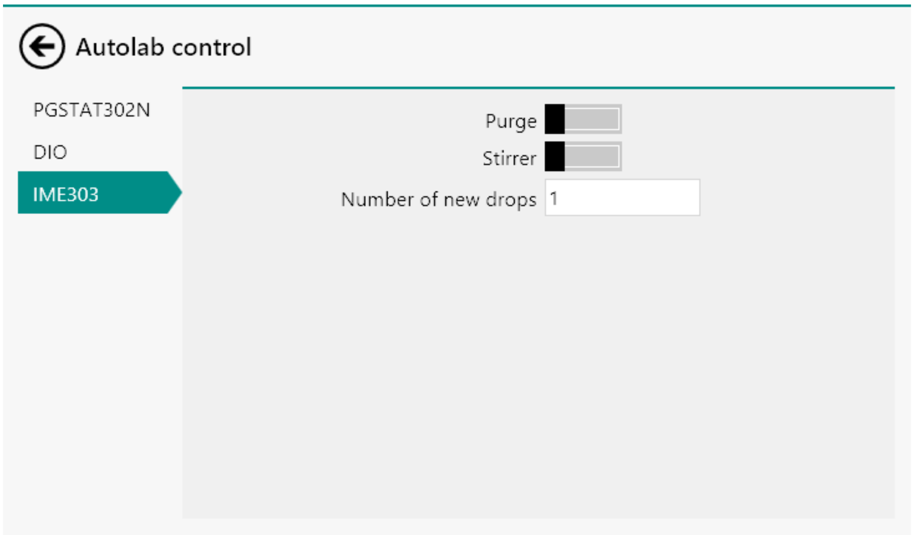


Figure 1217 The IME303 module settings

NOTE

A 500 ms settling time is used each time the tapper is activated. This settling time can be adjusted in the hardware setup.

16.3.2.14.4 IME303 module manual control

The IME303 can be manually controlled, using the **Autolab display** provided in the instrument control panel *(see Figure 1218, page 1107)*. The dedicated manual control panel can be used to perform the following tasks:

- **Purge (on/off toggle):** this control can be used to switch the nitrogen purge of the Princeton Applied Research PAR303(A) Stand on or off.
- **Stirrer (on/off toggle):** this control can be used to switch the stirrer of the Princeton Applied Research PAR303(A) Stand on or off if the stirrer is installed.
- **Create new drop (button):** this control can be used to create a new drop by activating the tapper of the Princeton Applied Research PAR303(A) Stand.

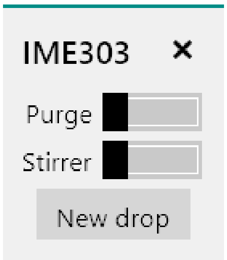


Figure 1218 Manual control of the IME303 provided in the Autolab display

NOTE

The manual control panel provided for the IME303 can be used to set or display the current settings of the IME303.

16.3.2.14.5 IME303 module restrictions

No restrictions apply when using the IME303 module.

16.3.2.14.6 IME303 module front panel controls

The front panel of the IME303 provides a number of controls and indicators, shown in *Figure 1219*.

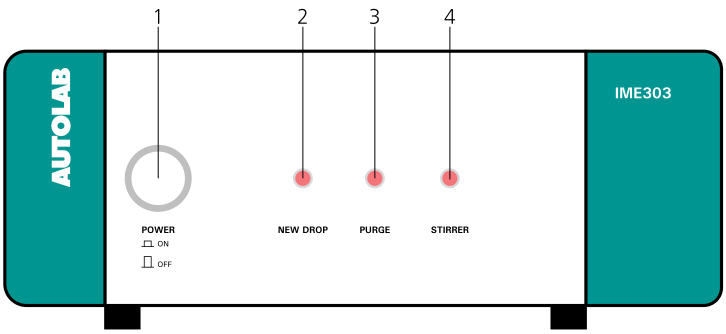


Figure 1219 Overview of the front panel of the IME303

1 Power On/Off button
For switching the IME303 on or off.

2 New drop indicator LED
Flashes when the tapper of the Princeton Applied Research PAR303(A) Stand is activated to create a new drop.

3 Purge LED
Indicates that the nitrogen purge is on when lit.

4 Stirrer LED
Indicates that the stirrer is on when lit.

16.3.2.14.7 IME303 module back plane connections

The back plane of the IME663 provides a number of connections, shown in *Figure 1220*.

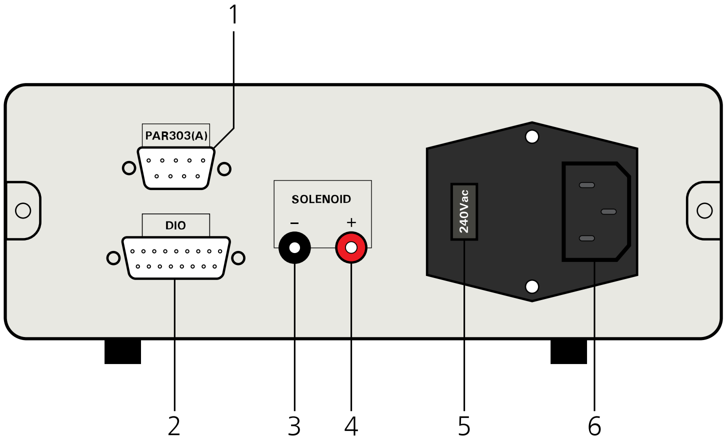


Figure 1220 Overview of the back plane of the IME303

1 PAR303(A) connector
For connecting the IME303 to the J1 or Remote connector located on the back plane of the Princeton Applied Research PAR303(A) Stand.

2 DIO connector
For connecting the IME303 to the DIO connector of the Autolab.

3 SOLENOID - connector
For connecting the negative pole of a third party tapper.

4 SOLENOID + connector
For connecting the positive pole of a third party tapper.

5 Mains voltage indicator
Indicates the mains voltage settings of the IME303.

5 Mains connection socket
For connecting the IME303 to the mains supply.

NOTE

A third party drop tapper can be controlled with the IME303. Use the SOLENOID marked banana sockets located on the back plane of the IME303. Every time a new drop is created (through the a command or manual control of the IME303) a 15 V pulse will be generated between the + and - connectors.

CAUTION

Make sure that the mains voltage indicator is set properly before switching the IME303 on.

16.3.2.15 IME663 module

The IME663 is an external interface to the Metrohm 663 VA Stand *(see Figure 1221, page 1109)*.

Figure 1221 The Metrohm 663 VA Stand (left) and the IME663 interface (right)

The Metrohm 663 VA Stand is a polarographic stand which provides the means to perform electrochemical measurements using a mercury drop electrode. The mercury drops are formed at the very end of a narrow glass capillary. The Metrohm 663 VA Stand can operate in three different modes:

- **Dropping Mercury Electrode (DME):** in this mode mercury drops form at the end of the capillary. The drop grows until the weight of the drop exceeds the surface tension and the drop falls into the solution, leading to a new drop at end of the capillary.
- **Static Drop Mercury Electrode (SDME):** in this mode mercury drops are formed at the end of the capillary. The drop grows until a hardware-controlled size and it remains at the end of the capillary until the tapper, built into the Metrohm 663 VA stand, is activated. This dislodges the mercury drop, which falls into the solution, leading to a new, identical drop at the end of the capillary.
- **Hanging Drop Mercury Electrode (HDME):** in this mode a single mercury drop is formed at the end of the capillary. The drop grows until a hardware-controlled size and it remains at the end of the capillary. The tapper, can be activated if needed to dislodge this drop and create a new drop at the end of the capillary.

The IME663 provides manual and remote controls for the Metrohm 663 VA Stand. The following actions can be controlled through the IME663:

- **Stirrer on/off:** the stirrer of the Metrohm 663 VA Stand can be manually or remotely switched on or off.
- **Purge on/off:** the purge function of the Metrohm 663 VA Stand can be remotely switched on or off.
- **Create new drop:** the tapper of the Metrohm 663 VA Stand can be remotely activated. This knocks the current mercury drop from the electrode and created a new drop.

NOTE

Remote control of the Metrohm 663 VA Stand only works when the stand is set to **SDME** mode.

NOTE

For more information on the Metrohm 663 VA Stand, please consult the corresponding User Manual.

CAUTION

Take all necessary precautions when working with mercury. It is highly recommended to consult the Material Safety Data Sheet (MSDS) before operating the Metrohm 663 VA Stand. It is also recommended to dispose of the mercury waste properly.

16.3.2.15.1 IME663 module compatibility

The IME663 module is available for the following instruments:

- PGSTAT302N, PGSTAT302 and PGSTAT30
- PGSTAT128N and PGSTAT12
- PGSTAT100N and PGSTAT100
- PGSTAT101, M101, PGSTAT204 and M204
- PGSTAT20 and PGSTAT10
- µAutolab II and µAutolab III

NOTE

The IME663 module is **not** compatible with the Autolab instruments not listed above.

16.3.2.15.2 IME663 module scope of delivery

Depending on the type of instrument it is connected to, the IME663 is the following items:

- **For PGSTAT302N, 302, 30, 128N, 12, 100N and 100:**
  - IME663 interface.
  - Power cable.
  - Autolab to IME663 DIO cable: a cable fitted with a female 25 pin SUB-D connector (labeled DIO) and a female 15 pin SUB-D connector (labeled IME).
  - Stirrer cable.
  - IME663 to Metrohm 663 VA Stand cable: a cable fitted with a female 9 pin SUB-D connector and a female 26 contact Metrohm cartridge connector (together with two green ground cables).
  - Electrode adapters.
- **For PGSTAT101, M101, PGSTAT204, M204:**
  - IME663.
  - Power cable.
  - Autolab to IME663 DIO cable: a cable fitted with a female 15 pin SUB-D connector (labeled DIO) and a female 15 pin SUB-D connector (labeled IME).
  - Stirrer cable.
  - IME663 to Metrohm 663 VA Stand cable: a cable fitted with a female 9 pin SUB-D connector and a female 26 contact Metrohm cartridge connector (together with two green ground cables).
  - Electrode adapters.
- **For PGSTAT10, PGSTAT20, µAutolab II and µAutolab III:**
  - IME663.
  - Power cable.
  - Autolab to IME663 DIO cable: a cable fitted with a female 25 pin SUB-D connector (labeled DIO) and a female 15 pin SUB-D connector (labeled IME).
  - Stirrer cable.
  - IME663 to Metrohm 663 VA Stand cable: a cable fitted with a female 9 pin SUB-D connector, a female 26 contact Metrohm cartridge connector (together with a green ground cable) and a cell cable fitted with a male, 7 pin DIN connector.

16.3.2.15.3 IME663 module settings

The IME663 module settings are defined in the NOVA software. The following user-definable setting is available, through the **Autolab control** command *(see Figure 1222, page 1112)*:

- **Purge:** this control can be used to switch the nitrogen purge of the Metrohm 663 VA Stand on or off.
- **Stirrer:** this control can be used to switch the stirrer of the Metrohm 663 VA Stand on or off if the stirrer switch on the IME663 is set to remote control.
- **Number of new drops:** this control can be used to create the specified number of new drops by activating the tapper of the Metrohm 663 VA Stand as many times.

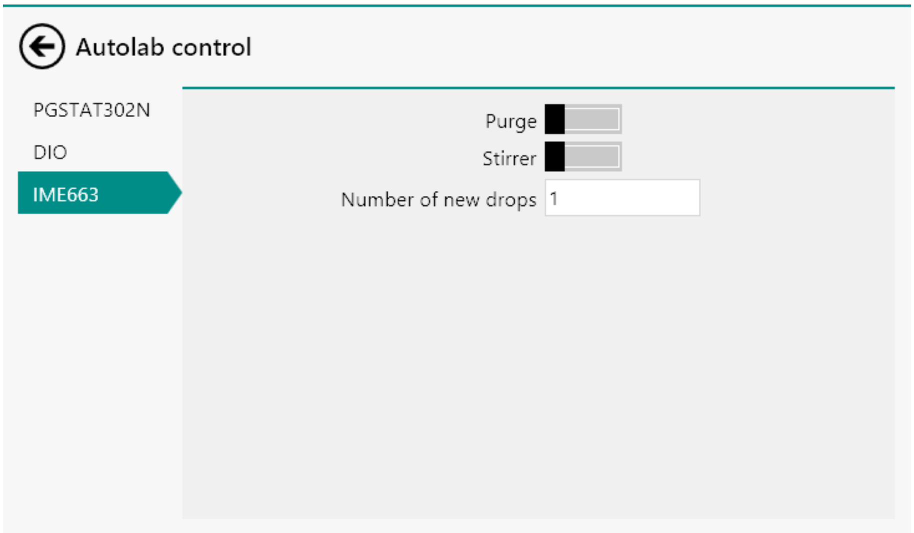


Figure 1222 The IME663 module settings

NOTE

A 500 ms settling time is used each time the tapper is activated. This settling time can be adjusted in the hardware setup.

16.3.2.15.4 IME663 module manual control

The IME663 can be manually controlled, using the **Autolab display** provided in the instrument control panel *(see Figure 1223, page 1113)*. The dedicated manual control panel can be used to perform the following tasks:

- **Purge (on/off toggle):** this control can be used to switch the nitrogen purge of the Metrohm 663 VA Stand on or off.

- **Stirrer (on/off toggle):** this control can be used to switch the stirrer of the Metrohm 663 VA Stand on or off if the stirrer switch on the IME663 is set to remote control.
- **Create new drop (button):** this button can be used to create a new drop by activating the tapper of the Metrohm 663 VA Stand.

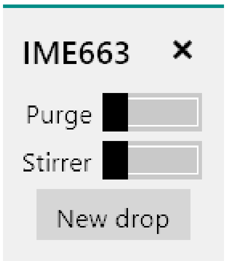


Figure 1223 Manual control of the IME663 provided in the Autolab display

NOTE

The manual control panel provided for the IME663 can be used to set or display the current settings of the IME663.

16.3.2.15.5 IME663 module restrictions

No restrictions apply when using the IME663 module.

16.3.2.15.6 IME663 module front panel controls

The front panel of the IME663 provides a number of controls and indicators, shown in *Figure 1224*.

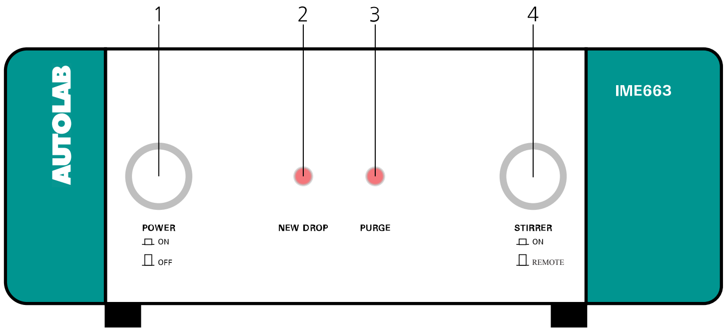


Figure 1224 Overview of the front panel of the IME663

1 Power On/Off button
For switching the IME663 on or off.

2 New drop indicator LED
Flashes when the tapper of the Metrohm 663 VA Stand is activated to create a new drop.

3 Purge LED
Indicates that the nitrogen purge is on when lit.

4 Stirrer On/Remote button
For switching the stirrer on or to software control. When the button is engaged, the stirrer is on. When the button is disengaged, the stirrer is controlled by the software.

16.3.2.15.7 IME663 module back plane connections

The back plane of the IME663 provides a number of connections, shown in *Figure 1225*.

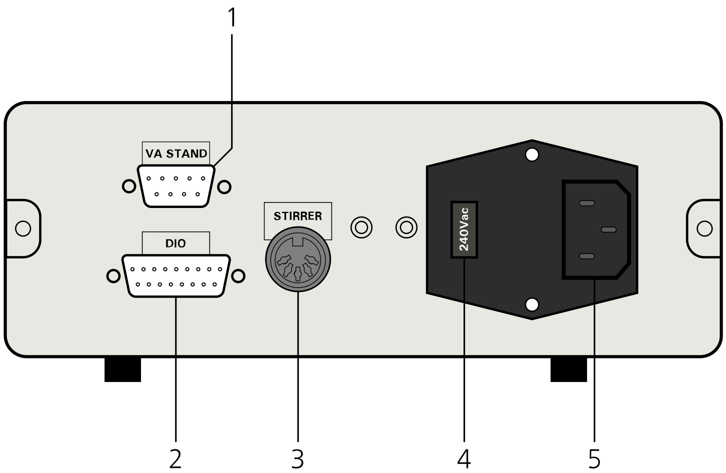


Figure 1225 Overview of the back plane of the IME663

1 VA STAND connector
For connecting the IME663 to the cartridge connector of the Metrohm 663 VA Stand (labeled H) and the ground and instrument front panel (in the case of PGSTAT10, PGSTAT20, µAutolab II and µAutolab III).

2 DIO connector
For connecting the IME663 to the DIO connector of the Autolab.

3 STIRRER connector
For connecting the IME663 to the stirrer connector on the back plane of the Metrohm 663 VA Stand (labeled E).

4 Mains voltage indicator
Indicates the mains voltage settings of the IME663.

5 Mains connection socket
For connecting the IME663 to the mains supply.

CAUTION

Make sure that the mains voltage indicator is set properly before switching the IME663 on.

16.3.2.15.8 Metrohm 663 VA Stand controls

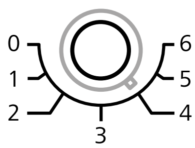
The Metrohm 663 VA Stand has a number of controls located on the cover of the instrument, on the right-hand side of the electrochemical cell.

The following controls are available:

1 stirrer/RDE

This rotating knob controls the rotation rate of the built-in stirrer or rotating disc electrode (if present). Seven positions are available, numbered 0 to 6. Each position corresponds to 500 RPM.

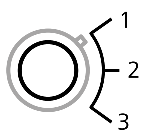
stirrer/RDE



2 drop size

This rotating knob controls the size of the drop formed at the end of the capillary. Three positions are available, numbered 1 to 3. The drop size increases as this control is switched from 1 to 3.

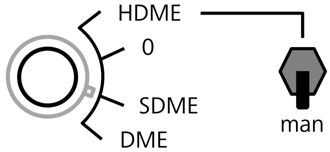
drop size



3 mode selector

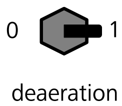
This rotating knob controls the operation mode of the Multi-Mode Electrode (MME) or the Multi-Mode Electrode Pro (MME PRO) installed in the Metrohm 663 VA Stand. The control provides the choice between four positions:

- **HDME:** sets the MME/MME PRO to *hanging mercury drop electrode* mode. When this mode is selected, a drop can be manually created by using the **man** switch located next to the rotary knob. Activating this switch triggers the built-in tapper once.
- **0:** the MME/MME PRO is switched off.
- **SDME:** sets the MME/MME PRO to *static mercury drop electrode* mode. When this mode is selected, the MME/MME PRO can be remotely controlled by NOVA using the IME663.
- **DME:** sets the MME/MME PRO to *dropping mercury electrode* mode.



4 deareation

This switch can be used to manually switch the N_2 purge on or off. This control can be used to overrule the purge control provided through the NOVA software controls.



16.3.2.15.9 IME663 and Metrohm 663 VA Stand installation

The IME663 and Metrohm 663 VA Stand can be used in combination with any compatible instrument. The installation and configuration can be carried out by the end-user at any time.

Depending on the type of instrument it is connected to, the IME663 and Metrohm 663 VA Stand have to be installed according to a specific procedure:

1. For the PGSTAT10, PGSTAT20, µAutolab II and µAutolab III, please refer to *Chapter 16.3.2.15.9.1*.
2. For the PGSTAT101, M101, PGSTAT204 and M204, please refer to *Chapter 16.3.2.15.9.2*.
3. For all other instruments, please refer to *Chapter 16.3.2.15.9.3*.

16.3.2.15.9.1 IME663 and Metrohm 663 VA Stand installation

The following steps describe how to install the IME663 and Metrohm 663 VA Stand in combination with an Autolab PGSTAT. These steps apply to the PGSTAT302N, 302, 30, 128N, 12, 100N and 100.

1 Connect the DIO cable

Connect the DIO cable, supplied with the IME663, to the DIO connector, located on the back plane of the IME663 (item 2 in *Figure 1225*). Connect the other end of the cable to one of the two DIO connectors (P1 or P2) located on the back plane of the Autolab PGSTAT.

2 Connect the Stirrer cable

Connect the Stirrer cable, supplied with the IME663, to the STIRRER connector, located on the back plane of the IME663 (item 3 in *Figure 1225*). Connect the other end of the cable to the E connector located on the back plane of the Metrohm 663 VA Stand.

3 Connect the VA Stand cable

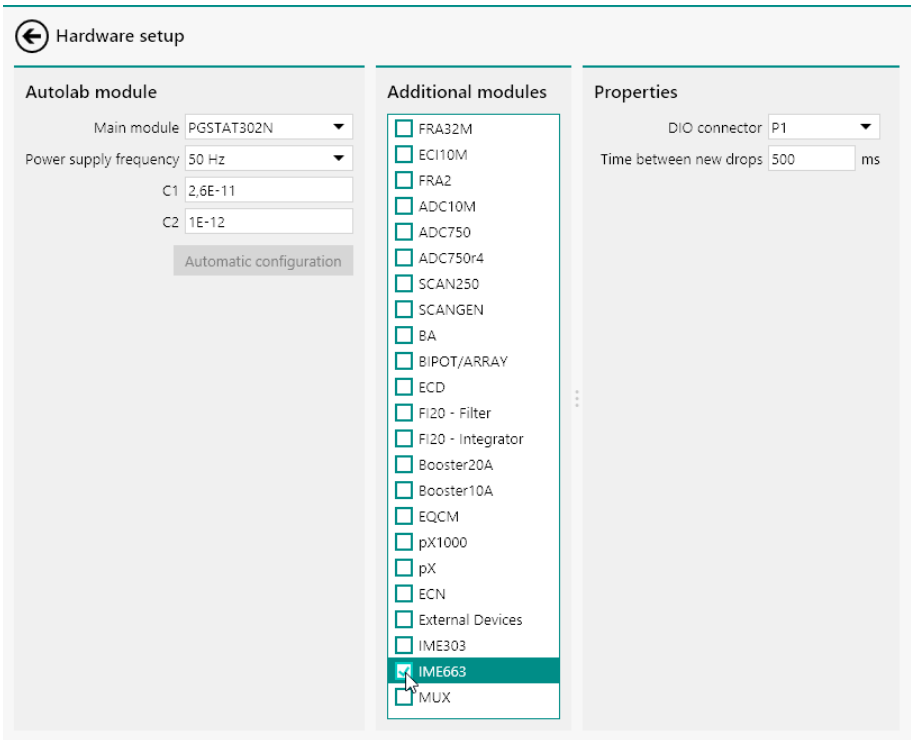
Connect the VA Stand cable, supplied with the IME663, to the VA STAND connector, located on the back plane of the IME663 (item 1 in *Figure 1225*). Connect the other end to the H cartridge connector located on the back plane of the Metrohm 663 VA Stand.

4 Connect the electrodes

Connect the RE, S, WE and CE connectors from the PGSTAT to the matching electrodes installed in the Metrohm 663 VA Stand using the supplied adapter cables.

5 Specify the hardware setup

Adjust the hardware setup and specify the DIO connector (P1 or P2) in the **Properties** panel.



16.3.2.15.9.2 IME663 and Metrohm 663 VA Stand installation (PGSTAT101, M101, PGSTAT204, M204)

The following steps describe how to install the IME663 and Metrohm 663 VA Stand in combination with an Autolab PGSTAT. These steps apply to the PGSTAT101, M101, PGSTAT204 and M204.

1 Connect the DIO cable

Connect the DIO cable, supplied with the IME663, to the DIO connector, located on the back plane of the IME663 (item 2 in *Figure 1225*). Connect the other end of the cable to the DIO connector located on the front panel of the Autolab PGSTAT.

2 Connect the Stirrer cable

Connect the Stirrer cable, supplied with the IME663, to the STIRRER connector, located on the back plane of the IME663 (item 3 in *Figure 1225*). Connect the other end of the cable to the E connector located on the back plane of the Metrohm 663 VA Stand.

3 Connect the VA Stand cable

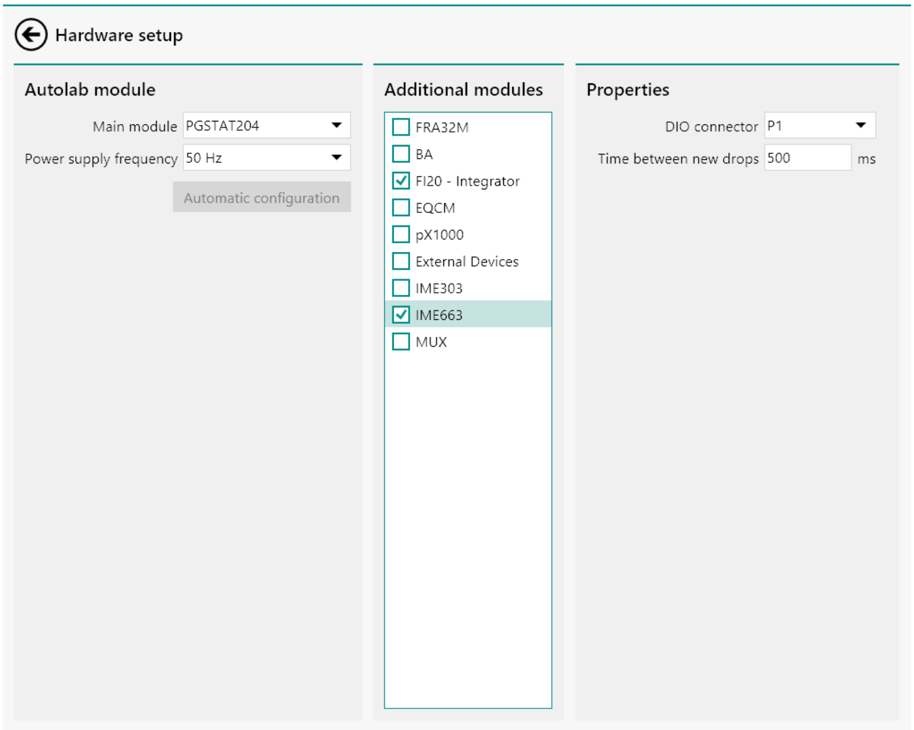
Connect the VA Stand cable, supplied with the IME663, to the VA STAND connector, located on the back plane of the IME663 (item 1 in *Figure 1225*). Connect the other end to the H cartridge connector located on the back plane of the Metrohm 663 VA Stand.

4 Connect the electrodes

Connect the RE, S, WE and CE connectors from the PGSTAT to the matching electrodes installed in the Metrohm 663 VA Stand using the supplied adapter cables.

5 Specify the hardware setup

Adjust the hardware setup.



16.3.2.15.9.3 IME663 and Metrohm 663 VA Stand installation (PGSTAT10, PGSTAT20, µAutolab II, µAutolab III)

The following steps describe how to install the IME663 and Metrohm 663 VA Stand in combination with an Autolab PGSTAT. These steps apply to the PGSTAT10, PGSTAT20, µAutolab II and µAutolab III.

1 Connect the DIO cable

Connect the DIO cable, supplied with the IME663, to the DIO connector, located on the back plane of the IME663 (item 2 in *Figure 1225*). Connect the other end of the cable to one of the two DIO connectors (P1 or P2) located on the back plane of the Autolab PGSTAT.

2 Connect the Stirrer cable

Connect the Stirrer cable, supplied with the IME663, to the STIRRER connector, located on the back plane of the IME663 (item 3 in *Figure 1225*). Connect the other end of the cable to the E connector located on the back plane of the Metrohm 663 VA Stand.

3 Connect the VA Stand cable to the Metrohm 663 VA Stand

Connect the VA Stand cable, supplied with the IME663, to the VA STAND connector, located on the back plane of the IME663 (item 1 in *Figure 1225*). Connect the other end to the H cartridge connector located on the back plane of the Metrohm 663 VA Stand.

4 Remove the cell cable

Remove the cell cable from the front panel of the Autolab instrument.

NOTE

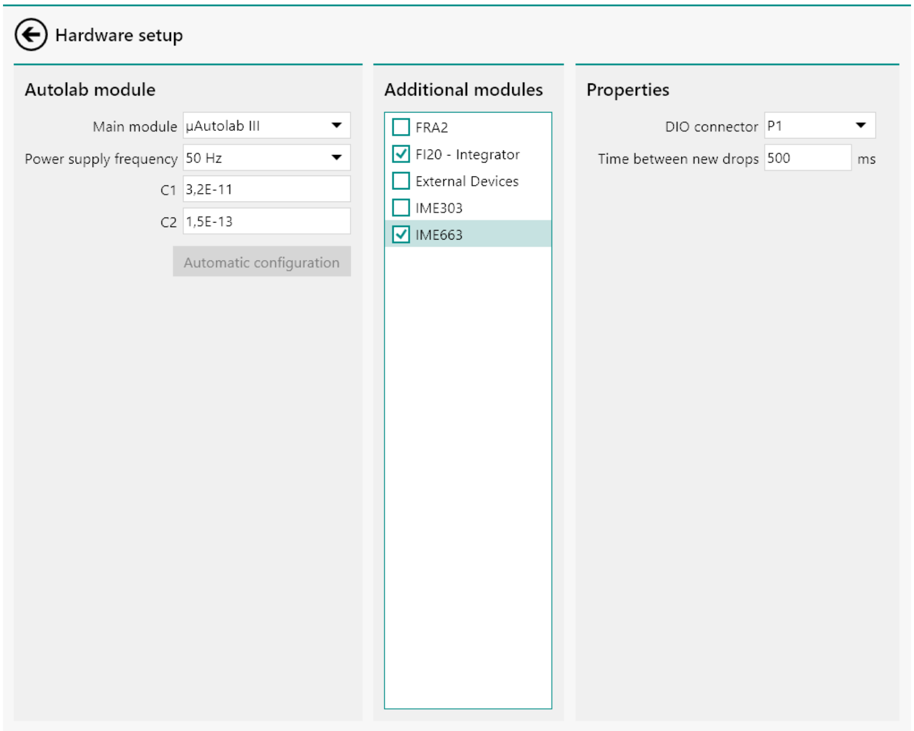
It is recommended to store this cable carefully for future use.

5 Connect the VA Stand cable to the Autolab

Connect the cell cable connector, embedded in the VA Stand cable to the front panel of the Autolab.

6 Specify the hardware setup

Adjust the hardware setup and specify the DIO connector (P1 or P2) in the **Properties** panel.



16.3.2.16 MUX module

The MUX is an optional module for the Autolab PGSTAT and the Multi Autolab. With the MUX module, it is possible to multiplex the PGSTAT electrode connections. Depending on the type of MUX used, measurements on several electrochemical cells or working electrodes can be done sequentially

The MUX module is available in three standard configurations:

- **MUX-MULTI4:** in this configuration, shown in *Figure 1226*, the WE, S, RE and CE leads from the PGSTAT are multiplexed to 4 (or more) sets of connections (see Figure 1). This means that sequential measurements can be performed on multiple independent electrochemical cells. It is possible to add up to 16 MUX-MULTI4 boxes in series in order to multiplex up to 64 independent electrochemical cells.

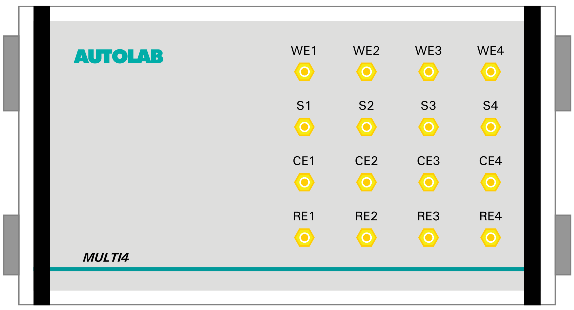


Figure 1226 The MUX-MULTI4 switch box

NOTE

The MUX-MULTI4 is typically used for sequential measurements on individual electrochemical cells or for measurements requiring different electrode arrangements in the same cell (like Van der Pauw measurements).

- **MUX-SCNR8:** in this configuration, shown in *Figure 1227*, the RE and S leads from the PGSTAT are multiplexed to 8 (or more) sets of connections (see Figure 2). This means that the differential amplifier of the PGSTAT can be multiplexed across as many individual cells. It is possible to add up to 16 MUX-SCNR8 boxes in series in order to multiplex up to 128 sets of RE and S electrodes.

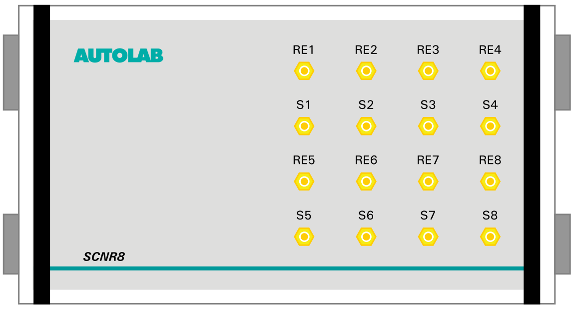


Figure 1227 The MUX-SCNR8 switch box

NOTE

The MUX-SCNR8 is intended to be used for measurements on individual cells in a series of cells. The most common example is the measurement of cell voltages of individual cells or sections of cells in a fuel cell stack.

- **MUX-SCNR16:** in this configuration, shown in *Figure 1228*, the WE lead from the PGSTAT is multiplexed to 16 (or more) WE connections (see Figure 3). This means that sequential measurements can be performed on multiple working electrodes sharing a common reference electrode and counter electrode. It is possible to add up to 16 MUX-SCNR16 boxes in series in order to multiplex up to 255 working electrodes.

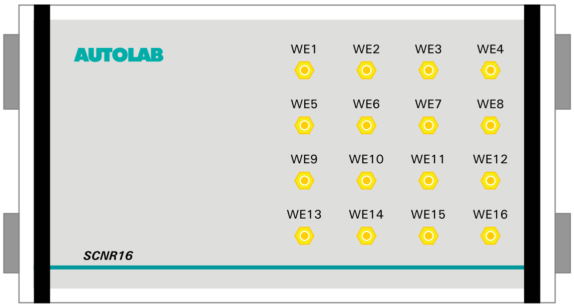


Figure 1228 The MUX-SCNR16 switch box

NOTE

The MUX-SCNR16 is intended to be used for measurements on individual working electrodes contained in a single electrochemical cell. Corrosion and sensor measurements usually benefit from this hardware extension.

CAUTION

Whenever the active channel of the MUX is changed, a 8 ms settling time is added before the next command is executed.

16.3.2.16.1 MUX module compatibility

The MUX module is available for the following instruments:

- PGSTAT302N, PGSTAT302 and PGSTAT30
- PGSTAT128N and PGSTAT12
- M101
- PGSTAT204/M204
- PGSTAT20
- PGSTAT10

NOTE

The MUX module is **not** compatible with the Autolab instruments not listed above.

16.3.2.16.2 MUX module scope of delivery

The MUX module is supplied with the following **common** items:

- MUX module.
- MUX module label.
- Control cable.
- Connection plugs.

Depending on the type of MUX, the following **specific** items are provided:

- For the **MUX-MULTI4**
  - Electrode connection cable for WE, S, CE and RE.
  - MULTI4 switch box.
  - 16 SMB to banana cables (4 labeled WE1 to WE4, 4 labeled S1 to S4, 4 labeled CE1 to CE4 and 4 labeled RE1 to RE4).
  - 16 alligator clips (8 red and 8 black).
- For the **MUX-SCNR8**
  - Electrode connection cable for S and RE.
  - SCNR8 switch box.
  - 16 SMB to banana cables (8 labeled RE1 to RE8 and 8 labeled S1 to S8).
  - 16 alligator clips (8 red and 8 black).
- For the **MUX-SCNR16**
  - Electrode connection cable for WE.
  - SCNR16 switch box.
  - 16 SMB to banana cables (labeled WE1 to WE16).
  - 16 red alligator clips.

16.3.2.16.3 MUX hardware setup

To use the **MUX** module, the hardware setup needs to be adjusted. The checkbox for the module needs to be ticked *(see Figure 1229, page 1125)*.

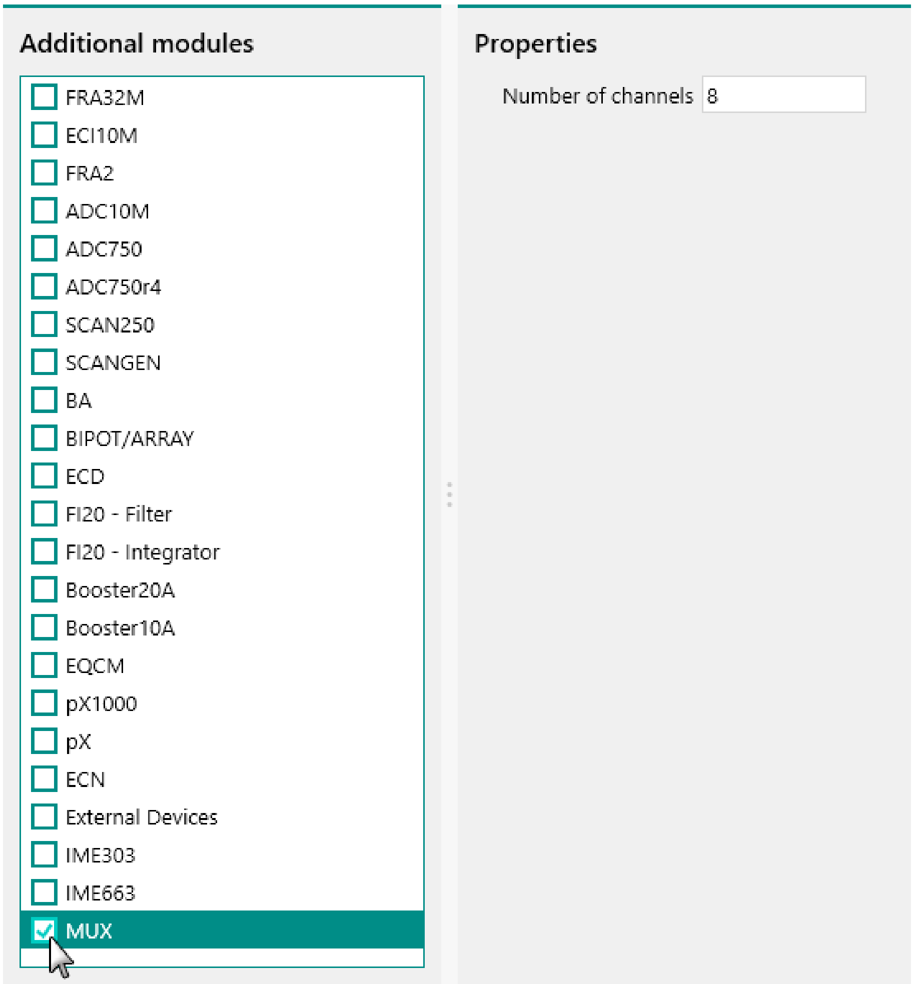


Figure 1229 The MUX module is selected in the hardware setup

The *Number of channels* properties located in the **Properties** panel can be used to specify the maximum number of channels connected to the **MUX** module *(see Figure 1229, page 1125)*.

16.3.2.16.4 MUX module settings

The MUX module settings are completely defined in the NOVA software. The following user-definable setting is available, through the **Autolab control** command *(see Figure 1230, page 1126)*:

- **Channel:** this control can be used to specify the active MUX channel. The number of available channels is defined in the hardware setup.

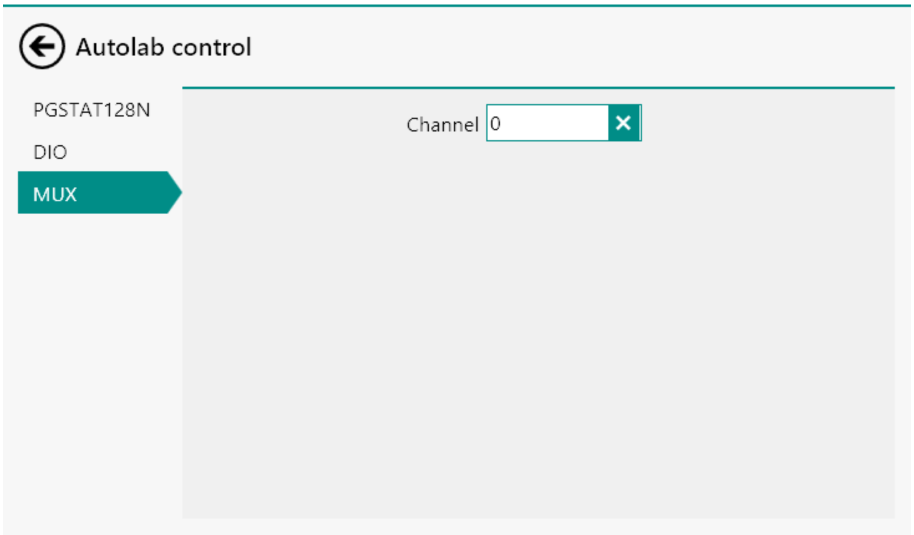


Figure 1230 The MUX module settings

NOTE

When the Channel is set to 0, no MUX channel is selected and the MUX is bypassed.

16.3.2.16.5 MUX module manual control

The MUX can be manually controlled, using the **Autolab display** provided in the instrument control panel *(see Figure 1231, page 1126)*. The dedicated manual control panel can be used to perform the following tasks:

- **Active channel:** this control can be used to define the active channel of the MUX. A value between 0 and the number of channels defined in the hardware setup can be specified.

Figure 1231 Manual control of the MUX provided in the Autolab display

The value can be specified directly as an integer of through the provided slider control *(see Figure 1232, page 1127)*.

Figure 1232 The active channel can be directly specified in the Autolab display

NOTE

When the Channel is set to 0, no MUX channel is selected and the MUX is bypassed.

NOTE

The manual control panel provided for the MUX can be used to set or display the current settings of the MUX.

16.3.2.16.6 MUX module restrictions

Restrictions apply when using the MUX module:

- **Maximum current:** the relays located in the MUX switchboxes are not suitable for currents higher than 5 A. This means that the MUX must not be used in combination with the Booster10A or the Booster20A.
- **Switching time:** the relays located in the MUX switchboxes are mechanical. This means that the time required to open or close the relays is in the range of a few ms.
- **Identical boxes:** when more than one MUX switch box is used in a daisy chain arrangement, the MUX switch boxes must be of the same type.

16.3.2.16.7 MUX module front panel connections

The MUX module is fitted with a single female, 15 pin SUB-D connector. This connector is used to connect the digital control cable *(see Figure 1233, page 1128)*.

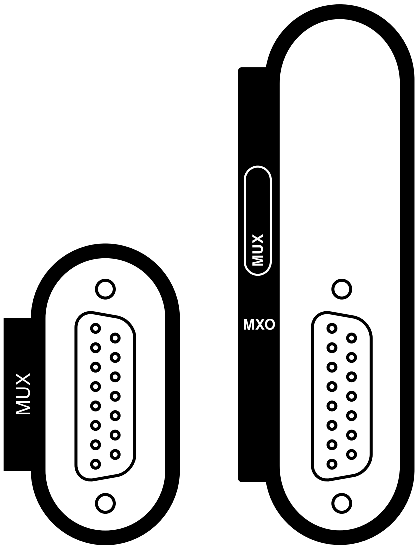


Figure 1233 The front panel labels of the MUX module (left: MUX module in PGSTAT, right: MUX module in Multi Autolab)

The MUX-MULTI4 switch box has the following connections *(see Figure 1234, page 1128)*:

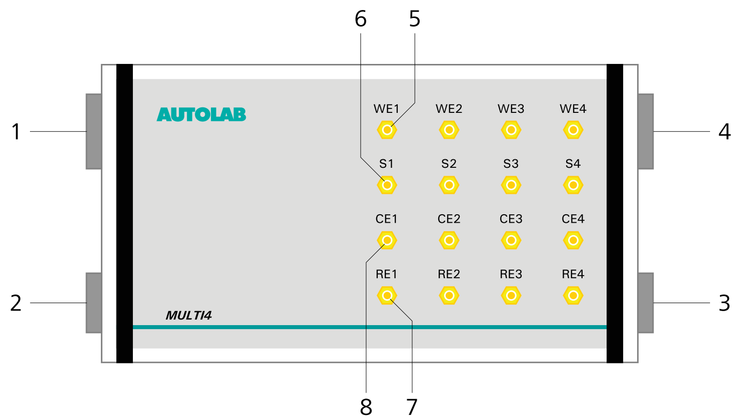


Figure 1234 The connections provided by the MUX-MULTI4 switch box

1	**Digital control connector** A 15 pin male SUB-D connector used to connect the digital control cable from the MUX module.	**2**	**Electrode connection cable connector** A 9 pin male SUB-D connector used to connect the electrode connection cable providing inputs for the WE, S, CE and RE from the Autolab PGSTAT.
3	**Digital control extension connector** A 15 pin female SUB-D connector used to extend the digital control of the MUX to the adjacent switch box.	**4**	**Electrode connection cable extension connector** A 9 pin female SUB-D connector used to extend the electrode connections from the Autolab PGSTAT to the adjacent switch box.

5 WE1 to WE4 connections
Output connections for attaching the WE1 to WE4 connections cables provided with the MULTI4 switch box.

6 S1 to S4 connections
Output connections for attaching the S1 to S4 connections cables provided with the MULTI4 switch box.

7 CE1 to CE4 connections
Output connections for attaching the CE1 to CE4 connections cables provided with the MULTI4 switch box.

8 RE1 to RE4 connections
Output connections for attaching the RE1 to RE4 connections cables provided with the MULTI4 switch box.

The MUX-SCNR8 switch box has the following connections *(see Figure 1235, page 1129)*:

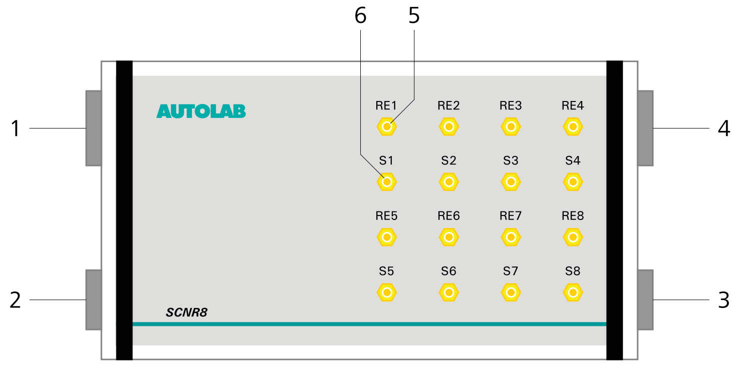


Figure 1235 The connections provided by the MUX-SCNR8 switch box

1 Digital control connector
A 15 pin male SUB-D connector used to connect the digital control cable from the MUX module.

2 Electrode connection cable connector
A 9 pin male SUB-D connector used to connect the electrode connection cable providing inputs for the RE and S from the Autolab PGSTAT.

3 Digital control extension connector
A 15 pin female SUB-D connector used to extend the digital control of the MUX to the adjacent switch box.

4 Electrode connection cable extension connector
A 9 pin female SUB-D connector used to extend the electrode connections from the Autolab PGSTAT to the adjacent switch box.

5 RE1 to RE8 connections
Output connections for attaching the RE1 to RE8 connections cables provided with the SCNR8 switch box.

6 S1 to S8 connections
Output connections for attaching the S1 to S8 connections cables provided with the SCNR8 switch box.

The MUX-SCNR16 switch box has the following connections *(see Figure 1236, page 1130)*:

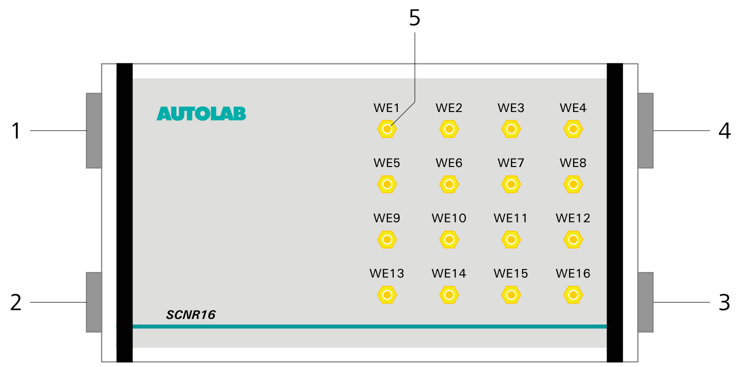


Figure 1236 The connections provided by the MUX-SCNR16 switch box

1 Digital control connector
A 15 pin male SUB-D connector used to connect the digital control cable from the MUX module.

2 Electrode connection cable connector
A 9 pin male SUB-D connector used to connect the electrode connection cable providing inputs for the WE from the Autolab PGSTAT.

3 Digital control extension connector
A 15 pin female SUB-D connector used to extend the digital control of the MUX to the adjacent switch box.

4 Electrode connection cable extension connector
A 9 pin female SUB-D connector used to extend the electrode connections from the Autolab PGSTAT to the adjacent switch box.

5 WE1 to WE16 connections
Output connections for attaching the WE1 to WE16 connections cables provided with the SCNR16 switch box.

16.3.2.16.8 MUX module testing

Two test procedures are provided for testing the MUX module:

- For the MULTI4 and the SCNR16 module option, please refer to *Chapter 16.3.2.16.8.1*.
- For the SCNR8 module option, please refer to *Chapter 16.3.2.16.8.2*.

16.3.2.16.8.1 MUX-SCNR16 and MUX-MULTI4 module testing

NOVA is shipped with a procedure which can be used to verify that the MUX module in combination with the **SCNR16** and the **MULTI4** option is working as expected.

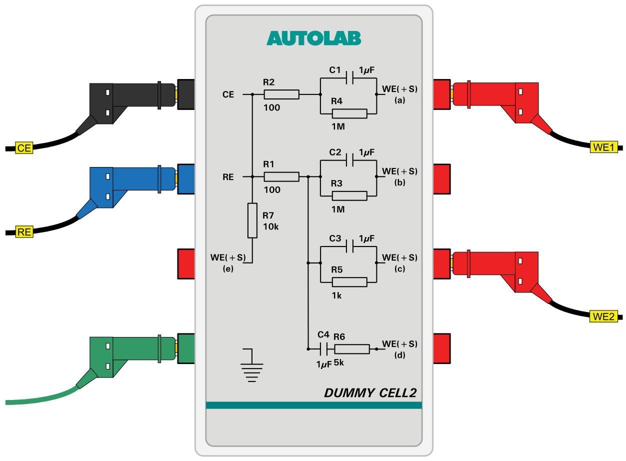
Follow the steps described below to run the test procedure.

1 Load the procedure

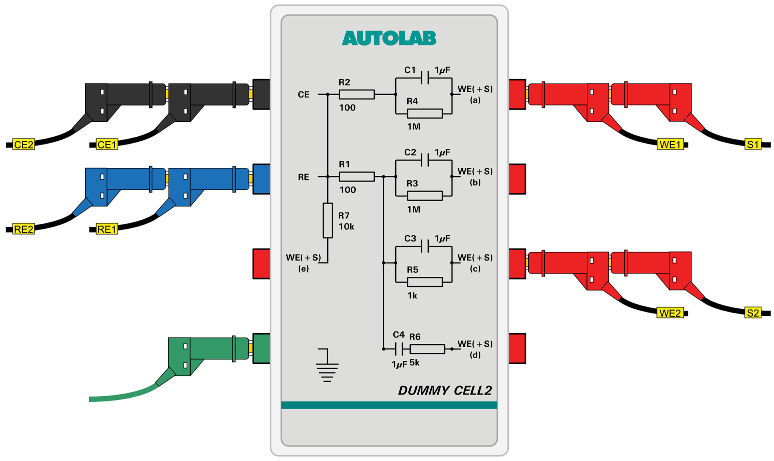
Load the **TestMUX-SCNR16-MULTI4** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test\TestMUX-SCNR16-MULTI4.nox)

2 Connect the Autolab dummy cell

For the SCNR16 option, connect the MUX channel 1 and 2 to the Autolab dummy cell circuit (a) and (c).



For the MULTI4 option, connect the MUX channel 1 and 2 to the Autolab dummy cell circuit (a) and (c).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1237*.

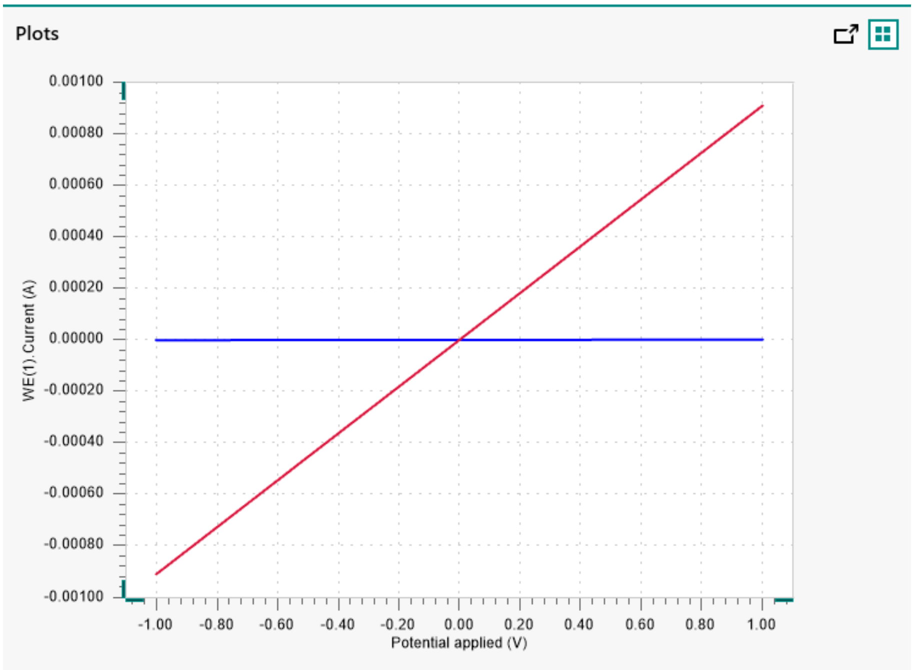
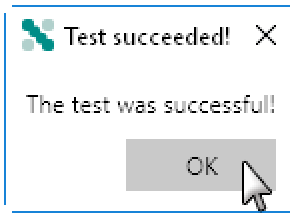


Figure 1237 The results of the TestMUX-SCNR16-MULTI4 procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestMUX-SCNR16-MULTI4 automatic evaluation of the data requires the following tests to succeed:

1. The inverted slope of the measured current versus the applied potential must be equal to 1000100 Ω± 5 % for channel 1 and must be equal to 1100 Ω± 5 %. for channel 2.
2. The intercept of the measured current versus the applied potential must be equal to ± 4 mV divided by 1000100 Ω± 5 % for channel 1 and must be equal to ± 4 mV divided by 1100 Ω± 5 % for channel 2.

All four conditions must be valid for the test to succeed.

16.3.2.16.8.2 MUX-SCNR8 module testing

NOVA is shipped with a procedure which can be used to verify that the MUX module in combination with the **SCNR8** option is working as expected.

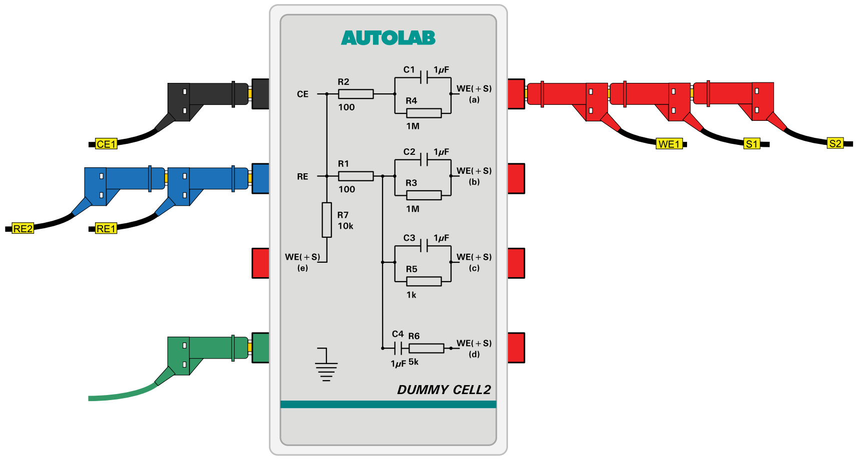
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestMUX-SCNR8** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases \Module test\TestMUX-SCNR8.nox)

2 Connect the Autolab dummy cell

Connect the MUX channel 1 and 2 to the Autolab dummy cell circuit (a).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1238*.

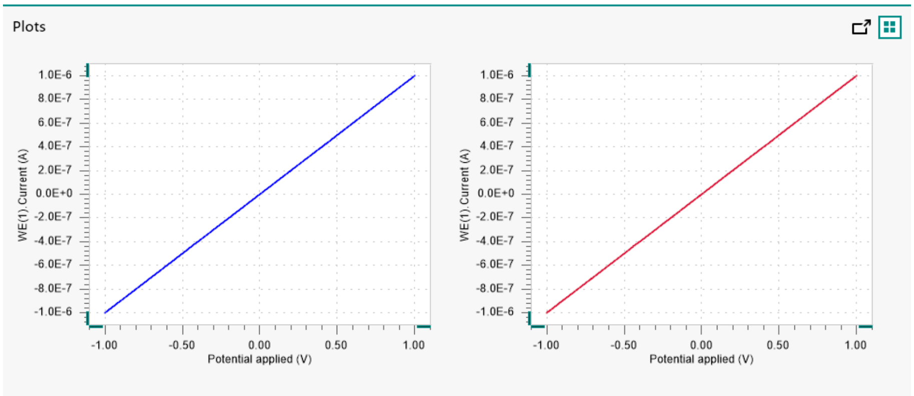
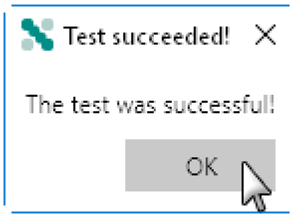


Figure 1238 The results of the TestMUX-SCNR8 procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestMUX-SCNR8 automatic evaluation of the data requires the following tests to succeed:

1. The inverted slope of the measured current versus the applied potential must be equal to 1000100 Ω± 5 % for both channels.
2. The intercept of the measured current versus the applied potential must be equal to ± 4 mV divided by 1000100 Ω± 5 % for both channels.

All four conditions must be valid for the test to succeed.

16.3.2.16.9 MUX module specifications

The MUX module is available in three versions:

- **MUX-MULTI4:** can be used to multiplex all four electrodes provided by the Autolab: CE, RE, S and WE. Using this module, it is possible to work on different electrochemical cells, sequentially.
- **MUX-SCNR8:** can be used to multiplex two independent electrodes provided by the Autolab. The most common use of this special module is to multiplex the S and RE connection from the differential amplifier in order to measure the potential difference across several electrochemical interfaces, sequentially.

- **MUX-SCNR16:** can be used to multiplex a single electrode provided by the Autolab. The most common use of this module is to multiplex the S+WE in order to perform electrochemical measurements on different working electrodes located in the same cell, sharing the counter and reference electrode. The measurements are performed sequentially.

The specifications of the MUX module are provided in *Table 53*.

Table 53 Specifications of the MUX module

Module	MULTI4	SCNR8	SCNR16
Number of channels	4-64	16-128	16-255
Increment per extension	4	8	16
Switching time	5 ms	5 ms	5 ms
Maximum current	2 A	2 A	2 A

NOTE

Inactive MUX channels are kept at open circuit potential at all times.

CAUTION

The MUX modules cannot be used in combination with the Autolab PGSTAT100N or PGSTAT100 and the PGSTAT302F. It is also not possible to use the MUX modules in combination with the Booster10A or the Booster20A.

16.3.2.17 pX module

The pX module is an extension module for the Autolab potentiostat/galvanostat. This module consists of a high input impedance differential electrometer that can be used to measure the p value of an indicator electrode, typically the pH. This differential electrometer can also be used as a voltmeter to measure a potential difference.

This module is compatible with all Metrohm sensors fitted with the F type Lemo connector as well as with a male BNC plug.

NOTE

The pX module is no longer available and it is now replaced by its successor module, the pX1000.

The pX module adds the following signals to the signal sampler *(see Figure 1239, page 1136)*:

- **pX.Voltage (V):** this signal corresponds to the potential difference measured by the pX module.
- **pX.pH:** this signal corresponds to the converted pH value, determined based on the voltage value measured by the pX.

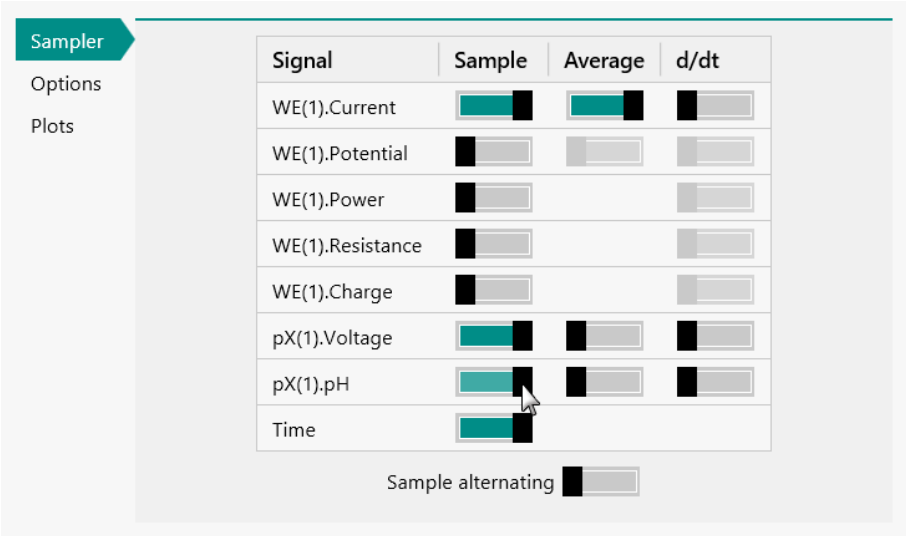


Figure 1239 The pX module provides the pX(1).Voltage and pX(1).pH signals

16.3.2.17.1 pX module compatibility

The pX module is available for the following instruments:

- PGSTAT302N, PGSTAT302 and PGSTAT30
- PGSTAT128N and PGSTAT12
- PGSTAT100N and PGSTAT100
- PGSTAT20
- PGSTAT10

NOTE

The pX module is **not** compatible with the Autolab instruments not listed above.

16.3.2.17.2 pX module scope of delivery

The pX module is supplied with the following items:

- pX module
- pX module label
- BNC connector cable
- 50 Ohm BNC plug
- Lemo to BNC adapter plug

16.3.2.17.3 pX hardware setup

To use the **pX** module, the hardware setup needs to be adjusted. The checkbox for the module needs to be ticked *(see Figure 1240, page 1137)*.

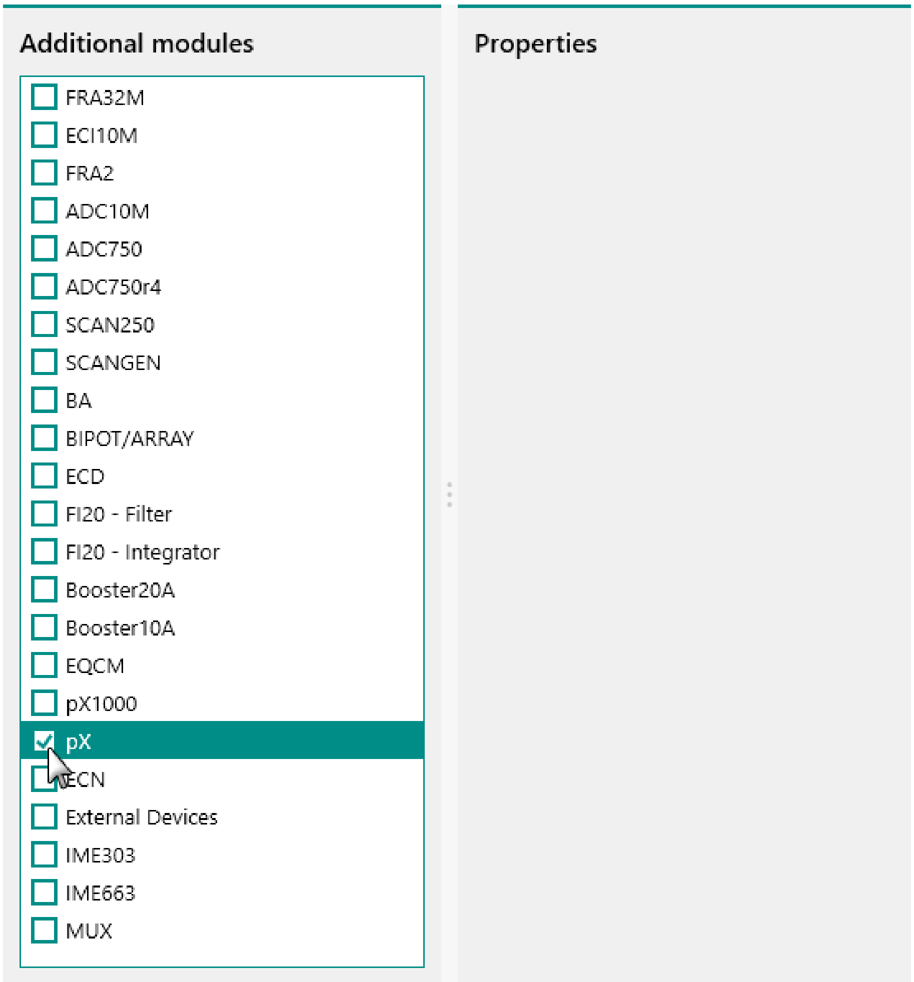


Figure 1240 The pX module is selected in the hardware setup

16.3.2.17.4 pX module settings

The pX module settings are controlled using the parts supplied with the module. When the sensor connected to the pX module is used outside of the cell containing the working electrode, a 50 Ohm BNC resistor must be connected to the ⊙ G BNC connector located on the pX module, above the → E input plug. This resistor is used to ground the pX module.

When the sensor connected to the pX module is located inside the electrochemical cell, the 50 Ohm BNC resistor must be removed.

NOTE

If the values measured through the pX module are unusually noisy, please verify that you are using the correct configuration.

CAUTION

When the sensor connected to the pX module is located in the electrochemical cell, the potential difference measured by the module during a measurement will be affected by the changes in electrical field in the solution generated by the polarization of the electrodes in the cell. Depending on several experimental factors like temperature, conductivity, ionic strength, cell and electrode geometry, etc., the effect on the potential difference measured by the pX module can be more or less pronounced. Trial and error may be required to determine the experimental conditions required to minimize these effects.

16.3.2.17.5 pX module restrictions

No restrictions apply when using the pX module.

16.3.2.17.6 pX module front panel connections

The pX module is fitted with two female BNC connectors, labeled ⊙G and →E, respectively *(see Figure 1241, page 1138)*.

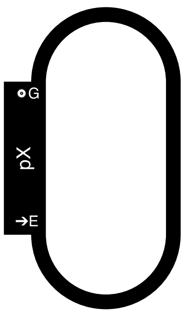


Figure 1241 The front panel label of the pX module

Both connectors located on the front panel of the pX module are input connectors. The ⊙G connector is used ground the input circuit of the pX module, using the provided 50 Ω plug. The →E is used to connect the pH sensor electrode. A BNC to LEMO converter is supplied with the module.

NOTE

The pX module must be grounded when the pH sensor is not located in the same cell as the working electrode.

16.3.2.17.7 pX module testing

NOVA is shipped with a procedure which can be used to verify that the **pX** module is working as expected.

CAUTION

The pX module must be grounded using the provided 50 Ω resistor plug.

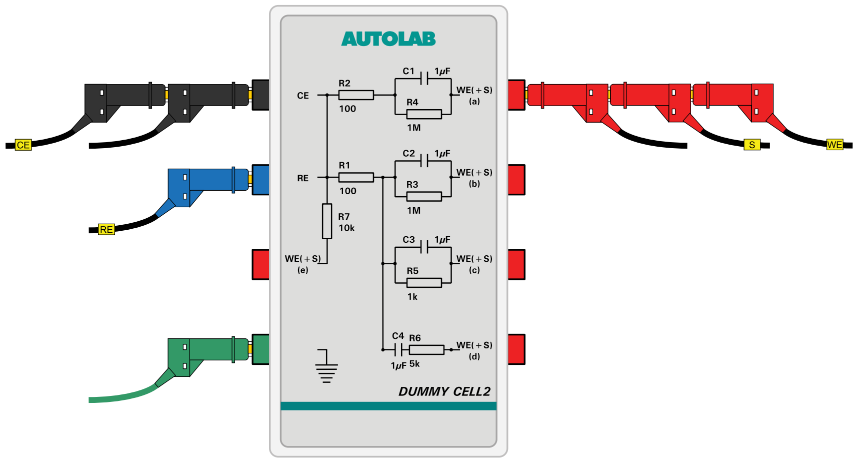
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestpX** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test \TestpX.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (a) and the pX module to the Autolab dummy cell (a).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1242*.

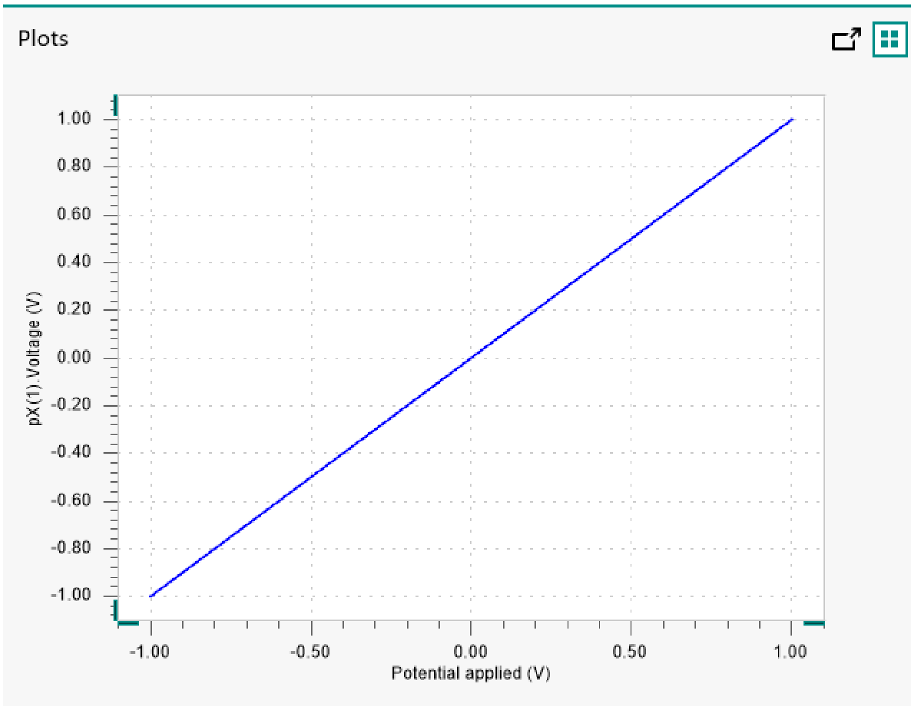
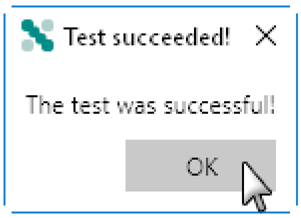


Figure 1242 The results of the TestpX procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestpX automatic evaluation of the data requires the following tests to succeed:

1. The inverted slope of the measured pX(1).Potential versus the applied potential must be equal to 1 ± 5 %.
2. The intercept of the measured pX(1).Potential versus the applied potential must be equal to ± 0.004.

Both conditions must be valid for the test to succeed.

16.3.2.17.8 pX module specifications

The specifications of the pX module are provided in *Table 54*.

Table 54 Specifications of the pX module

Specification	Value
Input impedance	> 100 GΩ
Input range	± 2.5 V
Module configuration	Hardware control
Pt1000/NTC temperature sensor	No
Potential resolution	7.6 μV (gain 10)
Potential accuracy	± 300 μV

16.3.2.18 pX1000 module

The pX1000 module is an extension module for the Autolab PGSTAT and the Multi Autolab. This module consists of a high input impedance differential electrometer that can be used to measure the p value of an indicator electrode, typically the pH. This differential electrometer can also be used as a voltmeter to measure a potential difference.

This module is compatible with all Metrohm sensors fitted with the F type Lemo connector.

The pX1000 module also provides input for a Pt1000 or NTC temperature sensor.

The pX1000 module adds the following signals to the signal sampler *(see Figure 1243, page 1142)*:

- **pX1000.Temperature (°C):** this signal corresponds to the temperature measured by the pX1000 module.
- **pX1000.Voltage (V):** this signal corresponds to the potential difference measured by the pX1000 module.
- **pX1000.pH:** this signal corresponds to the converted pH value, determined based on the voltage value measured by the pX1000.

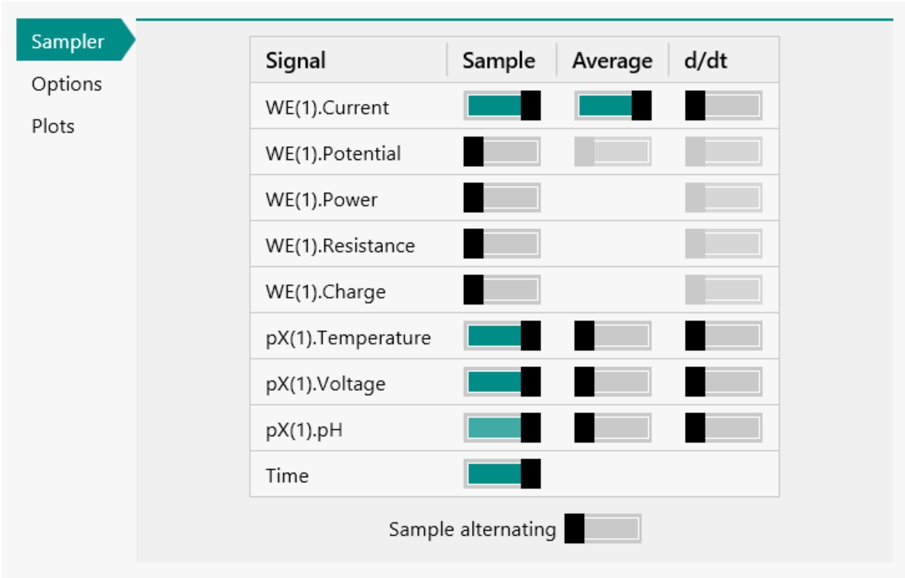


Figure 1243 The pX1000 module provides the pX1000(1).Temperature, pX1000(1).Voltage and pX1000(1).pH signals

16.3.2.18.1 pX1000 module compatibility

The pX1000 module is available for the following instruments:

- PGSTAT302N, PGSTAT302 and PGSTAT30
- PGSTAT128N and PGSTAT12
- PGSTAT100N and PGSTAT100
- M101
- PGSTAT204/M204

NOTE

The pX1000 module is **not** compatible with the Autolab instruments not listed above.

16.3.2.18.2 pX1000 module scope of delivery

The pX1000 module is supplied with the following items:

- pX1000 module
- pX1000 module label
- V+ connection cable
- V- connection cable
- Lemo connector cover

16.3.2.18.3 pX1000 hardware setup

To use the **pX1000** module, the hardware setup needs to be adjusted. The checkbox for the module needs to be ticked *(see Figure 1244, page 1143)*.

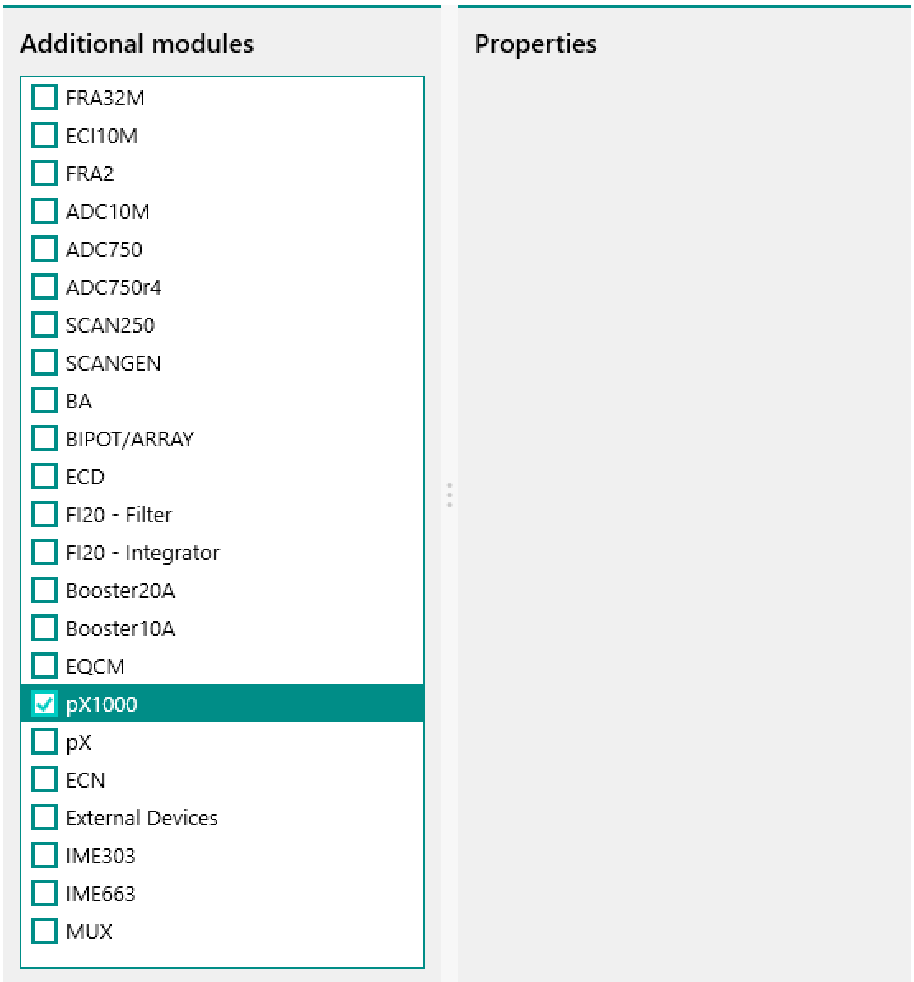


Figure 1244 The pX1000 module is selected in the hardware setup

16.3.2.18.4 pX1000 module settings

The pX1000 module settings are completely defined in the NOVA software. The following user-definable settings are available, through the **Autolab control** command *(see Figure 1245, page 1144)*:

- **Mode:** this drop-down control can be used to define whether a single input configuration (pH (single input)) or a differential input configuration (pX (differential input)) will be used in the measurements. A single input configuration is usually used in pH measurements, while a differential input configuration is required when the pX1000 is used in combination with two separate electrodes or as an additional differential electrometer.

- **Cell setup:** this drop-down control can be used to define the location of the sensor connected to the pX1000 module. In the Autolab potentiostat, the working electrode is connected to virtual ground. When the sensor connected to the pX1000 module is located in the same cell as the working electrode, the inputs of the pX1000 must be floating, to avoid leakage or ground loops.

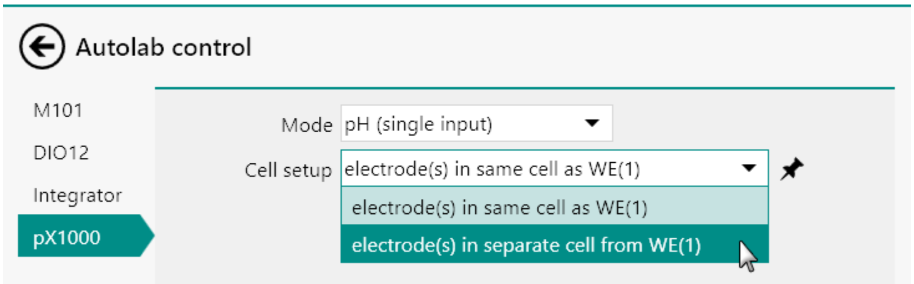


Figure 1245 The pX1000 module settings

NOTE

If the values measured through the pX1000 module are unusually noisy, please verify that you are using the correct configuration.

CAUTION

When the sensor connected to the pX1000 module is located in the electrochemical cell, the potential difference measured by the module during a measurement will be affected by the changes in electrical field in the solution generated by the polarization of the electrodes in the cell. Depending on several experimental factors like temperature, conductivity, ionic strength, cell and electrode geometry, etc., the effect on the potential difference measured by the pX1000 module can be more or less pronounced. Trial and error may be required to determine the experimental conditions required to minimize these effects.

16.3.2.18.5 pX1000 module restrictions

No restrictions apply when using the pX1000 module.

16.3.2.18.6 pX1000 module front panel connections

The pX1000 module is fitted with two LEMO connectors, labeled →pH/V+ and →V-, respectively and two 1 mm female banana connectors, labeled T, respectively red and black *(see Figure 1246, page 1145)*.

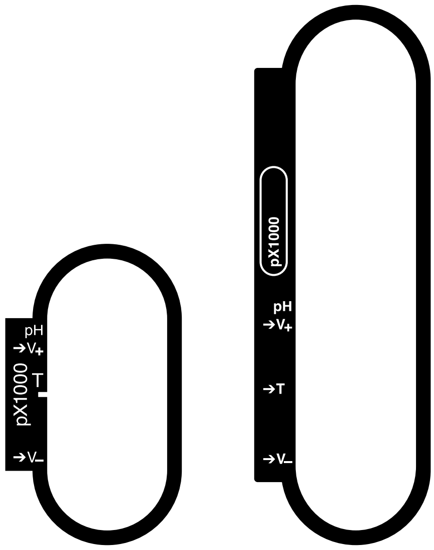


Figure 1246 The front panel labels of the pX1000 module (left: pX1000 module in PGSTAT, right: pX1000 module in Multi Autolab)

All four connectors located on the front panel of the pX1000 module are input connectors. The →pH/V+ connector is used to connect combined pH sensors, while the →pH/V+ and →V- connectors are used to connect pH sensors with a separate reference electrode or for differential potential measurements. The two temperature inputs are used to connect a compatible temperature sensor.

NOTE

The pX1000 module is compatible with any temperature sensor that complies with the IEC751 Standard.

16.3.2.18.7 pX1000 module testing

NOVA is shipped with a procedure which can be used to verify that the **pX1000** module is working as expected.

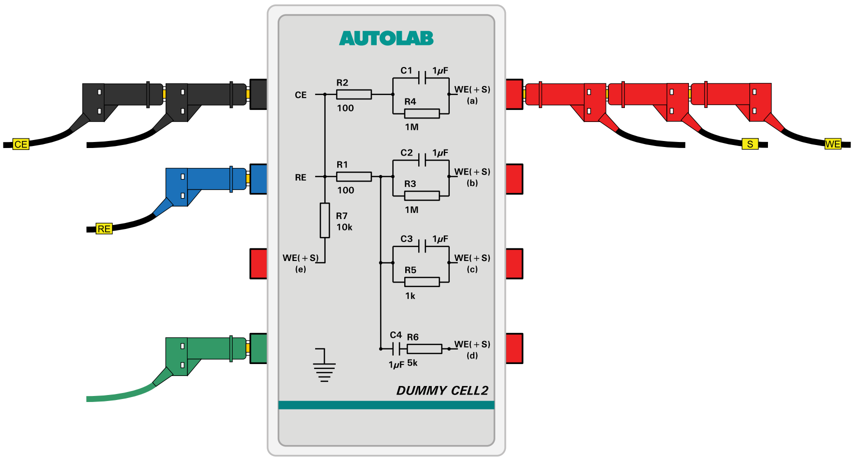
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestpX1000** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test\TestpX1000.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (a) and the pX1000 module to the Autolab dummy cell (a).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1247*.

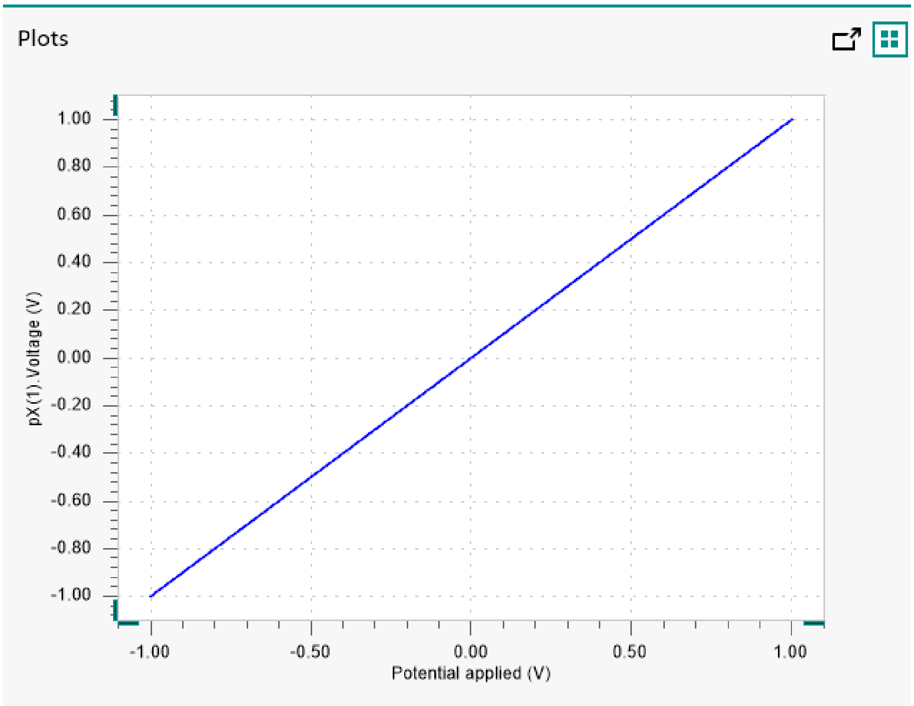
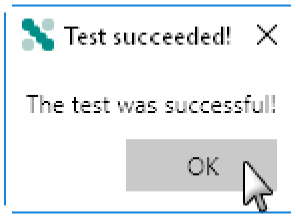


Figure 1247 The results of the TestpX1000 procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestpX1000 automatic evaluation of the data requires the following tests to succeed:

1. The inverted slope of the measured pX(1).Potential versus the applied potential must be equal to 1 ± 5 %.
2. The intercept of the measured pX(1).Potential versus the applied potential must be equal to ± 0.004.

Both conditions must be valid for the test to succeed.

16.3.2.18.8 pX1000 module specifications

The specifications of the pX1000 module are provided in *Table 55*.

Table 55 Specifications of the pX1000 module

Specification	Value
Input impedance	> 1000 GΩ
Input range	± 10 V
Module configuration	Software control
Pt1000/NTC temperature sensor	Yes
Potential resolution	30 µV (gain 10)
Potential accuracy	± 2 mV
Temperature resolution	0.015 °C
Temperature accuracy	± 0.5 °C

16.3.2.19 SCAN250 module

The SCAN250 module is an extension module for the Autolab PGSTAT. This module provides is a true linear scan generator which can be used to perform linear scan cyclic voltammetry measurements. Linear scan cyclic voltammetry measurement use a perfectly smooth potential scan, defined only by a scan range, in mV and a scan range, in V/s. Unlike the cyclic voltammetry staircase method, the sampling of the electrochemical signals is decoupled from the scan generation.

Using the SCAN250 module it is possible to measure both the faradic and the capacitive currents during a cyclic voltammetry. Therefore, this particular form of cyclic voltammetry is well suited for monitoring changes in the double layer structure or electrochemical reactions characterized by fast electron transfer kinetics.

NOTE

The SCAN250 can generate a potential scan between 10 mV/s and 250 kV/s.

Depending on the hardware configuration, the SCAN250 can be used in two different scan rate ranges:

- **Without the ADC10M or the ADC750 module:** if the instrument is **not** fitted with the ADC10M and ADC750 module it is possible to use the SCAN250 module up to 250 V/s.

- **With the ADC10M or ADC750 module:** if the instrument is fitted with the ADC10M *(see Chapter 16.3.2.1, page 977)* or the ADC750 *(see Chapter 16.3.2.2, page 983)* module it is possible to use the SCAN250 module up to 250 kV/s.

NOTE

The SCAN250 module only works in potentiostatic mode.

16.3.2.19.1 SCAN250 module compatibility

The SCAN250 module is available for the following instruments:

- PGSTAT302N, PGSTAT302 and PGSTAT30
- PGSTAT128N and PGSTAT12
- PGSTAT100N and PGSTAT100

NOTE

The SCAN250 module is **not** compatible with the Autolab instruments not listed above.

16.3.2.19.2 SCAN250 scope of delivery

The SCAN250 module is supplied with the following items:

- SCAN250 module
- SCAN250 module label

16.3.2.19.3 SCAN250 hardware setup

To use the **SCAN250** module, the hardware setup needs to be adjusted. The checkbox for the module needs to be ticked *(see Figure 1248, page 1150)*.

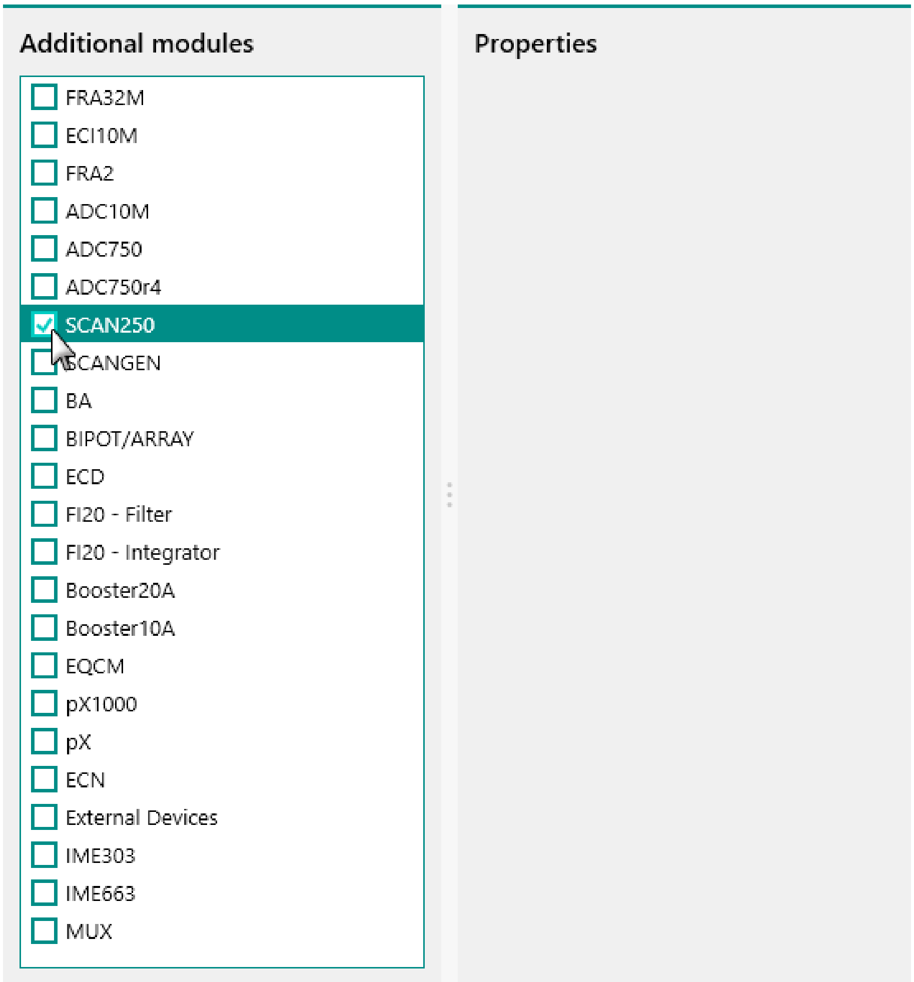


Figure 1248 The SCAN250 module is selected in the hardware setup

16.3.2.19.4 SCAN250 module settings

The SCAN250 module does not have any user-definable settings. All the module settings are directly controlled through the **CV linear scan** command.

16.3.2.19.5 SCAN250 module restrictions

Restrictions apply when using the SCAN250 module:

- **Potentiostatic control only:** the SCAN250 module can only be used in Potentiostatic mode.
- **Vertex overshoot or undershoot:** The SCAN250 module is fitted with a vertex detection circuit which is used to synchronize the reversal of the scan direction with the detection of the vertex. At very high scan rates, this circuit can cause the linear scan generator module to stop the scan at a potential value that no longer matches the experimental parameters. This potential difference depends on the scan rate.

16.3.2.19.6 SCAN250 module front panel connections

The SCAN250 module is fitted with a single female BNC connector, labeled ←V *(see Figure 1249, page 1151)*.

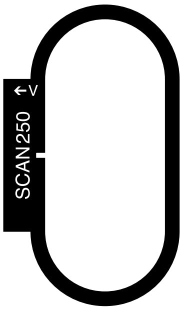


Figure 1249 The front panel label of the SCAN250 module

The signal provided through the ←V connector on the front panel corresponds to the output of scan generator signal of the SCAN250 module. The output signal is a voltage, referred to the instrument ground, corresponding to the specified potential scan.

NOTE

The output of the SCAN250 module does not include the offset potential which is created by the DAC164 module of the Autolab potentiostat/galvanostat.

NOTE

The front panel ←V BNC output is provided for information purposes only.

16.3.2.19.7 SCAN250 module test

NOVA is shipped with a procedure which can be used to verify that the integrator circuit of the **SCAN250** module is working as expected.

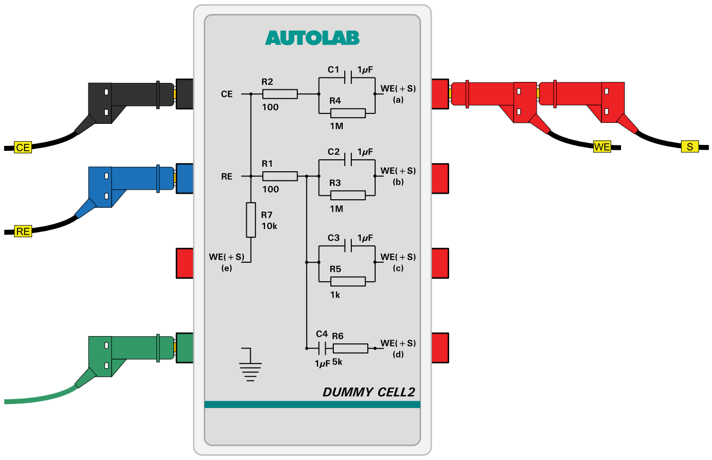
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestSCAN** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test\TestSCAN.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (a).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1250*.

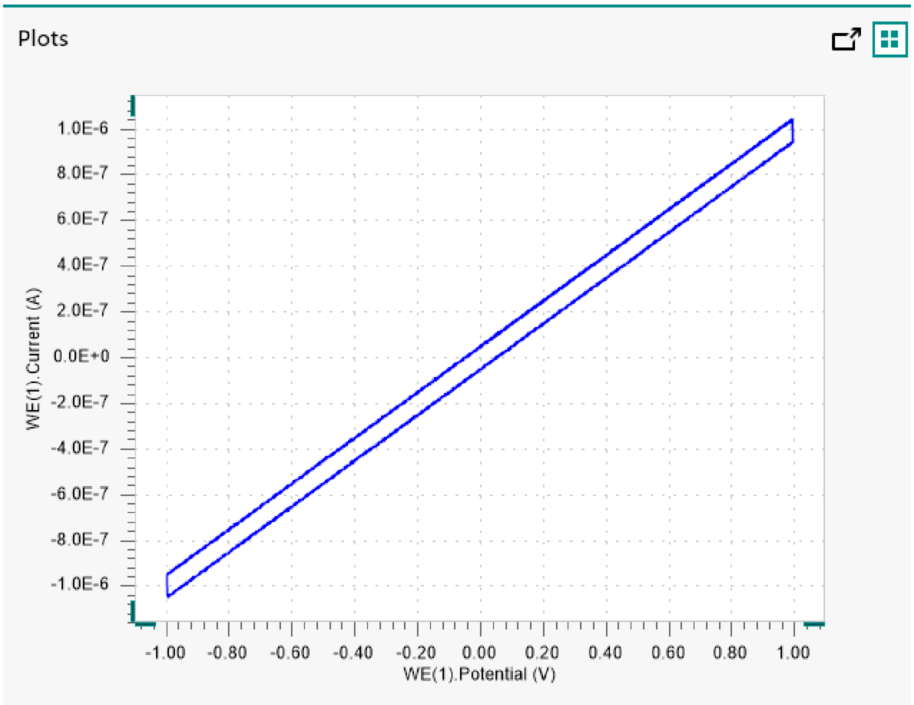
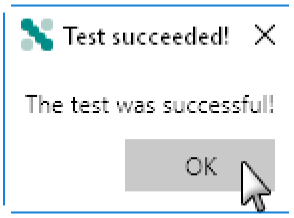


Figure 1250 The results of the TestSCAN procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestSCAN automatic evaluation of the data requires the following tests to succeed:

1. The inverted slope of the measured current versus the applied potential must be equal to 1000100 Ω± 5 %.
2. The calculated capacitance, determined from the integrated current signal, must be equal to 1 μF ± 10 %.

Both conditions must be valid for the test to succeed.

16.3.2.19.8 SCAN250 module specifications

The specifications of the SCAN250 module are provided in *Figure 56*.

Table 56 Specifications of the SCAN250 module

Specification	Value
Scan range	± 5 V
Vertex resolution	0.15 mV
Vertex accuracy	± 2 mV
Output offset	± 0.2 mV
Minimum scan rate	10 mV/s
Maximum scan rate	250 kV/s

16.3.2.20 SCANGEN module

The SCANGEN module is an extension module for the Autolab PGSTAT. This module provides is a true linear scan generator which can be used to perform linear scan cyclic voltammetry measurements. Linear scan cyclic voltammetry measurement use a perfectly smooth potential scan, defined only by a scan range, in mV and a scan range, in V/s. Unlike the cyclic voltammetry staircase method, the sampling of the electrochemical signals is decoupled from the scan generation.

> **NOTE**
>
> The SCANGEN module is no longer available and it is now replaced by its successor module, the SCAN250.

Using the SCANGEN module it is possible to measure both the faradic and the capacitive currents during a cyclic voltammetry. Therefore, this particular form of cyclic voltammetry is well suited for monitoring changes in the double layer structure or electrochemical reactions characterized by fast electron transfer kinetics.

> **NOTE**
>
> The SCANGEN can generate a potential scan between 10 mV/s and 10 kV/s.

Depending on the hardware configuration, the SCANGEN can be used in two different scan rate ranges:

- **Without the ADC10M or the ADC750 module:** if the instrument is **not** fitted with the ADC10M and ADC750 module it is possible to use the SCANGEN module up to 250 V/s.
- **With the ADC10M or ADC750 module:** if the instrument is fitted with the ADC10M *(see Chapter 16.3.2.1, page 977)* or the ADC750 *(see Chapter 16.3.2.2, page 983)* module it is possible to use the SCANGEN module up to 10 kV/s.

> **NOTE**
>
> The SCANGEN module only works in potentiostatic mode.

16.3.2.20.1 SCANGEN module compatibility

The SCANGEN module is available for the following instruments:

- PGSTAT302N, PGSTAT302 and PGSTAT30
- PGSTAT128N and PGSTAT12
- PGSTAT100N and PGSTAT100
- PGSTAT20
- PGSTAT10

NOTE

The SCANGEN module is **not** compatible with the Autolab instruments not listed above.

16.3.2.20.2 SCANGEN scope of delivery

The SCANGEN module is supplied with the following items:

- SCANGEN module
- SCANGEN module label

16.3.2.20.3 SCANGEN hardware setup

To use the **SCANGEN** module, the hardware setup needs to be adjusted. The checkbox for the module needs to be ticked *(see Figure 1251, page 1155)*.

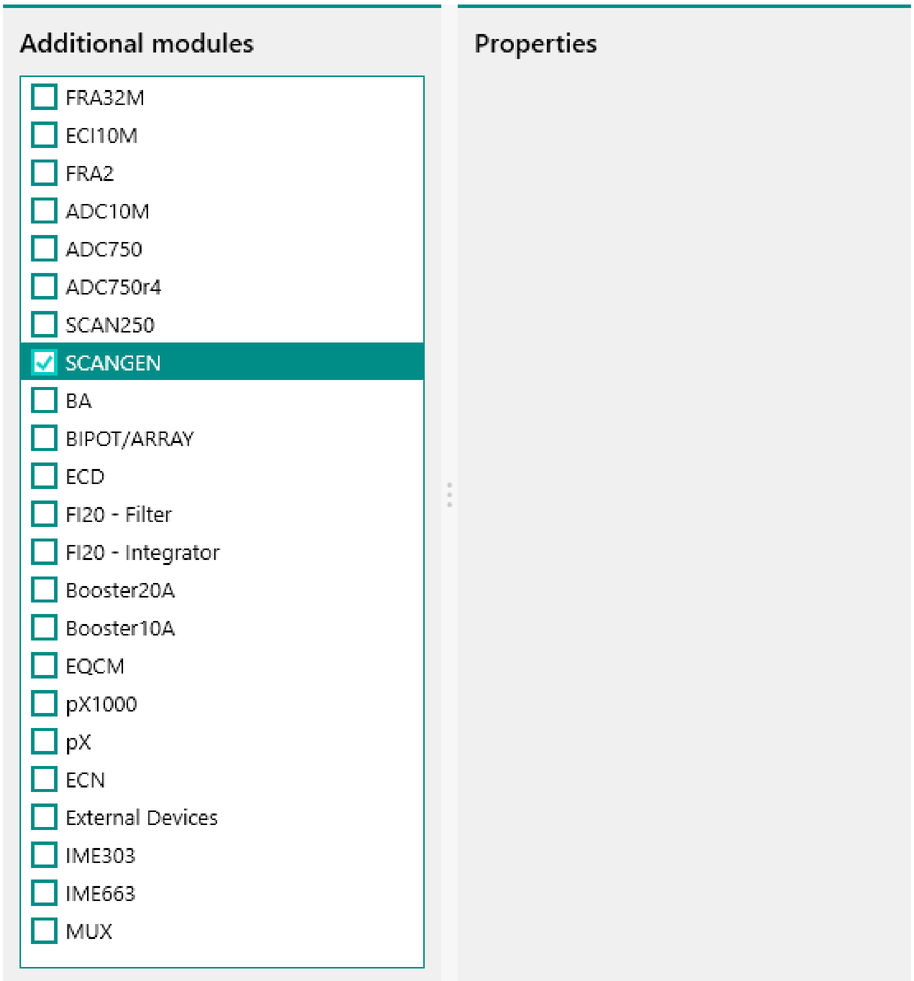


Figure 1251 The SCANGEN module is selected in the hardware setup

16.3.2.20.4 SCANGEN module settings

The SCANGEN module does not have any user-definable settings. All the module settings are directly controlled through the **CV linear scan** command.

16.3.2.20.5 SCANGEN module restrictions

Restrictions apply when using the SCANGEN module:

- **Potentiostatic control only:** the SCANGEN module can only be used in Potentiostatic mode.
- **Vertex overshoot or undershoot:** The SCANGEN module is fitted with a vertex detection circuit which is used to synchronize the reversal of the scan direction when the vertex is reached. At very high scan rates, this circuit can cause the linear scan generator module to stop the scan at a potential value that no longer matches the experimental parameters. This potential difference depends on the scan rate.

16.3.2.20.6 SCANGEN module front panel connections

The SCANGEN module is fitted with a single female BNC connector, labeled ←V *(see Figure 1252, page 1156)*.

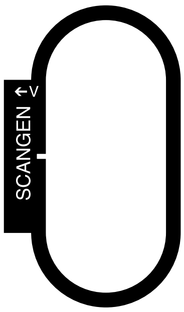


Figure 1252 The front panel label of the SCANGEN module

The signal provided through the ←V connector on the front panel corresponds to the output of scan generator signal of the SCANGEN module. The output signal is a voltage, referred to the instrument ground, corresponding to the specified potential scan.

NOTE

The output of the SCANGEN module does not include the offset potential which is created by the DAC164 module of the Autolab potentiostat/galvanostat.

NOTE

The front panel ←V BNC output is provided for information purposes only.

16.3.2.20.7 SCANGEN module test

NOVA is shipped with a procedure which can be used to verify that the integrator circuit of the **SCANGEN** module is working as expected.

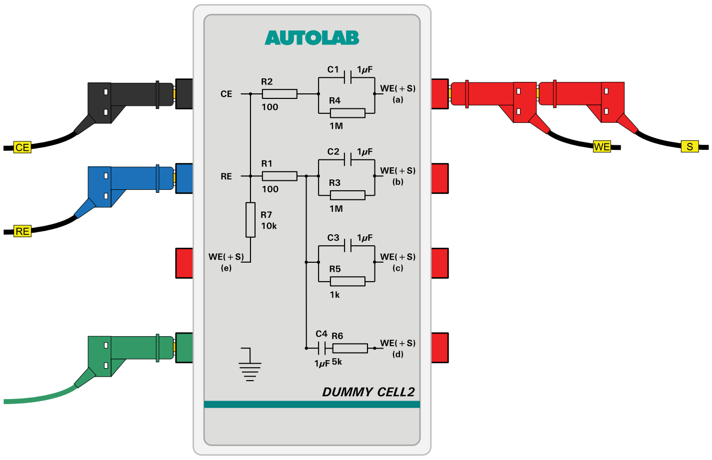
Follow the steps described below to run the test procedure.

1 Load the procedure

Load the **TestSCAN** procedure, provided in the NOVA 2.X installation folder (\Metrohm Autolab\NOVA 2.X\SharedDatabases\Module test\TestSCAN.nox)

2 Connect the Autolab dummy cell

Connect the PGSTAT to the Autolab dummy cell circuit (a).



3 Start the procedure

Start the procedure and follow the instructions on-screen. The test carries out a cyclic voltammetry measurement. At the end of the measurement, the measured data will be processed and a message will be shown. The measured data should look as shown in *Figure 1253*.

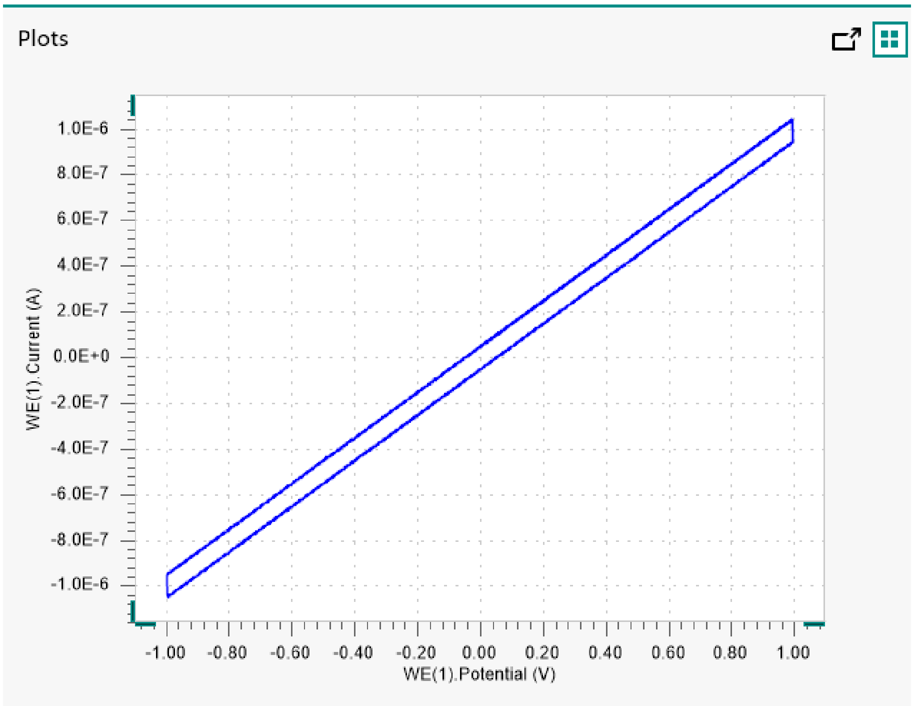
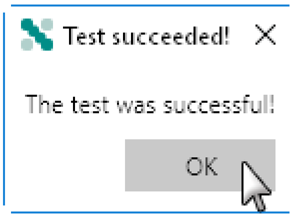


Figure 1253 The results of the TestSCAN procedure

4 Test evaluation

If the test is successful, a message will be shown at the end of the measurement.



The TestSCAN automatic evaluation of the data requires the following tests to succeed:

1. The inverted slope of the measured current versus the applied potential must be equal to 1000100 Ω± 5 %.
2. The calculated capacitance, determined from the integrated current signal, must be equal to 1 µF ± 10 %.

Both conditions must be valid for the test to succeed.

16.3.2.20.8 SCANGEN module specifications

The specifications of the SCANGEN module are provided in *Table 57*.

Table 57 Specifications of the SCANGEN module

Specification	Value
Scan range	± 5 V
Vertex resolution	2.5 mV
Vertex accuracy	± 5 mV
Output offset	± 1 mV
Minimum scan rate	10 mV/s
Maximum scan rate	10 kV/s

17 Diagnostics

NOVA is supplied with a **diagnostics** tool that can be used to test the Autolab instrument. This tool is provided as a standalone application and can be accessed from the start menu, in the Autolab group (Start menu – All programs – Autolab – Tools).

The diagnostics tool can be used to troubleshoot an instrument or perform individual tests to verify the correct operation of the instrument. Depending on the instrument type, the following items are required:

- **µAutolab type II, µAutolab type III and µAutolab type III/FRA2:** the standard Autolab dummy cell. For the diagnostics test, the circuit (a) is used.
- **PGSTAT101 and M101 module:** the internal dummy cell is used during the test, no additional items are required.
- **PGSTAT204 and M204 module:** the standard Autolab dummy cell. For the diagnostics test, the circuit (a) is used.
- **Other Autolab PGSTAT instruments:** the standard Autolab dummy cell and a 50 cm BNC cable. For the diagnostics test, the circuit (a) is used. The BNC cable must be connected between the ADC164 channel 2 and the DAC164 channel 2 on the front panel of the instrument .

NOTE

The **PGSTAT302F** must be tested in normal (grounded) mode.

17.1 Connecting the instrument

The Diagnostics application supports multiple Autolab instruments. When the application starts it detects all available instruments connected to the computer.

NOTE

If only one instrument is detected, the Diagnostics application will start immediately.

If more than one instrument is detected, a selection menu is displayed *(see Figure 1254, page 1161)*.

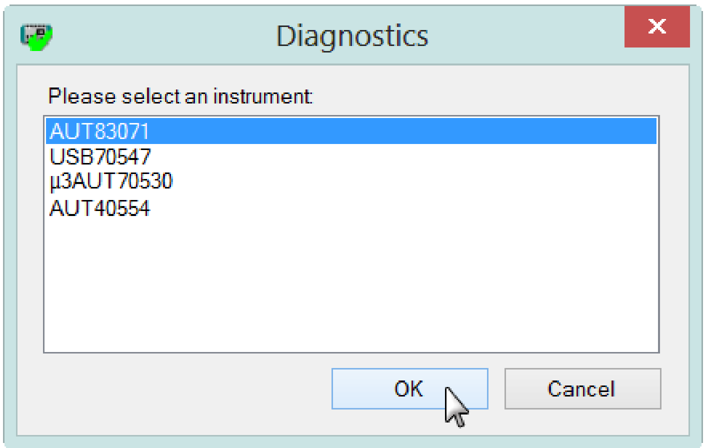


Figure 1254 A selection menu is displayed if more than one instrument is detected

When the diagnostics application is started with a Multi Autolab connected, the application will search for the available M101/M204 modules installed in the Multi Autolab and will list the available modules *(see Figure 1255, page 1161)*.

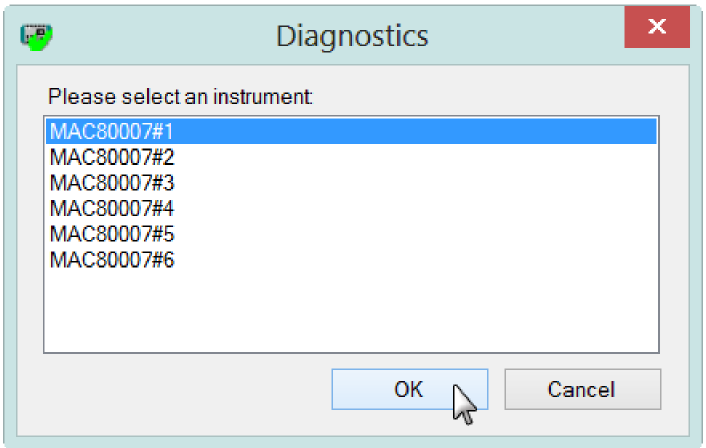


Figure 1255 A selection menu identifying the M101 or M204 modules by position is displayed when a Multi Autolab is detected by the diagnostics application

NOTE

The available instruments are identified by their serial number and position in the instrument, if applicable.

NOTE

Instruments with serial number beginning with AUT9 or with µ2AUT7, connected through an external USB interface, are identified by the serial number of the interface, USB7XXXX *(see Figure 1254, page 1161)*.

Select one of the available instruments and click the OK button to continue.

NOTE

The Diagnostics application can only test one instrument or one potentiostat/galvanostat module at a time.

17.2 Running the Diagnostics

When the application is ready, a series of tests can be performed on the connected instrument. Pressing the start button will initiate all the selected tests. A visual reminder will be displayed at the beginning of the test, illustrating the connections required for the test *(see Figure 1256, page 1163)*.

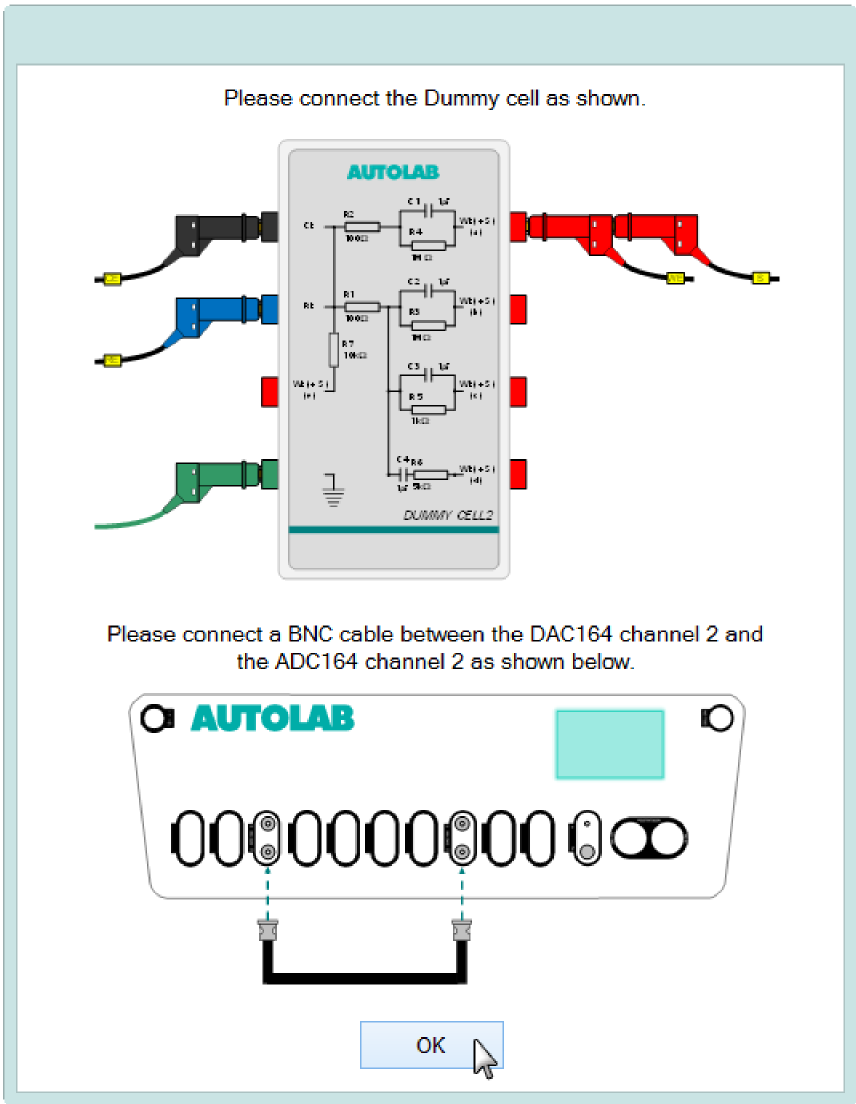


Figure 1256 A visual reminder is shown at the beginning of the Diagnostics test

During the test, the progress will be displayed and a successful test will be indicated by a green symbol and a progress bar *(see Figure 1257, page 1164)*.

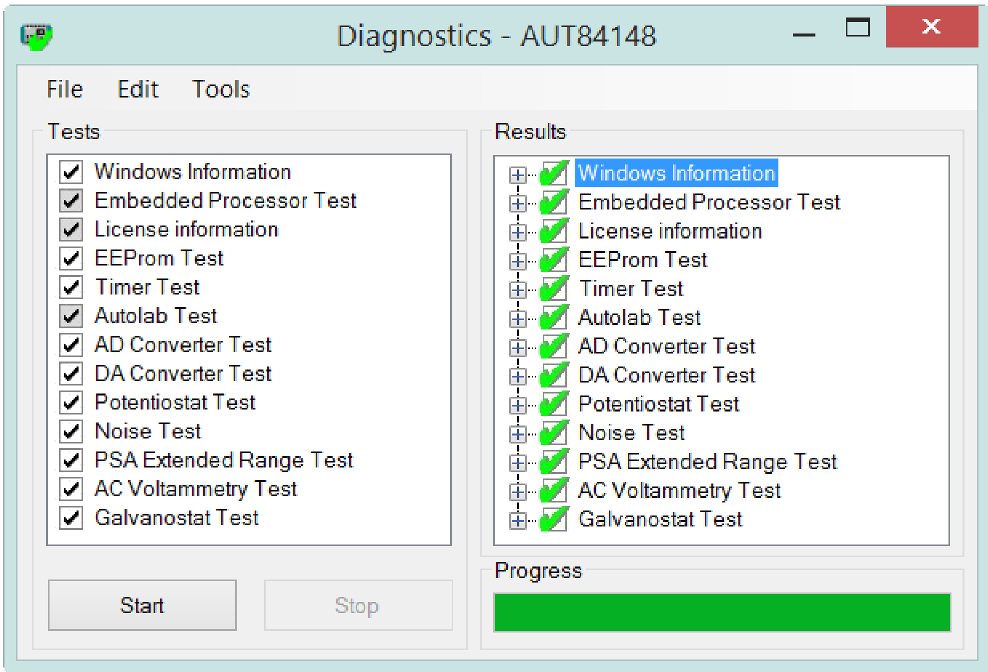


Figure 1257 The diagnostics report after all the tests have been performed successfully

If one or more of the tests fails, a red symbol will be used to indicate which test failed and what the problem is *(see Figure 1258, page 1164)*.

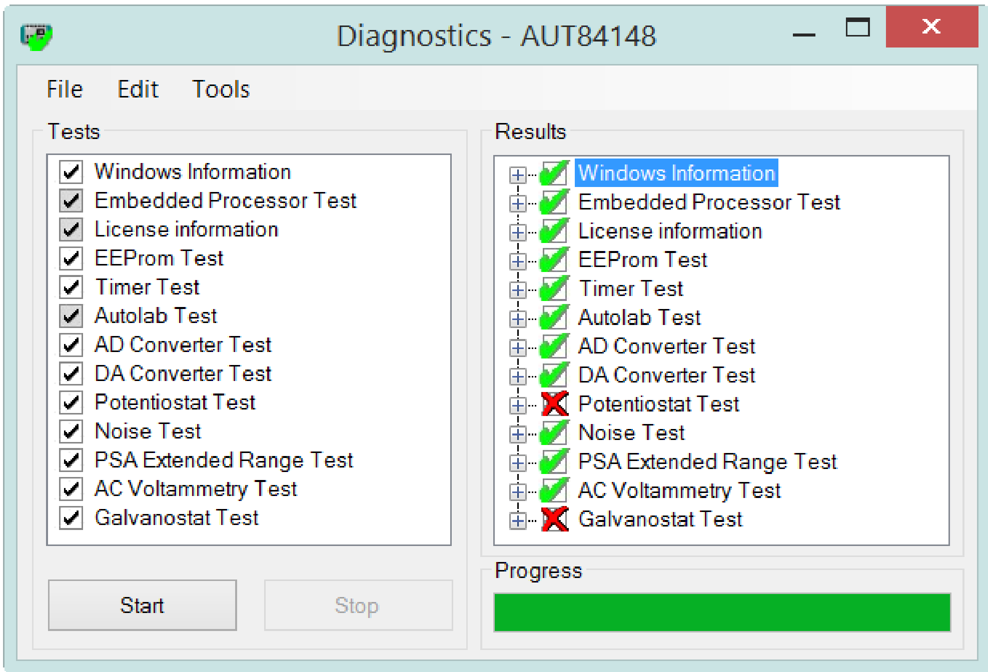


Figure 1258 A failed test will be indicated in the diagnostics tool

17.3 Integrator calibration

When a FI20 module or the on-board integrator is installed in the instrument, a message will be displayed at the end of the Integrator test *(see Figure 1259, page 1165)*.

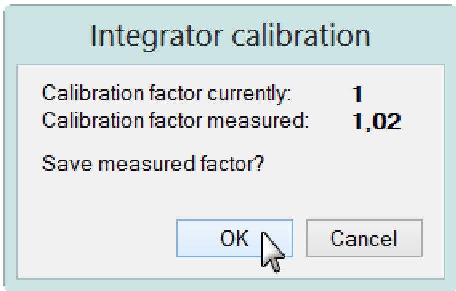


Figure 1259 The value of the measured Integrator calibration factor is displayed at the end of the integrator test

If the measured value differs from the stored value for the instrument, the new value can be stored in the hardware setup.

17.4 Diagnostics options

The Diagnostics application provides a number of options available through the provided menu .

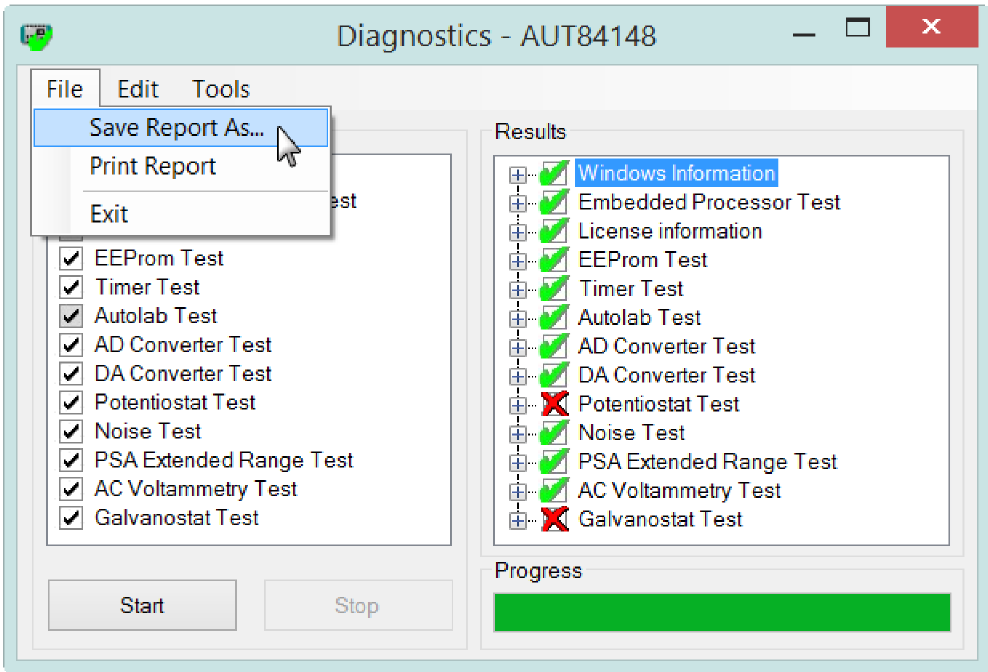


Figure 1260 The options of the Diagnostics tool can be accessed through the menu

The following options are available through the Menu in the Diagnostics application:

File	
Save Report As...	Used to save the Diagnostics report as a TXT file.
Print Report	Used to print the Diagnostics report.
Exit	Closes the Diagnostics application.

Edit	
Select All Tests	Activates all available tests provided by the Diagnostics application.
Deselect Optional Tests	Deactivates all optional tests provided by the Diagnostics application.
Hardware Setup	Opens the Hardware Setup window.
Select Instrument	Opens the instrument selection dialog window.

Tools	
Checksum error wizard	Opens the checksum error wizard. This option is only used for service purposes.

17.5 Firmware update

For some instruments, a firmware update may be required. If this is the case an update message will be displayed *(see Figure 1261, page 1166)*.

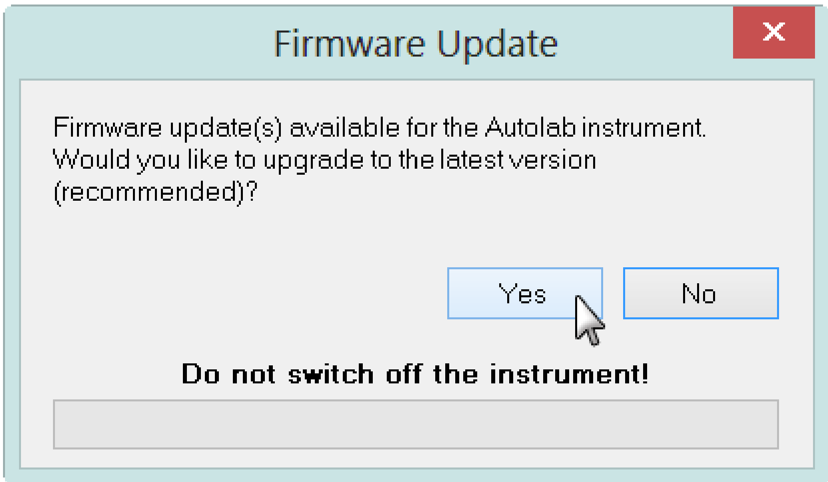


Figure 1261 An update message is displayed when the outdated firmware is detected

Clicking the Yes button when prompted will update the firmware. The firmware update is permanent. The Firmware Update window will close automatically at the end of the update and the diagnostics test will continue.

CAUTION

Do **not** switch off the instrument or disconnect the instrument during the firmware update since this will damage the instrument.

18 Warranty and conformity

This chapter provides information about warranty, safety and conformity the Autolab product range.

18.1 Warranty

The warranty on Autolab products is limited to defects or failures that are traceable to material, construction or manufacturing errors, which occur within 36 months from the day of delivery. In this case, the defects or failures will be rectified by Metrohm Autolab free of charge. Transport costs are to be paid by the customer, if applicable.

Glass breakage in the case of electrodes, cells or other parts is not covered by the warranty. Consumables (electrodes, QCM crystals, etc.) are not covered by the warranty.

If damage of the packaging is evident on receipt of the goods or if the goods show signs of transport damage after unpacking, the carrier must be informed immediately and a written damage report is demanded. Lack of an official damage report releases Metrohm Autolab from any liability to pay compensation.

If any instruments or parts have to be returned, the original packaging should be used. This applies to all instruments, electrodes, cells and other parts. If the original packaging is not available it can be ordered at Metrohm Autolab or at your local distributor. For damage that arises as a result of non-compliance with these instructions, no warranty responsibility whatsoever will be accepted by Metrohm Autolab.

Do not modify the cell cable or the differential amplifier cable connectors. These cables are designed for the best possible operation. Modifications of these connections, i.e. with other connectors, will lead to the loss of any warranty.

18.2 Spare part availability

All products designed, produced and tested by Metrohm Autolab are covered by 10 years spare part availability, from the day of delivery. Failures or defects experienced during this period will be rectified by Metrohm Autolab in such a way that the product will comply with all the original requirements and specifications. After the period of ten years, products supplied by Metrohm Autolab may no longer be serviceable. Metrohm Autolab will however attempt to repair any failure or defect beyond this time limit as long as spare parts remain available.

18.3 Declaration of conformity

This chapter provides the following certificates of conformity:

- For the PGSTAT128N, PGSTAT302N, PGSTAT100N and all derived instruments *(see Chapter 18.3.1, page 1169)*
- For the Autolab PGSTAT101, PGSTAT204, Multi Autolab M101 and Multi Autolab M204 *(see Chapter 18.3.2, page 1170)*

18.3.1 Declaration of Conformity

This is to certify the conformity to the standard specifications for electrical appliances and accessories, as well as to the standard specifications for security and to system validation issued by the manufacturing company.

Name of commodity

Autolab PGSTAT128N, PGSTAT302N, PGSTAT100N

Research potentiostat/galvanostat for electrochemical experimentation

This instrument has been built and has undergone final type testing according to the standards:

Electromagnetic compatibility

Emission:	EN61326-1 (1997) + A1 (1998) + A2 (2001) + A3 (2003)	
	EN61000-3-2 (2006)	
	EN61000-3-3 (1995) + A1 (2001) + A3 (2003)	

Immunity:	EN61326-1 (1997) + A1 (2001) + A3 (2003)

Safety specifications EN61010-1

This instrument is in conformity with EU directives 89/336/EEC and 73/23/EEC and fulfills the following specifications:

EN 61326	Electrical equipment for measurement, control and laboratory use – EMC requirements
EN 61010-1	Safety requirements for electrical equipment for measurement, control and laboratory use

Metrohm Autolab B.V. is holder of the TŰV-certificate of the quality system ISO 9001:2008 for quality assurance in development, production, sales and service of instruments and accessories for electrochemistry (registration number 7528/2.2).

Declaration of Conformity

Utrecht, October 1st, 2009

J. J. M. Coenen

Head of R&D

A. Idzerda

Head of Production

18.3.2 Declaration of Conformity

This is to certify the conformity to the standard specifications for electrical appliances and accessories, as well as to the standard specifications for security and to system validation issued by the manufacturing company.

Name of commodity

Autolab PGSTAT101, PGSTAT204, M101 and M204

Research potentiostat/galvanostat for electrochemical experimentation

This instrument has been built and has undergone final type testing according to the standards:

Electromagnetic compatibility

Emission:	EN61326-1 (2006)
	EN61000-3-2 (2006) + A2 (2009)
	EN61000-3-3 (2008)
Immunity:	EN61326-1 (2006)

Safety specifications EN61010-1 (2010)

This instrument is in conformity with EU directives 89/336/EEC and 73/23/EEC and fulfills the following specifications:

EN 61326	Electrical equipment for measurement, control and laboratory use – EMC requirements
EN 61010-1	Safety requirements for electrical equipment for measurement, control and laboratory use

Metrohm Autolab B.V. is holder of the TŰV-certificate of the quality system ISO 9001:2008 for quality assurance in development, production, sales and service of instruments and accessories for electrochemistry (registration number 7528/2.2).

Declaration of Conformity

Utrecht, October 6th, 2014

J. J. M. Coenen

Head of R&D

A. Idzerda

Head of Production

18.4 Environmental protection

The pictogram shown in *Figure 1262* located on the product(s) and / or accompanying documents means that used electrical and electronic equipment (WEEE) should not be mixed with general household waste. For proper treatment, recovery and recycling, please take this product(s) to designated collection points where it will be accepted free of charge.

Figure 1262 The WEEE logo shown on Autolab products

Alternatively, in some countries, you may be able to return your products to your local retailer upon purchase of an equivalent new product. Disposing of this product correctly will help save valuable resources and prevent any potential negative effects on human health and the environment, which could otherwise arise from inappropriate waste handling.

Please contact your local authority for further details of your nearest designated collection point. Penalties may be applicable for incorrect disposal of this waste, in accordance with you national legislation.

Index

D

L

M

N

O

P

Q

R

S

T

U

V

W