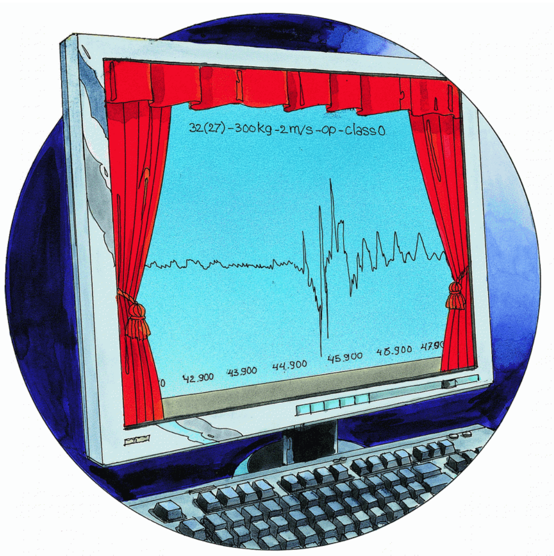
Autosoft 3000 Manual



Version number	1.5
Release date	10-02-2017
Author	A. Cosman

Table of Contents

Introduction

Autosoft 3000 is a software package designed to configure and control Autolog and Unilog data acquisition systems produced by Peekel Instruments B.V. It can control multiple Autolog 3000, Autolog 2005, Autolog 2100 and/or Unilog 2500 devices with BASE controller connected to the same PC.

To run Autosoft 3000, you need at least Windows XP (Windows 7 or higher recommended), a processor running at 1 GHz or higher and 512 Mb of memory. Depending on the measurement speed and amount of online visualization a faster processor and/or more memory may be required.
Make sure you install a recent Service Pack. For use under Windows XP, Service Pack 3 is required. Windows Vista and Windows 7, 8.x and 10 (32- and 64-bit) are supported as well.

To measure data using Autosoft 3000, first configure one or more measurement devices. After that, create channels to measure. Depending on the hardware, Autosoft 3000 can measure DC Voltage, Thermocouples, Pt-100, Strain gauges, Transducers, LVDT's and Digital Inputs and Outputs.
In addition, you can create Rosette channels for combinations of two or three strain gages. Virtual channels allow you to perform complex calculations on the measured data on-line.
After your devices and channels are configured, you can create measurement groups in which you can place any collection of channels to be measured. Numerical groups allow you to show measurement data on screen in numerical form, and also store measured data. Online graphics are supported using graphical groups, which can show up to 16 channels in a single graphical display.
In addition to all of this, Autosoft 3000 also supports autobalance measurements, alarms (which can not only be displayed, but can also trigger output relays and the start or stop of measurements) and the manual setting of output channels.

License

Autosoft 3000 is licensed for use on a single computer. The type and number of data acquisition devices that can be controlled by Autosoft 3000 depends on the license type.

The following license types are available, each of which requires a specific license code to use:

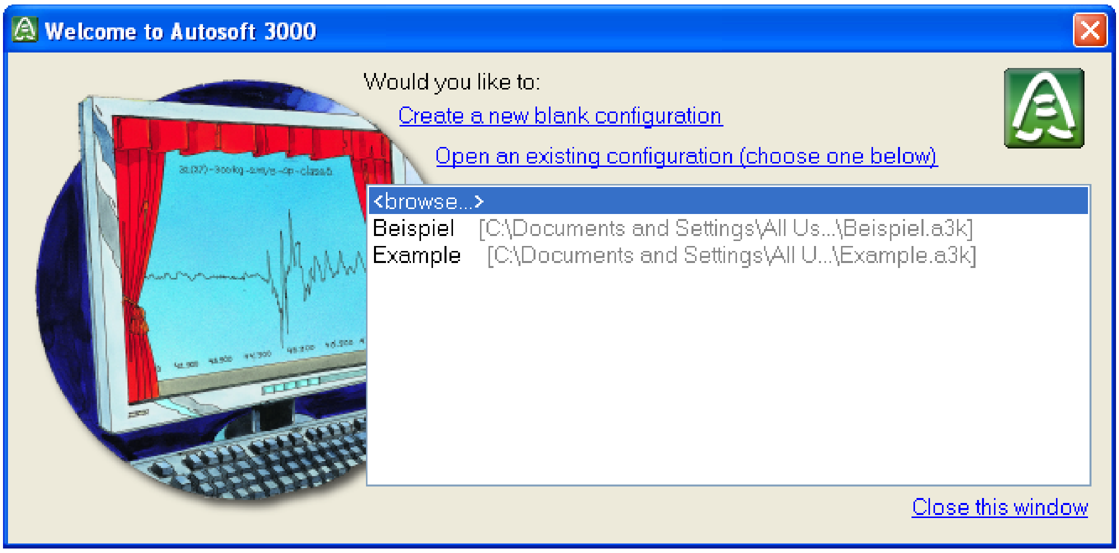
- Autosoft 3000-01: Allows the use of a single Autolog 3000 card in any type of housing.
- Autosoft 3000-03: Allows the use of a single Autolog 3003 device or any other type of Autolog 3000 housing containing no more than 3 data acquisition cards in total.
- Autosoft 3000-08: Allows the use of a single Autolog 3008 device or any other type of Autolog 3000 housing containing no more than 8 data acquisition cards in total.
- Autosoft 3000: Allows the use of any number of Autolog 3000 based data acquisition cards.
- Autosoft 3000-A2: Allows the use of any number of Autolog 3000 based data acquisition cards combined with any Autolog 2005, Autolog 2100 and/or Unilog 2500 devices with BASE controller.

Each license can be used to control a single measurement chain (PC + Measurement devices). If multiple measurement chains are used simultaneously, extra licenses of Autosoft 3000 are required.

For the sole purpose of off-line analysis and export of measurement data, a license can be installed on an additional computer.

First Steps

When you start Autosoft 3000 you will be greeted with the following dialog:



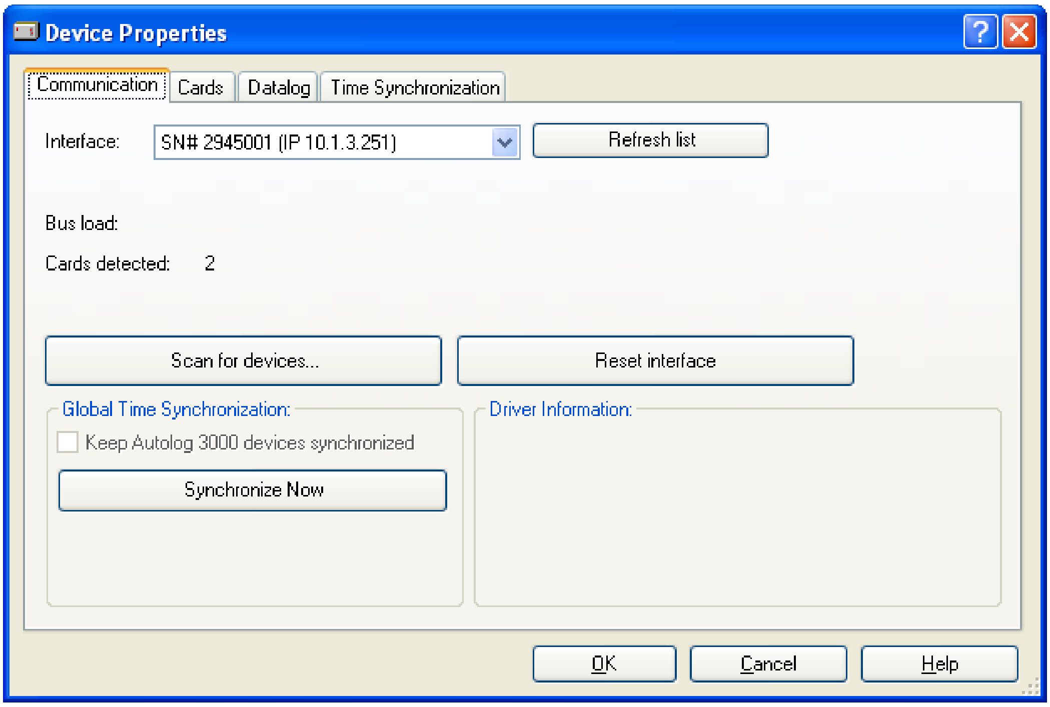
Click on the line "Create a new blank configuration" to start. Autosoft 3000 will ask you where to store the configuration, and create a configuration file with the extension ".a3k". It will also create a subdirectory for storing measurement data and other information. If, for example, you create a configuration named "Test", Autosoft 3000 will create a file "Test.a3k" and an accompanying directory "Test_Archive". If you need to move or rename the configuration, always make sure you move/rename both the configuration file and the archive directory.

Users of Autosoft NT can import Autosoft NT settings files (extension ".set"). To do this, open an existing configuration and use the dropdown list at the bottom of the open file dialog to select "Autosoft NT Configuration Files". Note that Autosoft 3000 can not import all aspects of the Autosoft NT settings file; you should always check if the import matches your expectations.

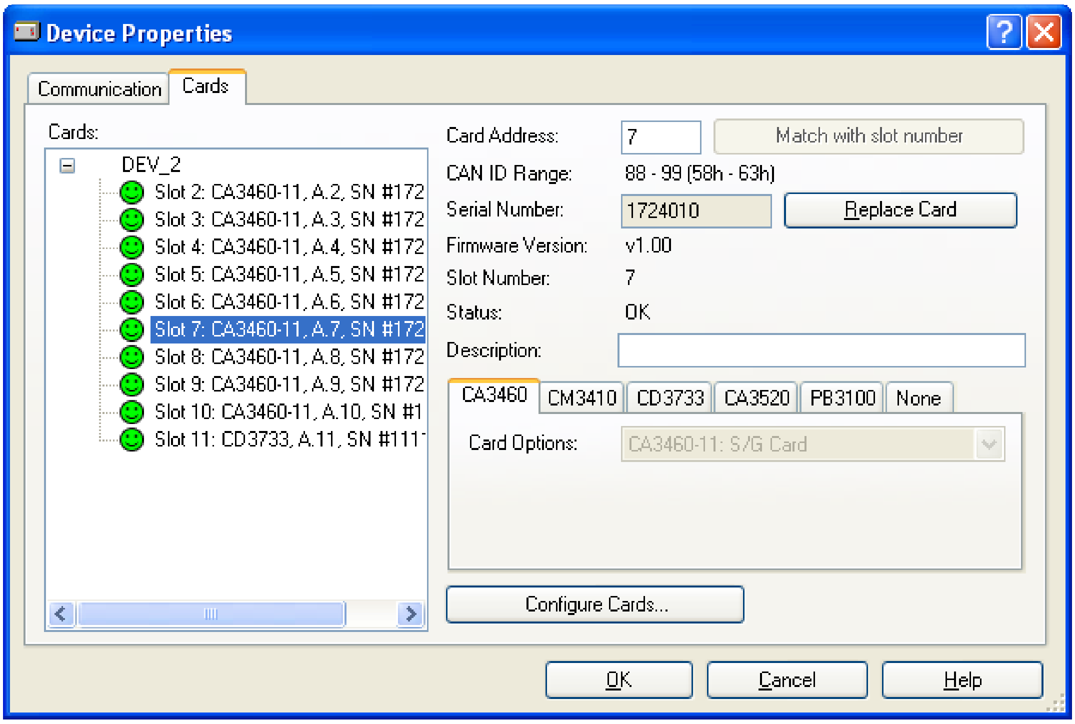
Adding an Autolog 3000

To add a new Autolog 3000 to your configuration, choose "New Autolog 3000" from the "Insert" menu.

In the dialog that appears, select the connection from the "Interface" dropdown list, then click "Scan for devices" to detect a connected Autolog 3000 and its cards. If all goes well the number of detected cards is shown and the "Driver Information" window shows the text "Connected."



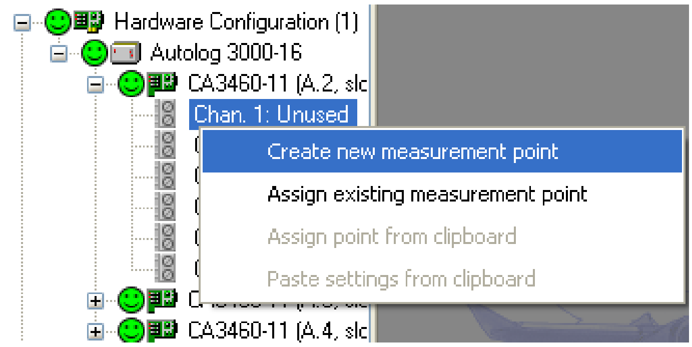
Click on the tab "Cards" to see and configure the individual data acquisition cards.



This part of the dialog allows you to review each card. For the CM3410 multiplexer card, this is the place where you need to consider which sensor types you wish to measure. Depending on the sensor type, you can configure the CM3410 to measure 2-wire, 4-wire, 6-wire or 8-wire sensors, or a combination of them.

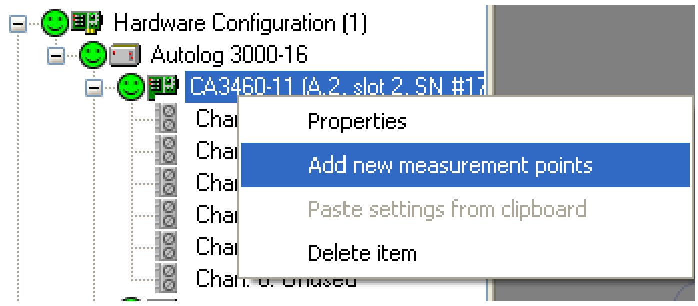
Configuring Sensors

To configure a new single measurement point, select an unused connection in the tree, right-click and select "Create new measurement point".

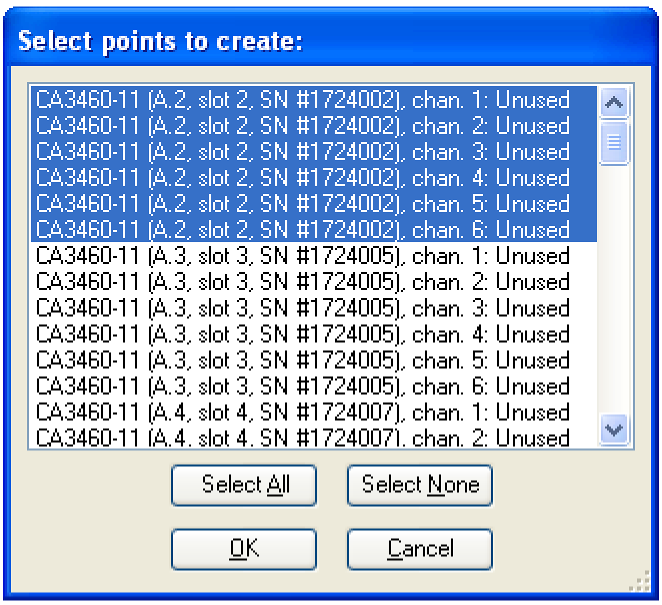


In the dialog that appears, click the "OK" button to create a new measurement point to attach to the chosen input channel.

Alternatively, to create a larger number of inputs with the same sensor settings, right-click a card or device and select "Add new measurement points".

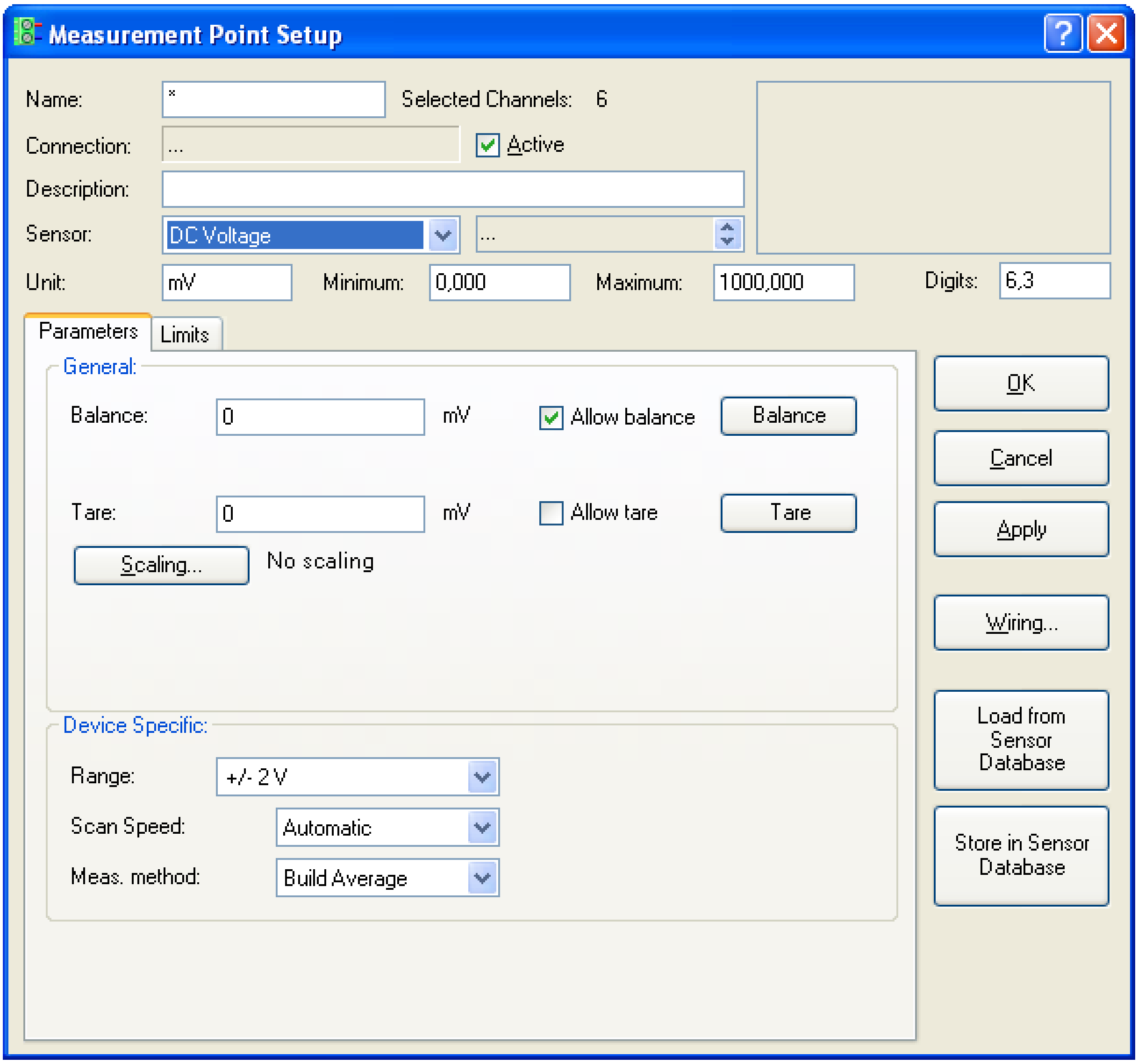


A dialog appears that allows you to select the inputs you want to configure.



Use shift-click and ctrl-click to select multiple input channels, marked in blue.

After selecting the input(s), the "Measurement Point Setup" dialog allows you to configure the inputs.



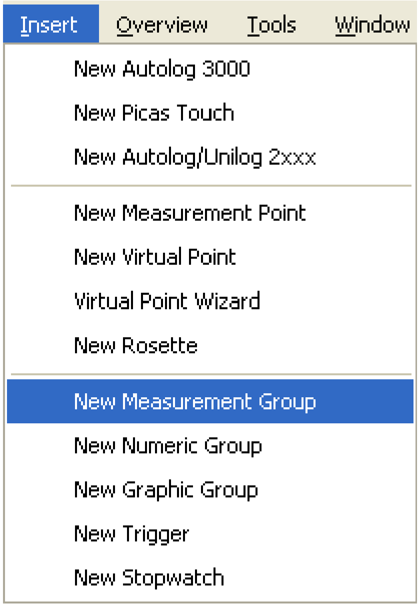
The upper part of the window allows you to configure basic (sensor-independent) settings like a name and description. After selecting the type of sensor you use, you should specify the range of values you expect to measure (minimum, maximum). This range is used to select appropriate default settings for the device and default scale settings for graphic displays. The digits field determines the default numerical presentation: '6.3' indicates max. 6 digits before, and 3 digits after the decimal point.

The lower parts shows the sensor-dependent settings, and is split into two parts: the general settings for a sensor apply to all device types, the device specific settings depend on the type of device used to measure the sensor.

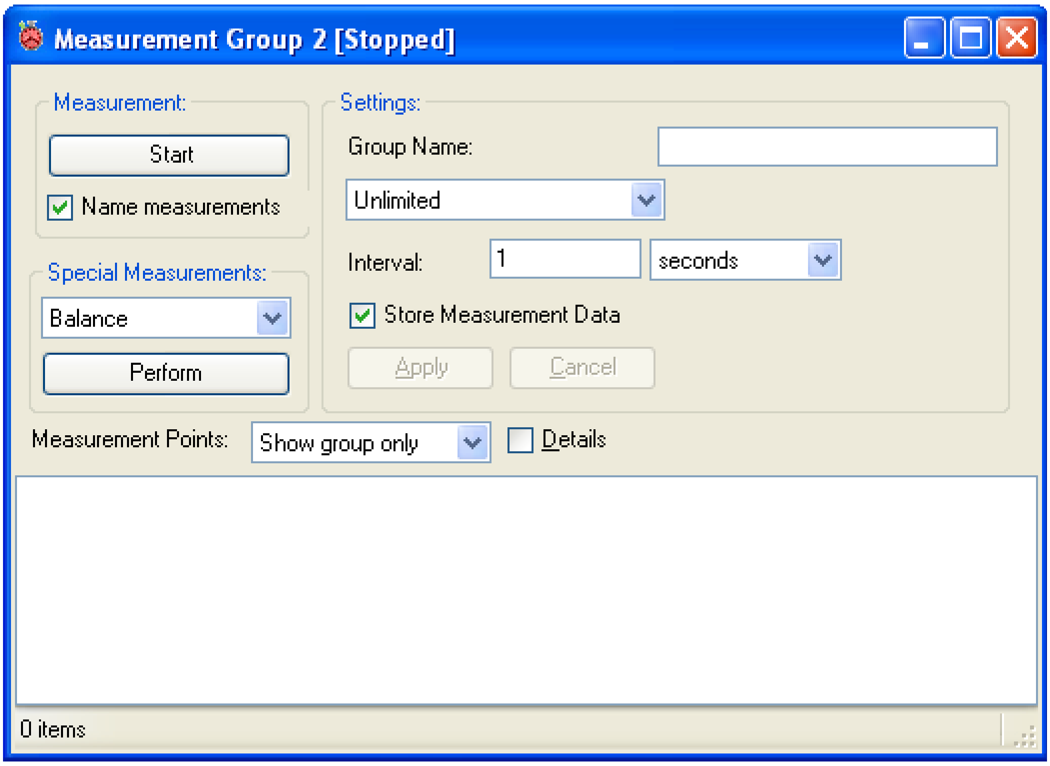
For more details information see the section on the Measurement Point Setup dialog.

Creating a Measurement Group

To start a measurement, you need to create a measurement group. Do this by selecting "New Measurement Group" from the "Insert" menu.



A new window will appear which contains the settings for the measurement group.

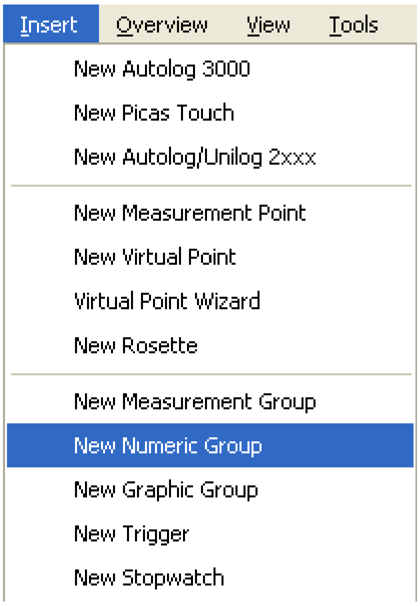


To add channels to the measurement group, you can drag and drop them from the tree on the left to the blank area at the bottom of this window. You can also drag and drop entire cards or devices into the group to quickly add all channels contained in it.

Shortcut to put all channels in a new measurement group: in the tree, drag the Autolog device line to the "Measurement Groups" line. Autosoft 3000 will ask for confirmation before creating a new measurement group containing all channels in the device.
Alternatively, click on the "Show group only" box in the measurement group window and change the selection to "Show all points". Now you can place check marks for all points to include in the measurement group.

Creating a Numeric or Graphic Group

To show the results of a running measurement, you need to create either a numeric or graphic group. To add a numeric group, select "New Numeric Group" from the "Insert" menu.



A blank window will appear, into which you can drag and drop channels in the same way as for measurement groups. Be aware that when you drop channels into a numeric group that already shows some other channels, were you drop the new channels will determine in which order they are inserted in the list. You can change the order in which the channels are shown by dragging and dropping inside the numeric window itself.

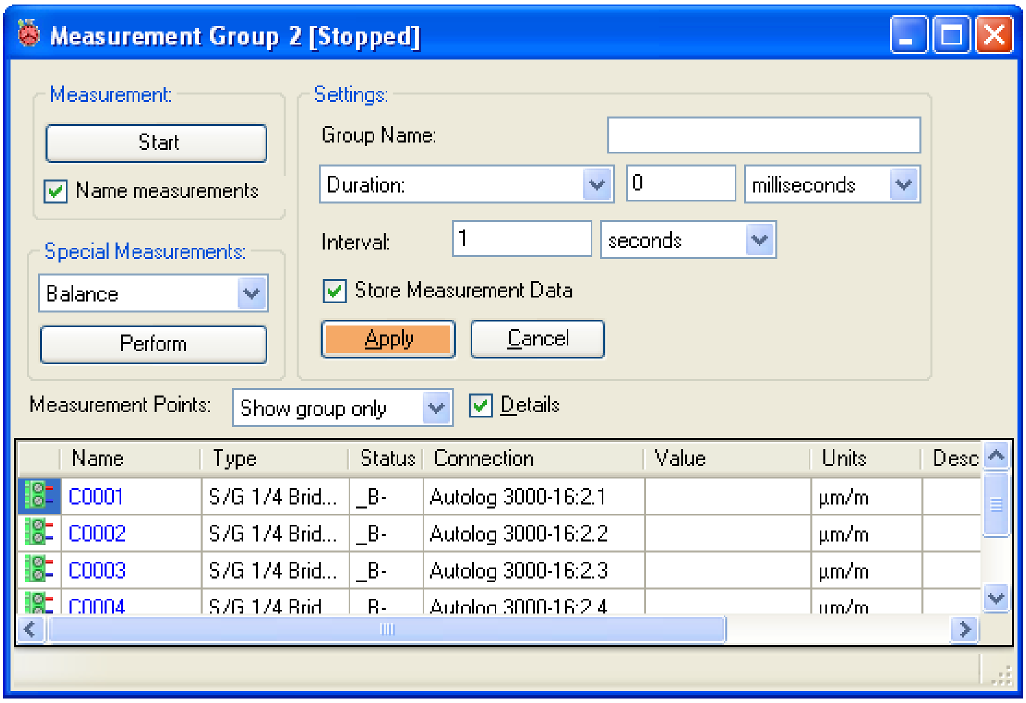
To change the settings for the numeric display, right-click inside the window and choose "Properties..." from the context menu.

To create a graphic group, the same general procedure can be followed. Note: the number of measurement points in a graphic group is limited to 16.

As with the measurement group, a quick way to create a numeric group containing all channels is to drag and drop the Autolog device in the tree view to the line "Numeric Groups", which will create a new numeric group containing all channels in the device.

Performing a measurement

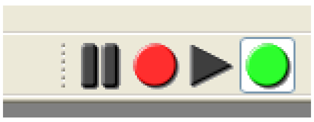
To perform a measurement, check the settings in the measurement group.



By default, the measurement group is configured to perform 1 measurement per second, after the start button is pressed, until the measurement is stopped manually. The check-box "Store Measurement Data" shows that the measurement data will be stored in the measurement archive (contained in the '_Archive' directory created when the configuration was saved for the first time).
If you uncheck the "Store Measurement Data" box, the measurement data from this group will not be saved!

Measurement data is stored in the archive in an optimized format only readable by Autosoft 3000 itself. Autosoft 3000 can retrieve data from it for graphic presentation (analyzing historic data in a graphic group) or for export to a format usable by other software (e.g. ASCII data).

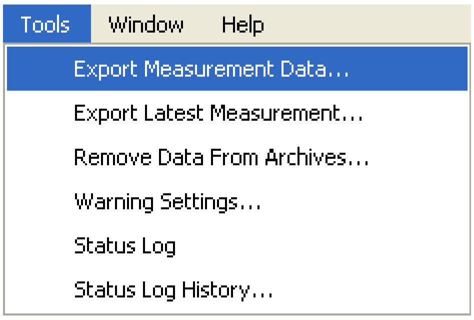
Use the Start button to manually start the measurement, and later to stop it. The current state of each measurement group is shown on the toolbar at the top right of the Autosoft 3000 main window.



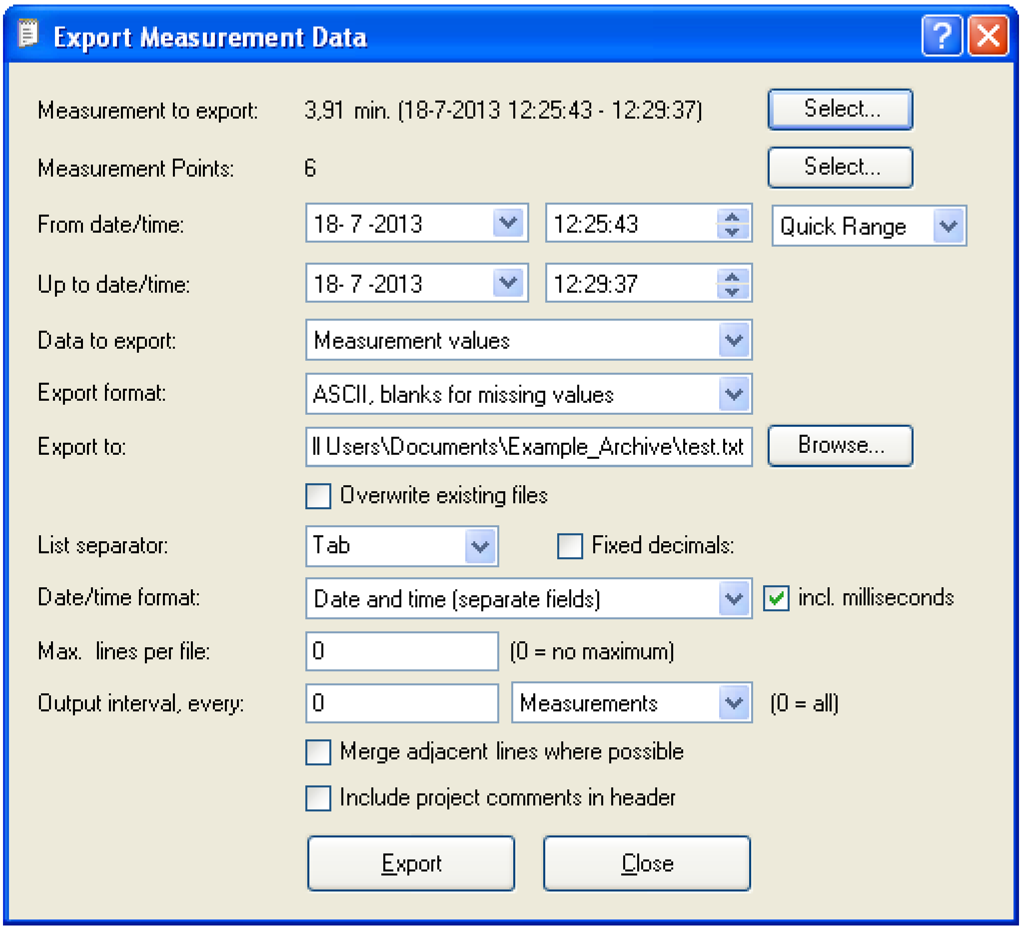
The example above shows the state of three measurement groups. The right-most is running and storing data (green circle). The black triangle represents a group with the "Store Measurement Data" box unchecked, which does not store any data. The red circle represents a group that does store data, but is not currently running.
When you hover the mouse of a button, it shows which measurement group it relates to and what the current status is.
The left-most button is the "Pause"-button, which allows you to temporarily disable all data storage, for example during set-up and testing of a configuration. To warn you that no data is stored during the time this button is pressed, it will continually blink red/black.

Exporting measurement data

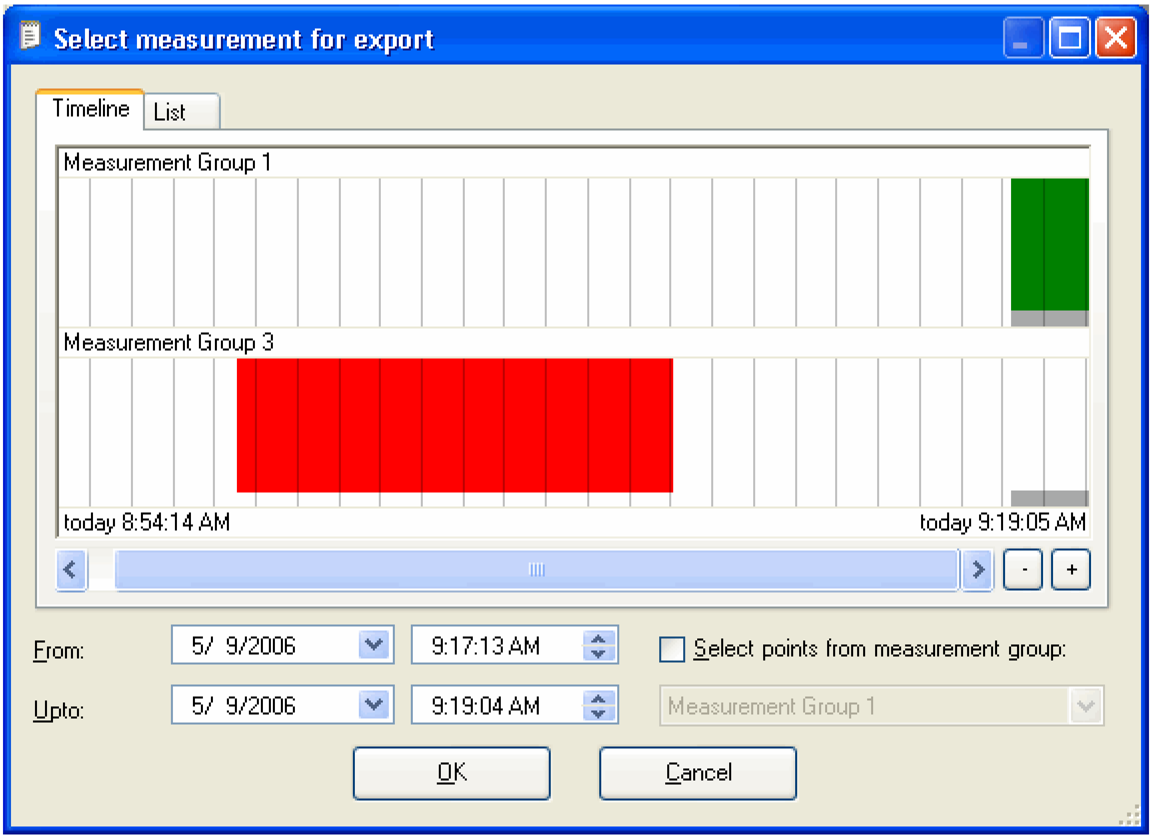
To retrieve measurement data from the historic archive, choose "Export Measurement Data" from the menu "Tools".



A dialog will show, which allows you to select which data you want to export and what the exported file should look like.



To make a selection based on measurements performed, click the top-right "Select..." button. A window will appear which shows a graphic time line of measurements performed. You can select a timeframe to export by clicking on a single measurement, clicking and dragging to select a specific time range, or shift-clicking to extend a time range to include multiple measurements. Use the scroll-wheel on your mouse to quickly zoom in and out (or use the +/- buttons in the dialog if your mouse does not have a scroll-wheel).



Alternatively, you can click on the "List" tab to choose measurements from a plain text list of measurements in reverse chronological order.

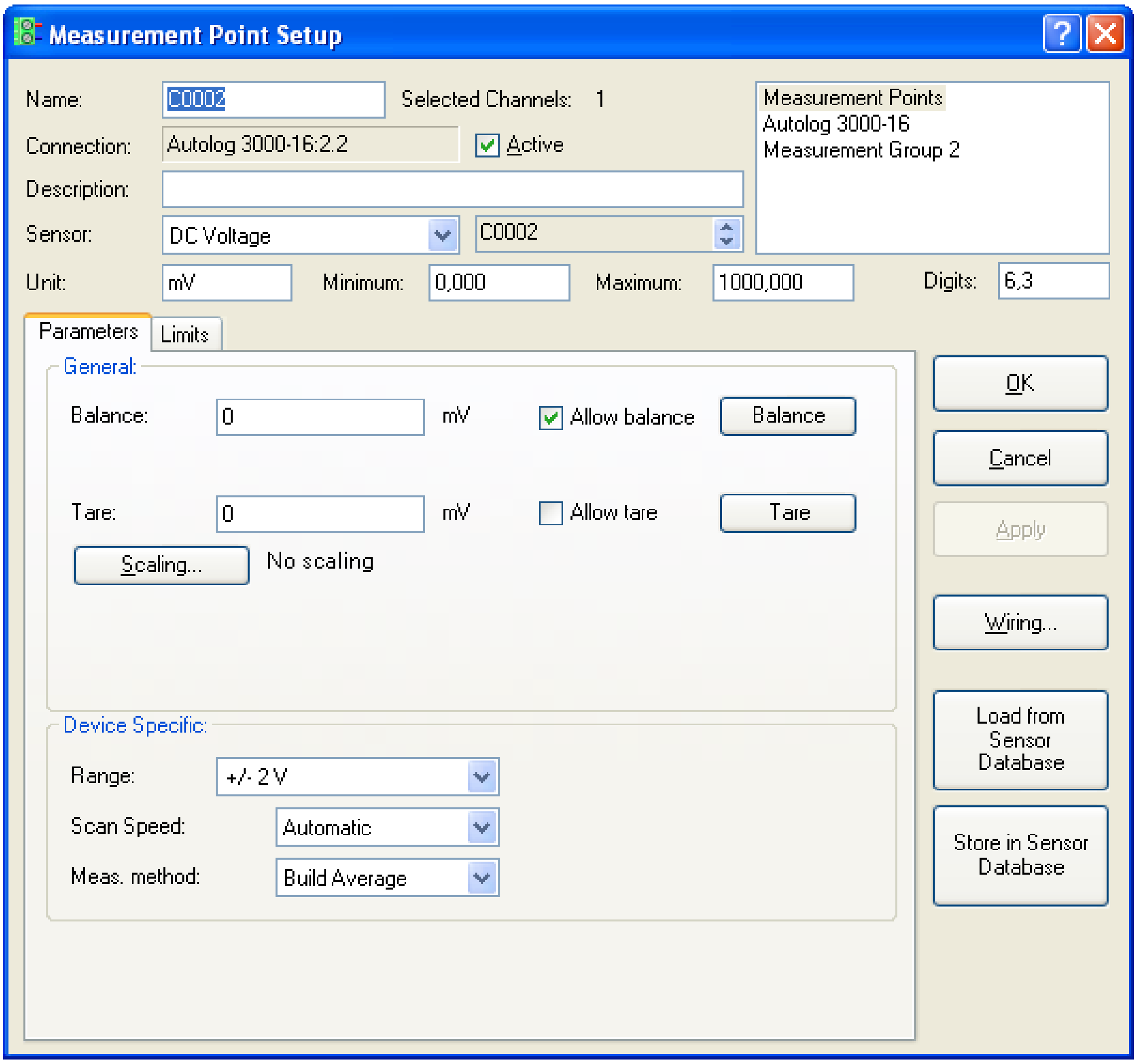
Measurement Points

Measurement points are used to acquire data and present measurement values. A measurement point can be used to measure an actual sensor. They can also be used as 'virtual points', were the values of other measurement points can be combined using formulas to create a derived measurement value. Rosette calculations form a special case, where the rosette can produce multiple output values. Each of these rosette outputs is a separate measurement point, whose value will be determined by the rosette configuration.

Measurement points can be defined independent of the actual hardware used to measure them, making it possible to attach the sensor to a different device or input by moving the measurement point to a different hardware connection. To facilitate this, settings for sensors are split into 2 sections: general and hardware-specific. Hardware-specific settings are only visible when the measurement point is assigned to a hardware connection. When this assignment changes, the hardware-specific settings will be chosen to match the sensor configuration.

Measurement Point Setup

In the measurement point setup dialog you can configure measurement points and their specific parameters.



The **name** of the point is set to '*' for new measurement points. You can change this name, but it should be no longer than 30 characters and may only contain letters, digits and underscores. The software will automatically enforce these rules. Channel names must be unique, if you choose an existing name it will automatically be changed upon storing the channel in the database. If you create more than one channel at the same time, sequential numbering will be appended to the name to ensure uniqueness.

On the top middle the dialog shows the number of points you are modifying or creating. **Be careful when modifying more than one points:** each setting you change will be applied to all points, but settings which show up blank are not the same for each point and will not be changed unless you choose a specific setting.

The **connection** shows to which physical channel the measurement point is connected positioned (device and channel number). You can change the connection by clicking on this field and making a new selection.

The **'active'** checkbox determines whether the measurement point is active or not. If you uncheck this box, the channel will be disabled and will no longer be measured.
The **description** of the point is a free-form text. You can use this text for any details you want to keep with the measurement point.

The **sensor field** shows which sensor type is to be measured. When configuring a measurement point, you can use the dropdown list to select the type of sensor. Only sensor types, which can actually be measured with the measurement point (and physical connection), will be shown in the list.

The measurement point name field next to the sensor field allows you to step through the consecutive points using the arrow buttons.
Beware: when you press the arrow buttons the settings for the current channel(s) will automatically be saved, as if you pressed the 'OK' button.

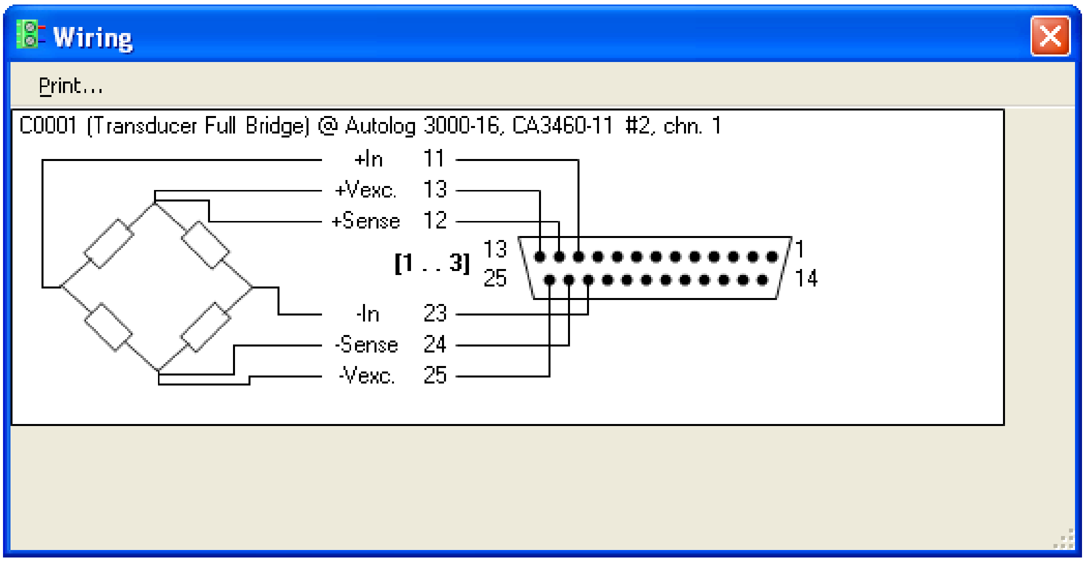
The **units** field allows you to set the units for the presentation of the measurement values. Depending on the type of sensor, either a specific selection of units is allowed, or the units can be chosen for the resulting value after scaling.

You can use the **minimum** and **maximum** fields to specify the range of values you want to measure. Setting this range correctly will ensure that Autosoft 3000 checks whether or not the chosen hardware settings allow you to measure the complete range of values. Also, these values are used to chose suitable default values for displaying in graphic and numeric groups.

Using the **digits** fields, you can specify the default presentation (in numeric and graphic groups) of numerical values for this measurement point. For example '6.3' sets a maximum of 6 pre-comma and exactly 3 post-comma digits.

The list on the top right of the dialog shows where the measurement point is located. In the example above, the channel is located in DEV_2. If the channel is placed in one or more groups, these will show up as well, allowing you to switch between the groups and configure the settings related to each group. **Beware:** when you select a new group the settings for the current group or device will automatically be saved, as if you pressed the 'OK' button.

The '**Wiring...**' button shows how a sensor should be connected.



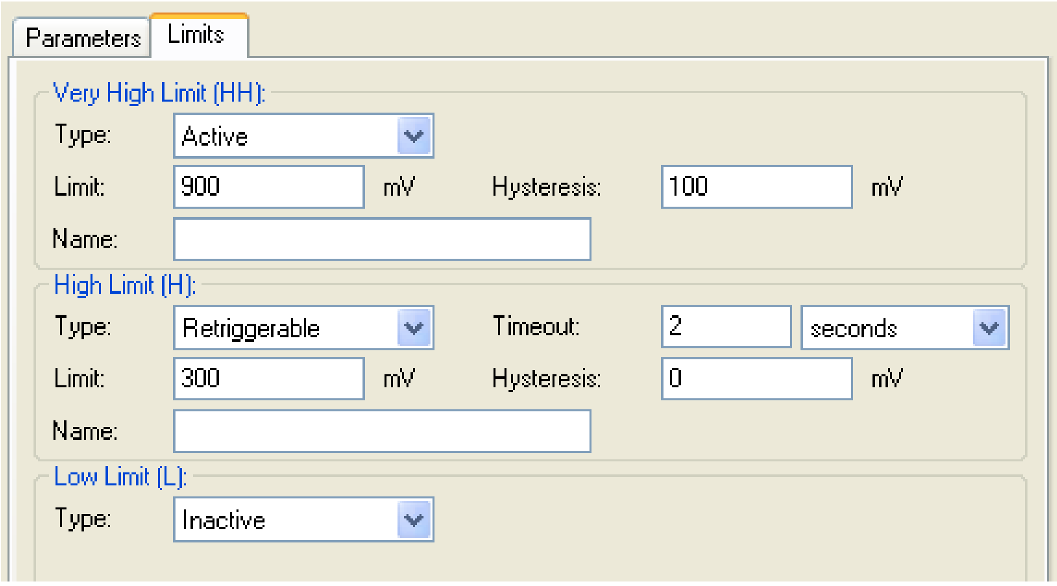
With the buttons '**Load from Sensor Database**' and '**Store in Sensor Database**' you can maintain a database of commonly used sensors in a separate file.

The lower part of the dialog shows settings that are dependent on the type of sensor and the hardware it is connected to. The following sensor types are available:

DC Voltage
Current
Pt-100
Thermocouple
Strain gauge
Transducer
LVDT
Resistance
Potentiometric
FBG Sensors
Digital Inputs and Outputs
Analog Outputs
Counters
Rosettes
Rosette Outputs
Virtual

The limits tab is available for almost all sensor types, to set limits for measurement values.

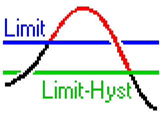
Measurement Point Limit Setup

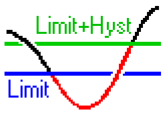


The limits configuration tab is part of the Measurement Point Setup dialog. Here, you can configure up to 4 different limits (High, Low, Very High and Very Low). When the measurement value surpasses the limit value, the event will be logged. Also, limits can be used to configure triggers, which allow you to perform specific actions based on limit exceedance.

In addition to the generic (Very) High/Low indication, each limit can be assigned its own **name**.

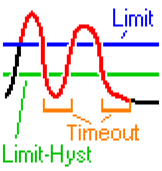
Use of limits with hysteresis:

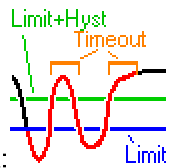
High limit: 

Low limit: 

The limit activates as soon as the measurement value exceeds the limit value, and stays activated until it falls below the **limit** - **hysteresis**. The **hysteresis** value is specified as an absolute, non-negative offset from the **limit** value.

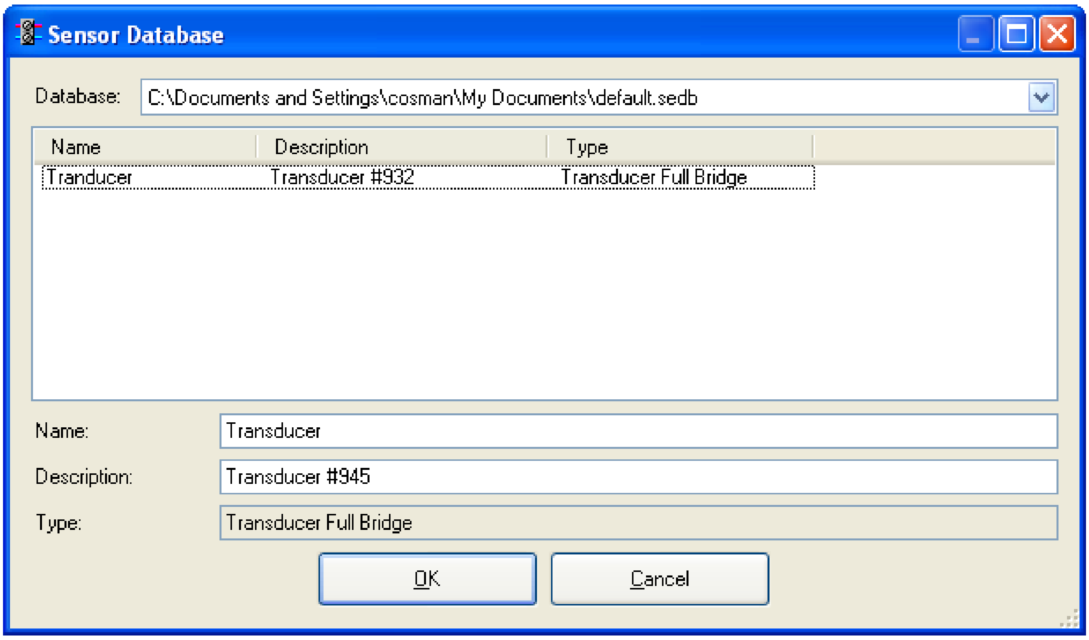
Use of retriggerable limits:

High limit: 

Low limit: 

The limit activates as soon as the measurement value exceeds the limit value, and stays activated until it falls and remains below the **limit** - **hysteresis** for at least the specified **timeout** time. The **hysteresis** value is specified as an absolute, non-negative offset from the **limit** value.

Sensor Database



The Sensor Database is a separate file (using the file extension .sedb) which can contain any number of predefined sensors. Every time you configure a sensor in Autosoft 3000 you can add it to a sensor database, so it can be used in another configuration at any time.

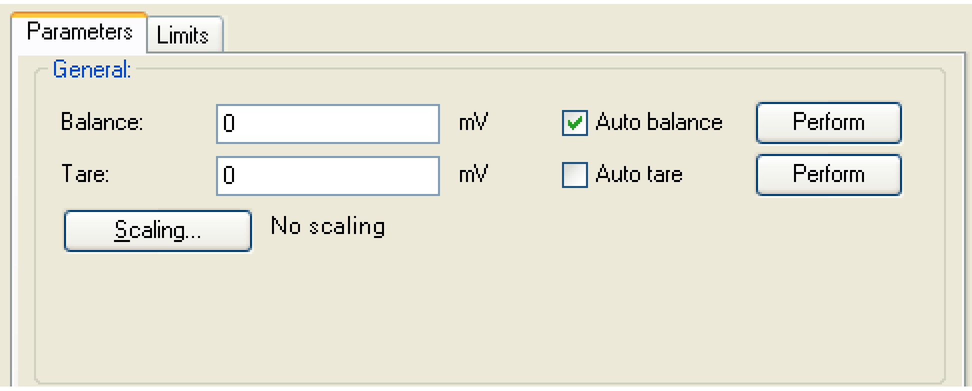
By default, a sensor database will be stored in the 'My Documents'-folder, and named 'default.sedb'. By clicking on the database file name you can see a list of recently used files or create / load a different database.

You can remove an existing sensor from the database by right-clicking on it and selecting '**Remove Sensor**'.

Sensor Database files are stored in XML format.

DC Voltage Settings

These settings are part of the measurement point setup dialog.

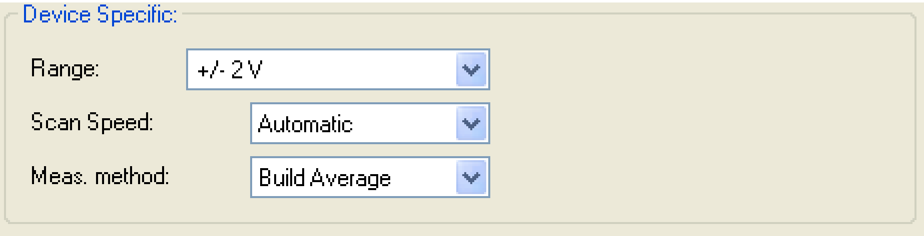


The **balance** value will only be used if the **auto balance** option is checked.
The **tare** value will only be used if the **auto tare** option is checked.

You can manually set a specific balance or tare value, which will then be subtracted from all measured values, thereby making the output 0 when the value you entered is measured.
Alternatively, you can assign the measurement point to a measurement group and perform either a balance or a tare measurement to determine the value.

You can scale in the input signal by clicking the scaling button. You can convert the mV signal to a more meaningful value by means of linear interpolation or a polynomial and setting appropriate **units** for the channel.

Settings specific to Autolog 3000

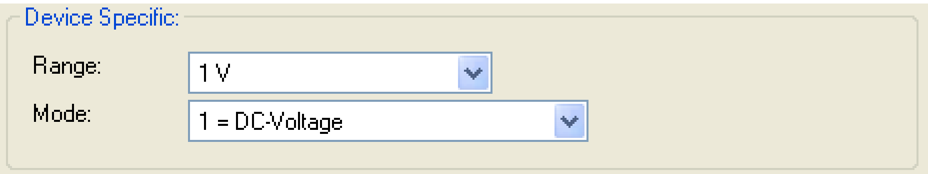


You can set the **range** for the measurement. If the selected range is not large enough to cover the range configured by the minimum and maximum value set for the channel, this field will be shown in red.

The **scan speed** determines how many measurements the Autolog 3000 card performs internally. For CA3460 cards, there are three possible scan speeds: 5 Hz, 100 Hz and 1 kHz. When set to **automatic** the speed is chosen to be the lowest available scan speed higher than or equal to the configured measurement speed.

The **meas.method** determines the way multiple measurement values are combined, in case the Autolog 3000 scan speed is higher than the configured measurement speed.

Settings specific to Autolog/Unilog 2xxx



The **range** of the DC measurement can have the following values:

Range	Actual measured range	Resolution
10 V	-13 V to 13 V	400 µV
1 V	-1300 mV to 1300 mV	40 µV
100 mV	-130 mV to 130 mV	4 µV
25 mV	-32.5 mV to 32.5 mV	1 µV

Note: The 25 mV range is only available when using the FDVM. If a channel is measured using the IDVM, the range will automatically fall back to 100 mV (with the corresponding resolution).

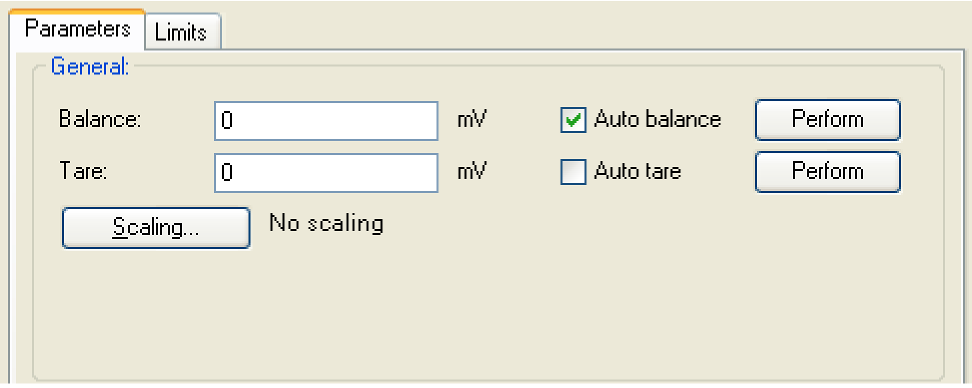
If the selected range is not large enough to cover the range configured by the minimum and maximum value set for the channel, this field will be shown in red.

You can manually set the **mode** for this channel when necessary. Use the default value '1 = DC-Voltage' to measure DC voltage.

For resistance measurements, use mode '5 = Resistance 4-wire' or '6 = Resistance 3-wire'. When measuring resistance you should be aware that 1 mV corresponds to 1 Ohm, since a constant current source of 1 mA is used to perform the measurements.
Note: Mode setting by software is only possible if the Autolog or Unilog has a CA-2568 mode selection card. If your device does not have this card, you must manually set the dip-switches on the cards to match the software mode setting. In this case, the software mode setting is only used as a reminder and for documentation purposes.

Current Settings

These settings are part of the measurement point setup dialog.

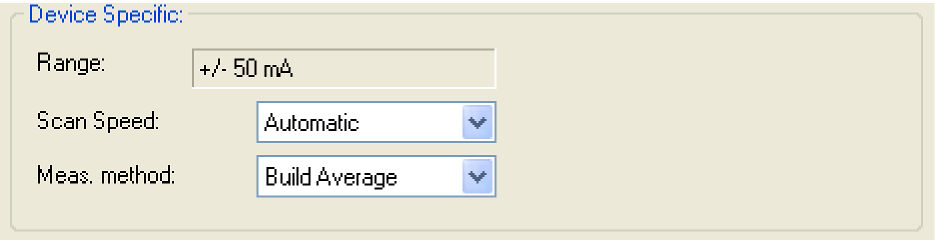


The **balance** value will only be used if the **auto balance** option is checked.
The **tare** value will only be used if the **auto tare** option is checked.

You can manually set a specific balance or tare value, which will then be subtracted from all measured values, thereby making the output 0 when the value you entered is measured.
Alternatively, you can assign the measurement point to a measurement group and perform either a balance or a tare measurement to determine the value.

You can scale in the input signal by clicking the scaling button. You can convert the current signal to a more meaningful value by means of linear interpolation or a polynomial and setting appropriate **units** for the channel.

Settings specific to Autolog 3000



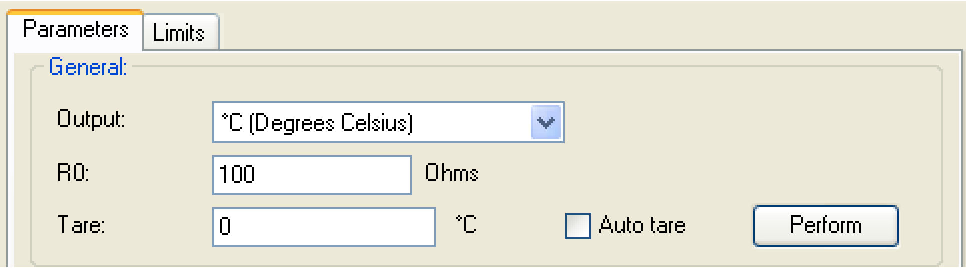
The **range** is fixed and only shown for documentation purposes.

The **scan speed** determines how many measurements the Autolog 3000 card performs internally. For CA3460 cards, there are three possible scan speeds: 5 Hz, 100 Hz and 1 kHz. When set to **automatic** the speed is chosen to be the lowest available scan speed higher than or equal to the configured measurement speed.

The **meas.method** determines the way multiple measurement values are combined, in case the Autolog 3000 scan speed is higher than the configured measurement speed.

Pt-100 Settings

These settings are part of the measurement point setup dialog.

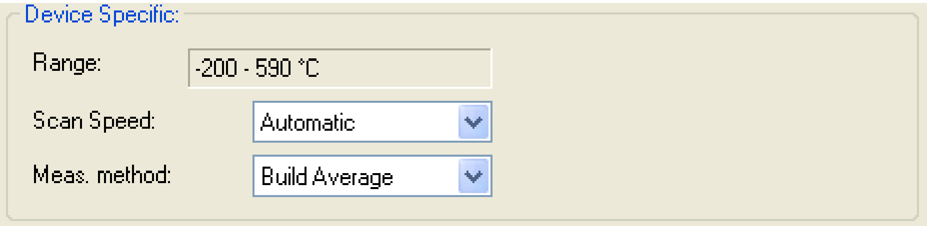


The **output** type determines the units used for presentation of this sensor.
The **R0** setting can be used to set the correct resistance of the Pt-sensor at 0 °C. Whether or not this value can be changed depends on the type of device used to measure this sensor. There are two possible reasons set a different R0 value. First, if you have a special sensor like a Pt-60, Pt-200 or Pt-1000, you can enter 60, 200 or 1000 for the R0 value. Second, if you know your Pt-100 has a slight imperfection and measures 100.5 Ohms instead of 100 Ohms at 0 °C, you can enter 100.5 for the R0 value.

The **tare** value will only be used if the **auto tare** option is checked.
You can manually set a specific tare value, if needed, or assign the measurement point to a measurement group and perform either a balance or a tare measurement to determine the value.

The formulas used to convert the signal in Ohms to degrees Celsius can be found in Appendix A.

Settings specific to Autolog 3000

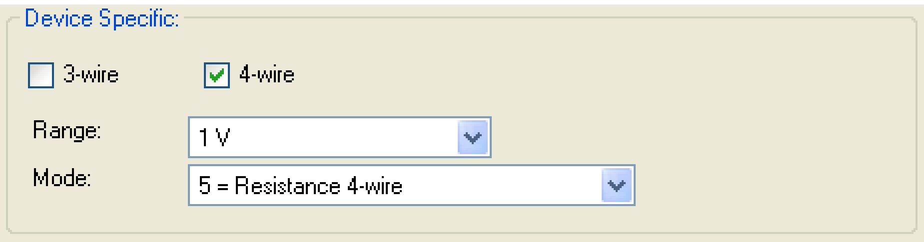


The **range** shows the available measurement range. It can not be changed and is only displayed for information.

The **scan speed** determines how many measurements the Autolog 3000 card performs internally. For CA3460 cards, there are three possible scan speeds: 5 Hz, 100 Hz and 1 kHz. When set to **automatic** the speed is chosen to be the lowest available scan speed higher than or equal to the configured measurement speed.

The **meas.method** determines the way multiple measurement values are combined, in case the Autolog 3000 scan speed is higher than the configured measurement speed.

Settings specific to Autolog/Unilog 2xxx

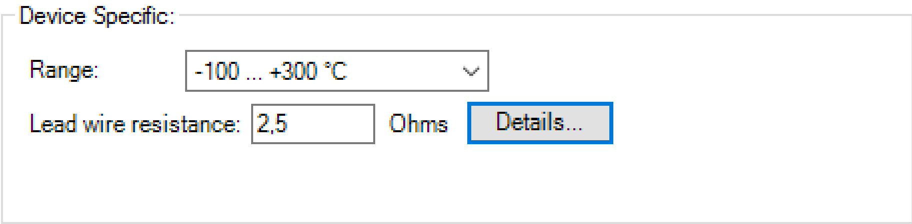


You can select between **3-wire** or **4-wire** measurement using the checkboxes. This will automatically update the mode for the channel. If the 4-wire checkbox is grayed, you are probably configuring this channel on a card, which cannot do 4-wire measurements.

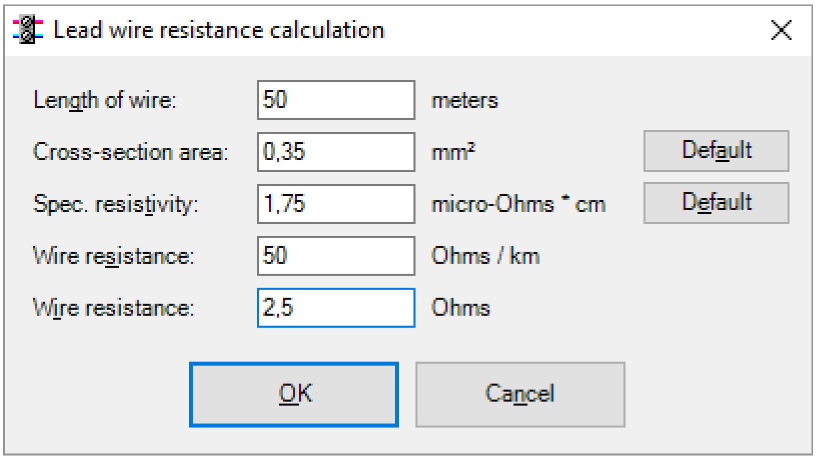
The DC **range** of the Pt-100 measurement should be **set to 1 V (default value) for best results**. You can, however, change the range for special purposes. A 100 mV range could be used to measure temperatures up to about 75 °C, for instance. If the selected range is not large enough to cover the range configured by the minimum and maximum value set for the channel, this field will be shown in red.

Note: Mode setting by software is only possible if the Autolog or Unilog has a CA-2568 mode selection card. If your device does not have this card, you must manually set the dip-switches on the cards to match the software mode setting. In this case, the software mode setting is only used as a reminder and for documentation purposes.

Settings specific to StrainBUSter



For StrainBUSter, the connection to a Pt-100 sensor should be compensated for the resistance of the lead wires to the sensor. This can be configured using the "**Details...**"-button, which opens the "Lead wire resistance calculation" dialog box:



Use this dialog to calculate the lead wire resistance. When measuring quarter bridge strain gauges or Pt-100 elements using 3 wires, the resistance of the lead wires can have a significant impact on the measured value. To reduce the error, the software can correct the measurement value for an estimated lead wire resistance.
BEWARE: If the lead wire resistance you enter is more than 2 times higher than the real lead wire resistance, you will increase instead of decrease the error! When in doubt, err on the side of caution: use a lower resistance value.

The individual items in this configuration page described:

Length of wire: The length of the lead wire in meters.

Cross-section area: The cross-section area of the conductive material of the wire in mm².

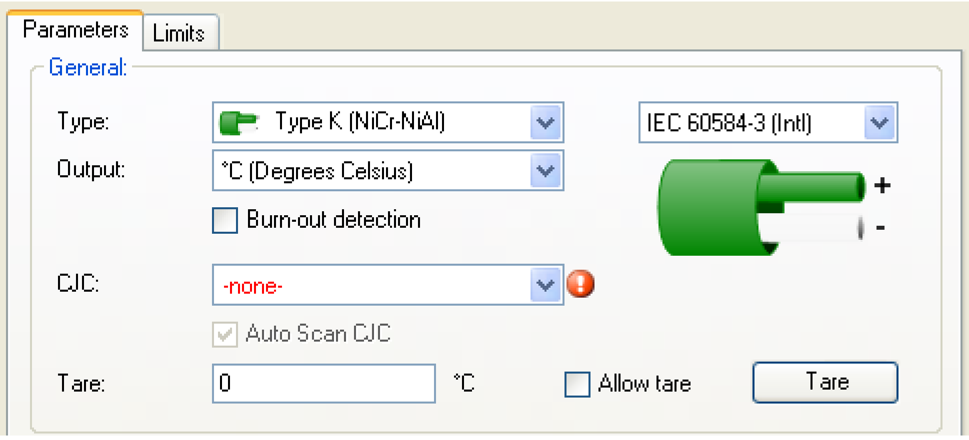
Spec. resistivity: The specific resistivity of the conductive material of the wire. The default value, 1.75 µOhms·cm, is an estimate for the resistivity of copper wire. The actual value depends on the exact type of cable.

Wire resistance (Ohms/km): The calculated resistance of the wire in Ohms per kilometer.

Wire resistance (Ohms): The calculated resistance of the specified length of wire in Ohms. This value will be used to perform the actual correction on the measurement value

Thermocouple Settings

These settings are part of the measurement point setup dialog.



The **type** of thermocouple is the first and most important parameter. All common thermocouple are supported. The color coded wiring graph shows the colors used for each thermocouple type based on several international standards.

You can check the **burn-out detection** box if you want the measuring hardware to recognize thermocouples which are burned out or not properly connected.

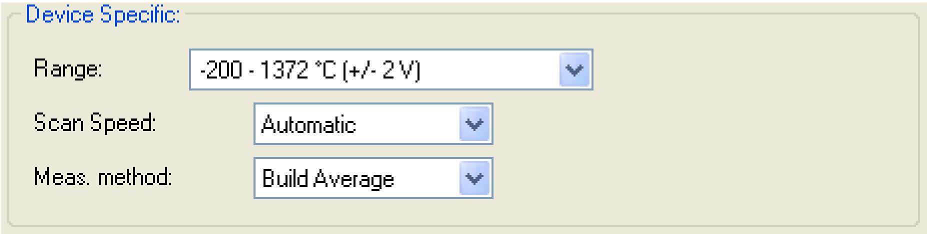
The **CJC** channel is used for Cold Junction Compensation and is usually a channel configured for Pt-100 measurement. If you do not use CJC your measurements will not be very accurate.
When selecting a CJC channel the list will show all Pt-100 channels configured. The CJC channel **must** reside in the same device (and will often be on the same card) as the ThermoCouple channel. Channels that are unsuitable to use for CJC are shown in red.

If you check the **Autoscan** box the CJC channel will automatically be measured whenever the ThermoCouple is measured. That is, it will silently and invisibly be added to every measurement group which contains the ThermoCouple channel.

The **tare** value will only be used if the **auto tare** option is checked.
You can manually set a specific tare value, if needed, or assign the measurement point to a measurement group and perform either a balance or a tare measurement to determine the value.

The formulas used to convert the signal in millivolts to degrees Celsius can be found in Appendix A.

Settings specific to Autolog 3000

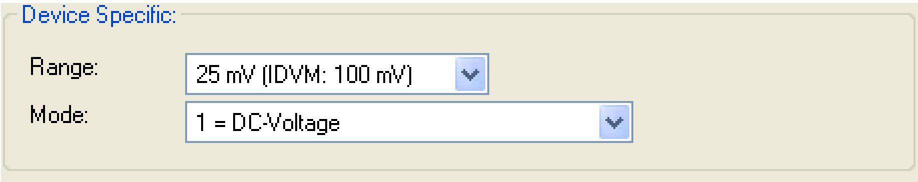


You can select a **range** for the device to use. If the selected range is not large enough to cover the range configured by the minimum and maximum value set for the channel, this field will be shown in red.

The **scan speed** determines how many measurements the Autolog 3000 card performs internally. For CA3460 cards, there are three possible scan speeds: 5 Hz, 100 Hz and 1 kHz. When set to **automatic** the speed is chosen to be the lowest available scan speed higher than or equal to the configured measurement speed.
Exception: when **burn-out detection** is active, the CA3460 always uses a scan speed of 1 kHz (scan speed can not be chosen and shows as '**automatic**').

The **meas.method** determines the way multiple measurement values are combined, in case the Autolog 3000 scan speed is higher than the configured measurement speed.

Settings specific to Autolog/Unilog 2xxx



The DC **range** of the Pt-100 measurement should be **set to 100 mV (default value) for best results**. You can, however, change the range for special purposes. It can have the following values:

Range (nominal)	Actual range ends	Resolution
10 V	-13 V to 13 V	400 µV
1 V	-1300 mV to 1300 mV	40 µV
100 mV	-130 mV to 130 mV	4 µV
25 mV	-32.5 mV to 32.5 mV	1 µV

Note: The 25 mV range is only available when using the FDVM. If a channel is measured using the IDVM, the range will automatically fall back to 100 mV (with the corresponding resolution).

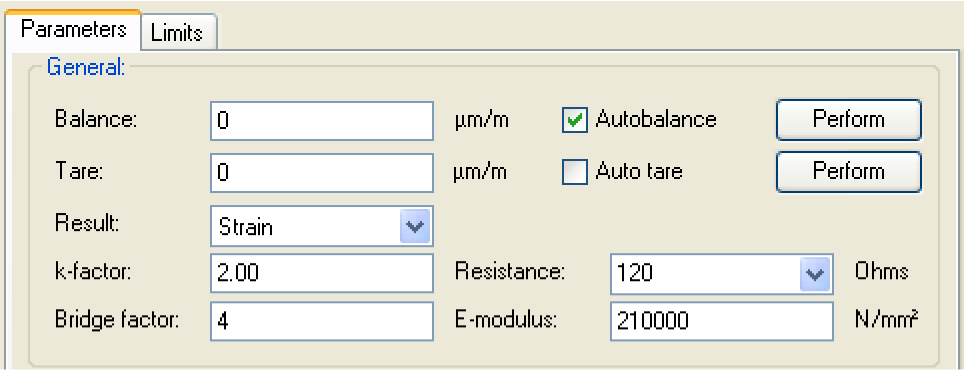
If the selected range is not large enough to cover the range configured by the minimum and maximum value set for the channel, this field will be shown in red.

You can manually set the **mode** for this channel when necessary. Use the default value '1 = DC-Voltage' (or 3 to include burn-out detection) unless you have a specific reason not to.

Note: Mode setting by software is only possible if the Autolog or Unilog has a CA-2568 mode selection card. If your device does not have this card, you must manually set the dip-switches on the cards to match the software mode setting. In this case, the software mode setting is only used as a reminder and for documentation purposes.

Strain Gauge Settings

These settings are part of the measurement point setup dialog.



The **balance** value will only be used if the **auto balance** option is checked.
The **tare** value will only be used if the **auto tare** option is checked.

You can manually set a specific balance or tare value, which will then be subtracted from all measured values, thereby making the output 0 when the value you entered is measured.
Alternatively, you can assign the measurement point to a measurement group and perform either a balance or a tare measurement to determine the value.

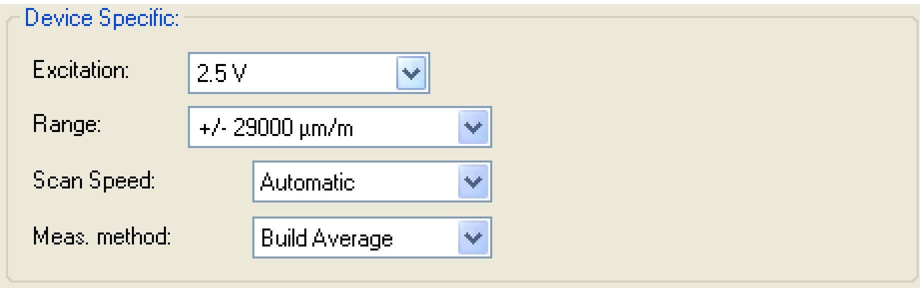
The **result type** can be set to Strain (default) in µm/m or Stress in N/mm^2.

The **E-modulus** will only be used for result type Stress, the **k-factor** for the material is always needed.

The **resistance** of the bridge is important for quarter bridge measurements, where the device needs to use a matching internal resistor to complete the bridge. It is also needed to perform calculations based on the shunt measurement in the Autolog 3000, and for the DCC measurement on the Autolog/Unilog 2xxx. In those cases where the measurement does not depend on knowledge of the resistance, this value is only used a reminder and for documentation purposes.

For half bridge and full bridge you need to enter the correct **bridge factor**.

Settings specific to Autolog 3000



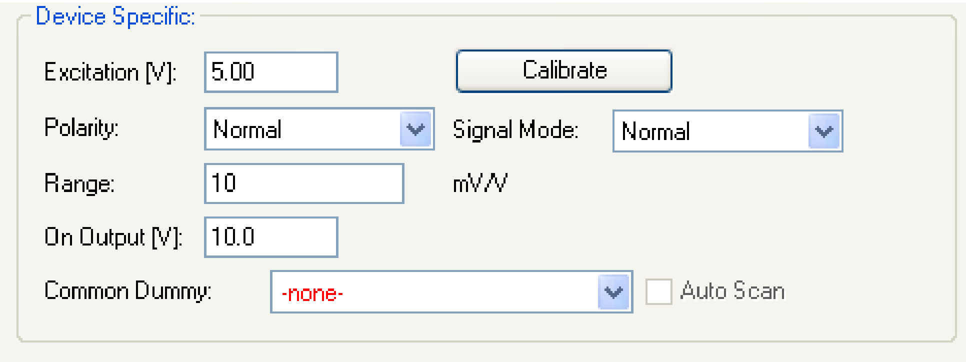
The **excitation** voltage for the bridge can be chosen between 0.5 Volts and 5 Volts. In general, the higher the excitation voltage, the better the signal-to-noise ratio. However, if you are using small strain gauges or apply them on materials that don't conduct heat very well, you should limit the excitation voltage to a lower value to avoid overheating the strain gauge.

You can set the **range** for the measurement. If the selected range is not large enough to cover the range configured by the minimum and maximum value set for the channel, this field will be shown in red.

The **scan speed** determines how many measurements the Autolog 3000 card performs internally. For CA3460 cards, there are three possible scan speeds: 5 Hz, 100 Hz and 1 kHz. When set to **automatic** the speed is chosen to be the lowest available scan speed higher than or equal to the configured measurement speed.

The **meas.method** determines the way multiple measurement values are combined, in case the Autolog 3000 scan speed is higher than the configured measurement speed.

Settings specific to CA3520/CA2CF card



The **excitation** voltage for the bridge can be chosen between 0.5 Volts and 5 Volts. In general, the higher the excitation voltage, the better the signal-to-noise ratio. However, if you are using small strain gauges or apply them on materials that don't conduct heat very well, you should limit the excitation voltage to a lower value to avoid overheating the strain gauge.

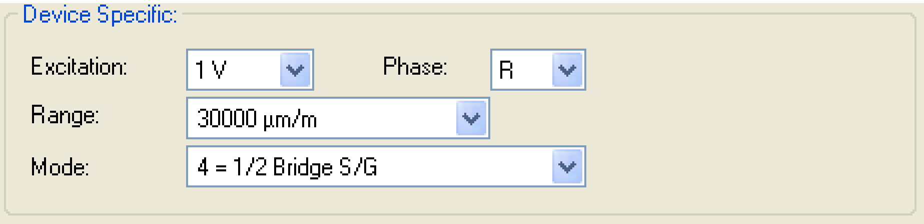
The **calibrate** button will cause the CA3520/CA2CF to measure the actual excitation voltage returned from the sensor, to correct for any loss caused by cable length.

The **polarity** and **signal mode** determine whether the signal is inverted and which component (R or C) is measured.

You can set the **range** for the measurement. If the selected range is not large enough to cover the range configured by the minimum and maximum value set for the channel, this field will be shown in red.

The **On Output** voltage determines what output voltage on the BNC-connector corresponds with the chosen range. If the analog output is not used, this should be left at 10 V.

Settings specific to Autolog/Unilog 2xxx with CA2005 (Carrier Frequency)



The **excitation** voltage for the bridge can be 1 Volts or 5 Volts. In general, 5 Volts will give a better signal-to-noise ratio. If you are using small strain gauges or apply them on materials that don't conduct heat very well, you should choose 1 Volts to avoid overheating the strain gauge.

The default **phase** (R) will suffice in most cases.

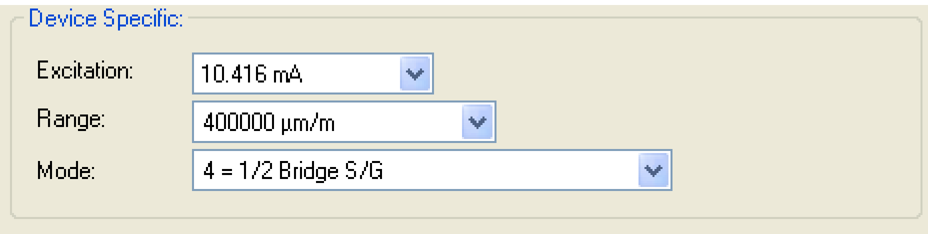
The **range** can be set to the following values:

Range (nominal)	Range end	Resolution
300000 µm/m	600000 µm/m	5 µm/m
30000 µm/m	60000 µm/m	0.5 µm/m
3000 µm/m	6000 µm/m	0.05 µm/m

For quarter bridges, the non linearity of the Wheatstone Bridge is automatically corrected. The formula used to do this can be found in Appendix A.

For half bridge and full bridge, the correct **mode** will automatically be chosen. For **quarter bridge**, make sure you choose the correct **mode**. Modes 8, 9 and 10 will use 120 Ohms, 240 Ohms and 350 Ohms resistance for internal bridge completion. Modes 11 to 15 will use 1 of 5 common dummies. To use a common dummy mode, you must connect an external common dummy resistor of an appropriate value.
Note: Mode setting by software is only possible if the Autolog or Unilog has a CA-2568 mode selection card. If your device does not have this card, you must manually set the dip-switches on the cards to match the software mode setting. In this case, the software mode setting is only used as a reminder and for documentation purposes.

Settings specific to Autolog/Unilog 2xxx with VCA2055 (Constant Current)



The **excitation** current for the bridge can be 20.833 mA, 10.416 mA, 5.208 mA, 7.143 mA, 3.571 mA or 1.786 mA. The first three values (20.8, 10.4 and 5.2) are usually used for 120 or 240 Ohms resistance, the last three values (7.1, 3.5 and 1.8) for 350 Ohms. You are free to choose other combinations if desired.

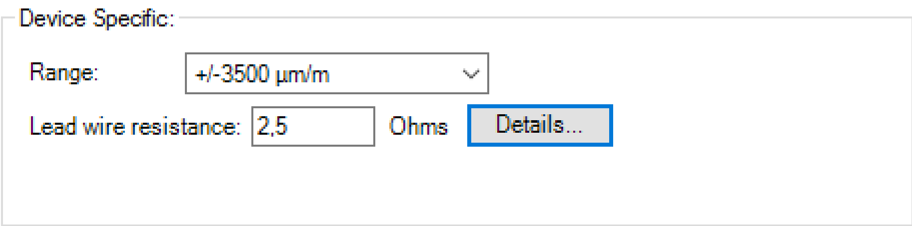
There are three possible **ranges** for the measurement. The first two are always available, the last one only when measuring using an FDVM. If you choose the third (smallest) range, the second range will automatically be used for IDVM measurements.
The actual range depends on the chosen **excitation** and **resistance**. The presented ranges are nominal ranges and are automatically updated when excitation or resistance is changed.
The following equations apply to all ranges:

Range end = 1.3 x nominal range
Resolution = 0.00004 x nominal range

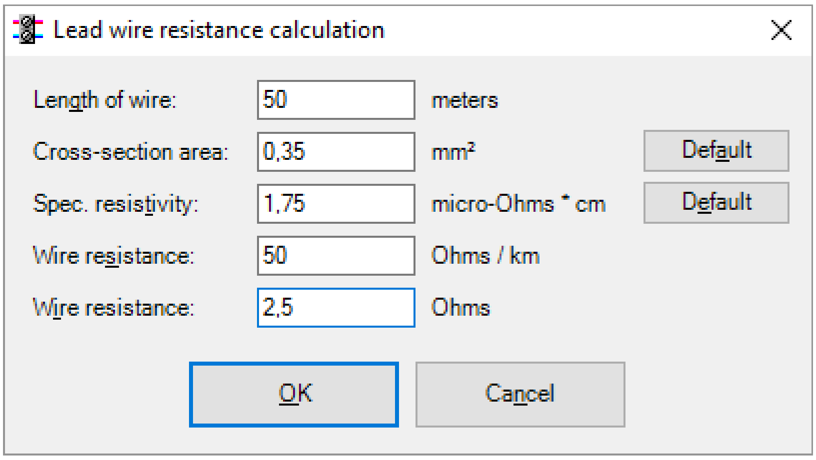
For quarter bridges, the non linearity of the Wheatstone Bridge is automatically corrected. The formula used to do this can be found in Appendix A.

For half bridge and full bridge, the correct mode will automatically be chosen. For **quarter bridge**, make sure you choose the correct **mode**. Modes 8, 9 and 10 will use 120 Ohms, 240 Ohms and 350 Ohms resistance for internal bridge completion. Modes 11 to 15 will use 1 of 5 common dummies. To use a common dummy mode, you must connect an external common dummy resistor of an appropriate value.
Note: Mode setting by software is only possible if the Autolog or Unilog has a CA-2568 mode selection card. If your device does not have this card, you must manually set the dip-switches on the cards to match the software mode setting. In this case, the software mode setting is only used as a reminder and for documentation purposes.

Settings specific to StrainBUSter with 1/4 bridge Strain Gauge



For StrainBUSter, the connection to a 1/4-bridge strain gauge should be compensated for the resistance of the lead wires to the sensor. This can be configured using the "**Details...**"-button, which opens the "Lead wire resistance calculation" dialog box:



Use this dialog to calculate the lead wire resistance. When measuring quarter bridge strain gauges or Pt-100 elements using 3 wires, the resistance of the lead wires can have a significant impact on the measured value. To reduce the error, the software can correct the measurement value for an estimated lead wire resistance.
BEWARE: If the lead wire resistance you enter is more than 2 times higher than the real lead wire resistance, you will increase instead of decrease the error! When in doubt, err on the side of caution: use a lower resistance value.

The individual items in this configuration page described:

Length of wire: The length of the lead wire in meters.

Cross-section area: The cross-section area of the conductive material of the wire in mm².

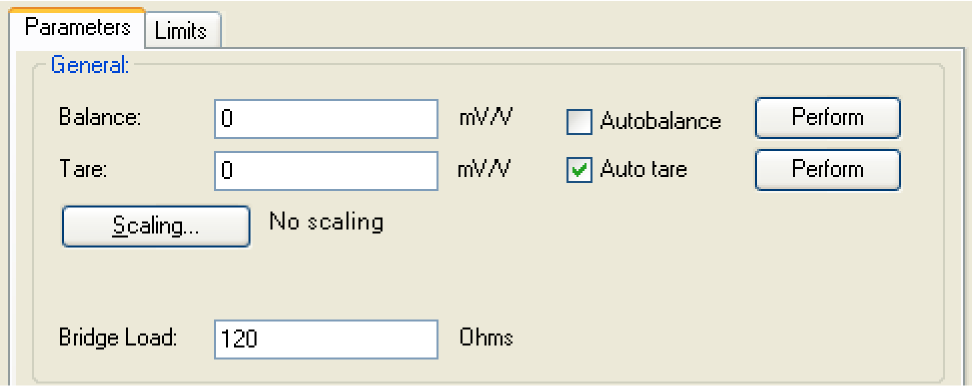
Spec. resistivity: The specific resistivity of the conductive material of the wire. The default value, 1.75 µOhms·cm, is an estimate for the resistivity of copper wire. The actual value depends on the exact type of cable.

Wire resistance (Ohms/km): The calculated resistance of the wire in Ohms per kilometer.

Wire resistance (Ohms): The calculated resistance of the specified length of wire in Ohms. This value will be used to perform the actual correction on the measurement value.

Transducer Settings

These settings are part of the measurement point setup dialog.



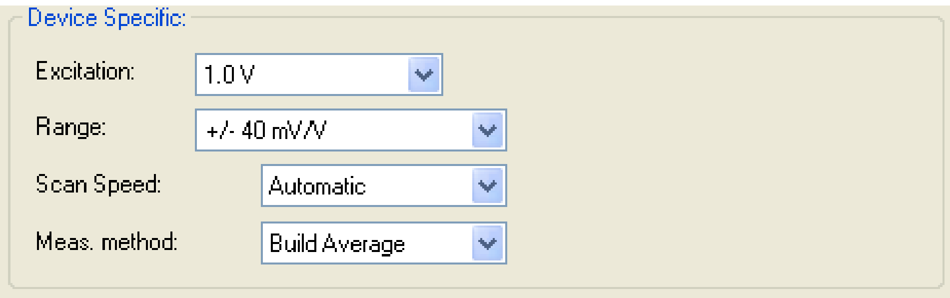
The **balance** value will only be used if the **auto balance** option is checked.
The **tare** value will only be used if the **auto tare** option is checked.

You can manually set a specific balance or tare value, which will then be subtracted from all measured values, thereby making the output 0 when the value you entered is measured.
Alternatively, you can assign the measurement point to a measurement group and perform either a balance or a tare measurement to determine the value.

You can scale in the input signal by clicking the scaling button. You can convert the mV/V signal to a more meaningful value by means of linear interpolation or a polynomial and setting appropriate **units** for the channel.

The **bridge load** is needed for shunt measurements on the Autolog 3000 and for constant current based measurements (VCA2055) on the Autolog/Unilog 2xxx. In those cases where the measurement does not depend on knowledge of the bridge load, this value is only used a reminder and for documentation purposes.

Settings specific to Autolog 3000



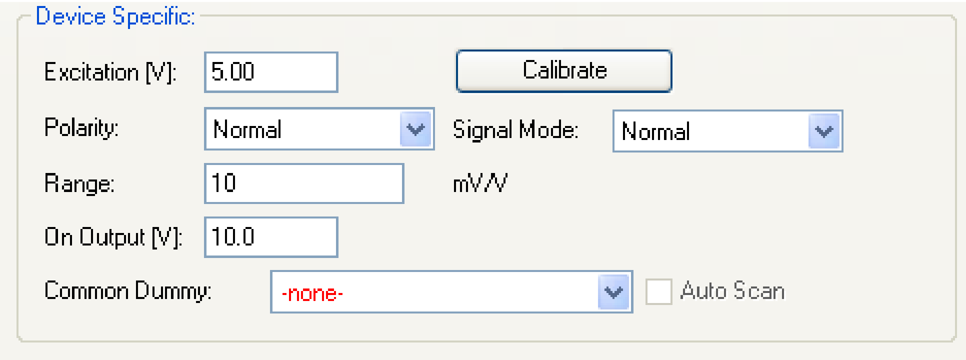
The **excitation** voltage for the bridge can be chosen between 0.5 Volts and 5 Volts. In general, the higher the excitation voltage, the better the signal-to-noise ratio. However, if you are using small strain gauges or apply them on materials that don't conduct heat very well, you should limit the excitation voltage to a lower value to avoid overheating the strain gauge.

You can set the **range** for the measurement. If the selected range is not large enough to cover the range configured by the minimum and maximum value set for the channel, this field will be shown in red.

The **scan speed** determines how many measurements the Autolog 3000 card performs internally. For CA3460 cards, there are three possible scan speeds: 5 Hz, 100 Hz and 1 kHz. When set to **automatic** the speed is chosen to be the lowest available scan speed higher than or equal to the configured measurement speed.

The **meas.method** determines the way multiple measurement values are combined, in case the Autolog 3000 scan speed is higher than the configured measurement speed.

Settings specific to CA3520/CA2CF card



The **excitation** voltage for the bridge can be chosen between 0.5 Volts and 5 Volts. In general, the higher the excitation voltage, the better the signal-to-noise ratio. Depending on the type of transducer however, higher excitation voltages may not be allowed.

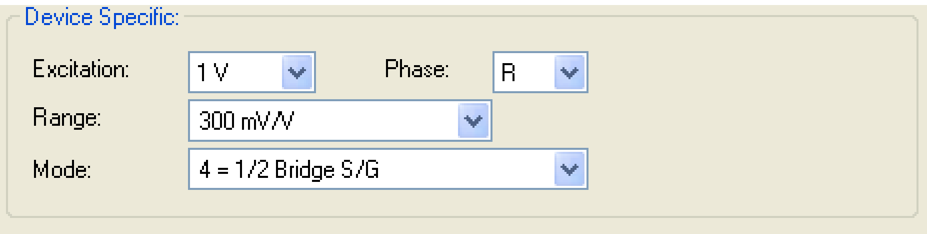
The **calibrate** button will cause the CA3520/CA2CF to measure the actual excitation voltage returned from the sensor, to correct for any loss caused by cable length.

The **polarity** and **signal mode** determine whether the signal is inverted and which component (R or C) is measured.

You can set the **range** for the measurement. If the selected range is not large enough to cover the range configured by the minimum and maximum value set for the channel, this field will be shown in red.

The **On Output** voltage determines what output voltage on the BNC-connector corresponds with the chosen range. If the analog output is not used, this should be left at 10 V.

Settings specific to Autolog/Unilog 2xxx with CA2005 (Carrier Frequency)



The **excitation** voltage for the transducer can be 1 Volts or 5 Volts. Be careful not to choose the excitation voltage too high! The measurement value is normalized, so the choice between 1 and 5 Volts does not influence the measurement value.

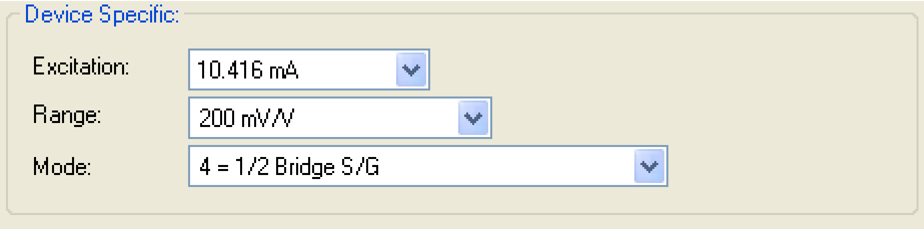
The default **phase** (R) will suffice in most cases.

The **range** can be set to the following values:

Range end	Resolution
300 mV/V	2.5 µV/V
30 mV/V	0.25 µV/V
3 mV/V	0.025 µV/V

For half bridge and full bridge, the **mode** will automatically be chosen but can be modified manually if needed.
Note: Mode setting by software is only possible if the Autolog or Unilog has a CA-2568 mode selection card. If your device does not have this card, you must manually set the dip-switches on the cards to match the software mode setting. In this case, the software mode setting is only used as a reminder and for documentation purposes.

Settings specific to Autolog/Unilog 2xxx with VCA2055 (Constant Current)



The **excitation** current for the bridge can be 20.833 mA, 10.416 mA, 5.208 mA, 7.143 mA, 3.571 mA or 1.786 mA. The first three values (20.8, 10.4 and 5.2) are usually used for 120 or 240 Ohms resistance, the last thee values (7.1, 3.5 and 1.8) for 350 Ohms. You are free to choose other combinations if desired.

The **range** can be set to the following values:

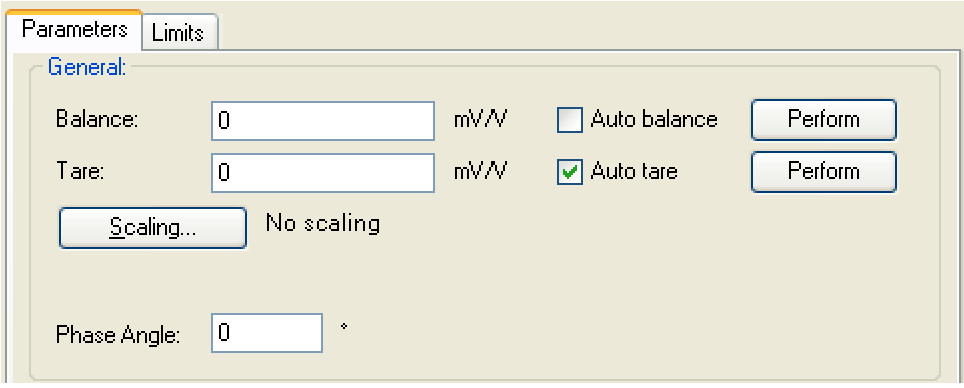
Range (nominal)	Actual range ends	Resolution
1 V	-1300 mV to 1300 mV	40 µV
100 mV	-130 mV to 130 mV	4 µV
25 mV	-32.5 mV to 32.5 mV	1 µV

Note: The 25 mV range is only available when using the FDVM. If a channel is measured using the IDVM, the range will automatically fall back to 100 mV (with the corresponding resolution).

For half bridge and full bridge, the **mode** will automatically be chosen but can be modified manually if needed.
Note: Mode setting by software is only possible if the Autolog or Unilog has a CA-2568 mode selection card. If your device does not have this card, you must manually set the dip-switches on the cards to match the software mode setting. In this case, the software mode setting is only used as a reminder and for documentation purposes.

LVDT Settings

These settings are part of the measurement point setup dialog.



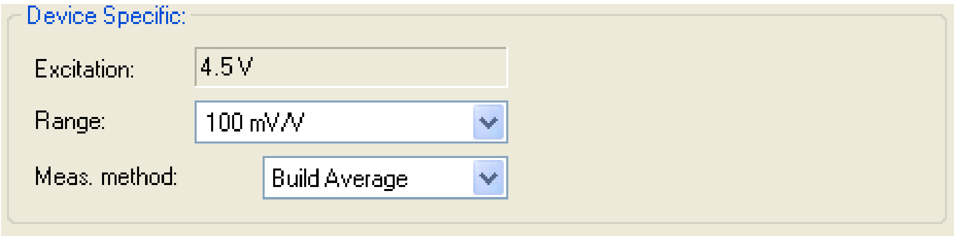
The **balance** value will only be used if the **auto balance** option is checked.
The **tare** value will only be used if the **auto tare** option is checked.

You can manually set a specific balance or tare value, which will then be subtracted from all measured values, thereby making the output 0 when the value you entered is measured.
Alternatively, you can assign the measurement point to a measurement group and perform either a balance or a tare measurement to determine the value.

You can scale in the input signal by clicking the scaling button. You can convert the mV/V signal to a more meaningful value by means of linear interpolation or a polynomial and setting appropriate **units** for the channel.

The **phase angle** is needed on the Autolog 3000 to correct the lowered measurement value as the phase angle increases. For example, a phase angle of 10° will result in a measured value that is about 98.5% of the actual value, so the value is multiplied by 1.015 to correct for this discrepancy.

Settings specific to Autolog 3000



The **excitation** voltage for LVDT measurements is fixed at 4.5 V and is only shown for documentation purposes.

You can set the **range** for the measurement. If the selected range is not large enough to cover the range configured by the minimum and maximum value set for the channel, this field will be shown in red.

The **scan speed** determines how many measurements the Autolog 3000 card performs internally. For CA3460 cards, there are three possible scan speeds: 5 Hz, 100 Hz and 1 kHz. When set to **automatic** the speed is chosen to be the lowest available scan speed higher than or equal to the configured measurement speed.

The **meas.method** determines the way multiple measurement values are combined, in case the Autolog 3000 scan speed is higher than the configured measurement speed.

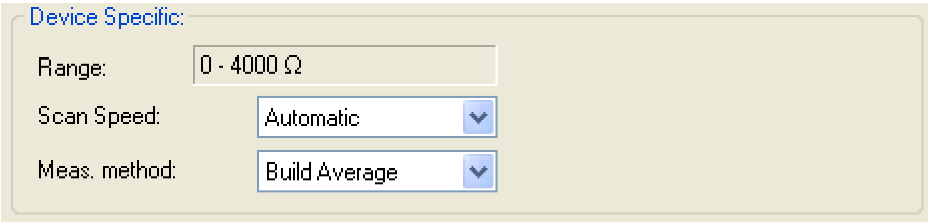
Settings specific to Autolog/Unilog 2xxx

The **sensitivity** of the LVDT measurement is a factory setting made by Peekel Instruments on the card. You should match this factory setting in the software. The default value for the sensitivity is 100.

The **mode** setting is not used for LVDT measurements and therefore can not be altered.

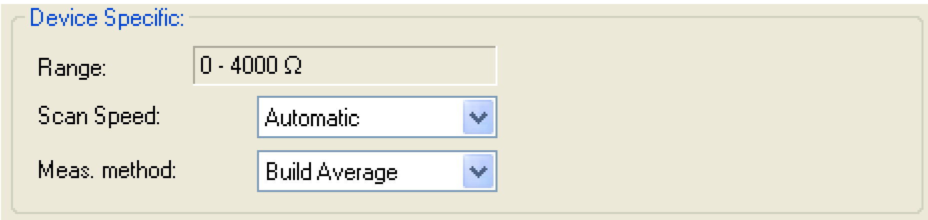
Resistance Settings

These settings are part of the measurement point setup dialog.



There are no specific settings to be made for a resistance measurement. You can scale in the input signal by clicking the scaling button. You can convert the resistance signal to a more meaningful value by means of linear interpolation or a polynomial and setting appropriate **units** for the channel.

Settings specific to Autolog 3000



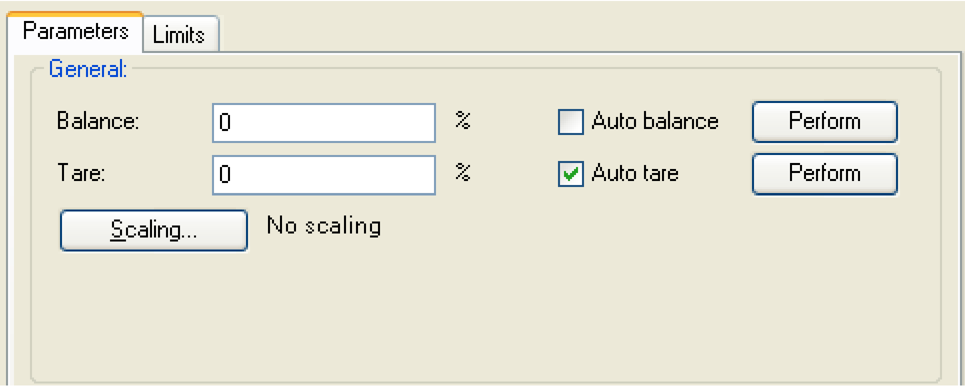
The **range** is fixed and only shown for documentation purposes.

The **scan speed** determines how many measurements the Autolog 3000 card performs internally. For CA3460 cards, there are three possible scan speeds: 5 Hz, 100 Hz and 1 kHz. When set to **automatic** the speed is chosen to be the lowest available scan speed higher than or equal to the configured measurement speed.

The **meas.method** determines the way multiple measurement values are combined, in case the Autolog 3000 scan speed is higher than the configured measurement speed.

Potentiometric Settings

These settings are part of the measurement point setup dialog.

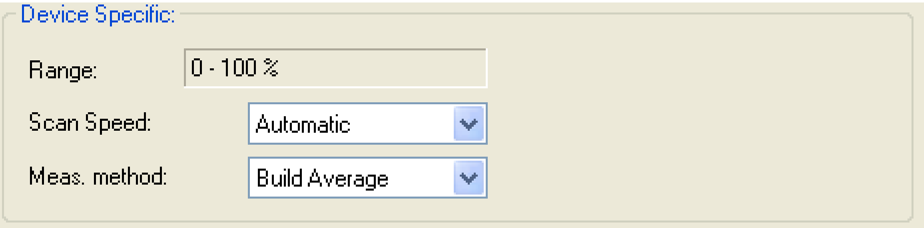


The **balance** value will only be used if the **auto balance** option is checked.
The **tare** value will only be used if the **auto tare** option is checked.

You can manually set a specific balance or tare value, which will then be subtracted from all measured values, thereby making the output 0 when the value you entered is measured.
Alternatively, you can assign the measurement point to a measurement group and perform either a balance or a tare measurement to determine the value.

You can scale in the input signal by clicking the scaling button. You can convert the %-value to a more meaningful value by means of linear interpolation or a polynomial and setting appropriate **units** for the channel.

Settings specific to Autolog 3000



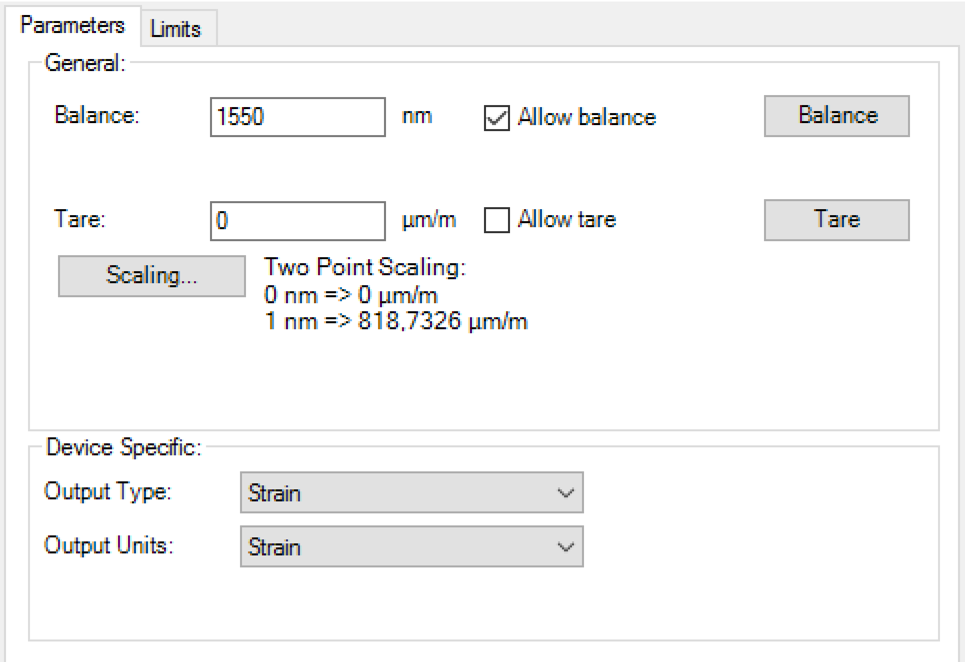
The **range** is fixed and only shown for documentation purposes.

The **scan speed** determines how many measurements the Autolog 3000 card performs internally. For CA3460 cards, there are three possible scan speeds: 5 Hz, 100 Hz and 1 kHz. When set to **automatic** the speed is chosen to be the lowest available scan speed higher than or equal to the configured measurement speed.

The **meas.method** determines the way multiple measurement values are combined, in case the Autolog 3000 scan speed is higher than the configured measurement speed.

FBG Settings

These settings are part of the measurement point setup dialog.



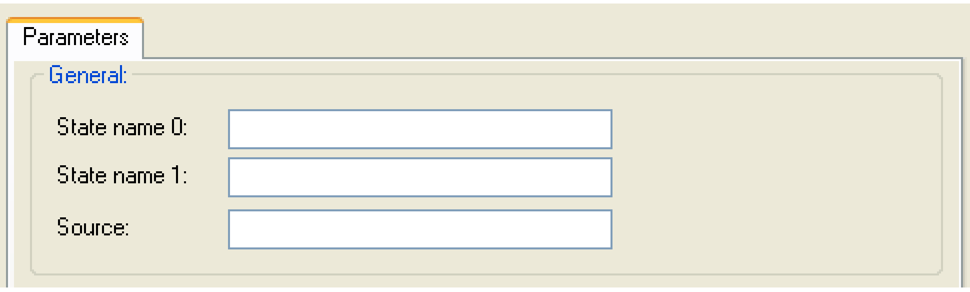
For Fiber Bragg Grating sensors, the raw input value is always measured in nm (nanometers), and the base wavelength of the sensor is set as the default balance value.

The lower section (Device Specific) gives the option to select standard scaling options to convert the measured signal to either strain or temperature.

When **Strain** is selected, the output can be expressed as strain in µm/m or stress in N/mm^2. When **Temperature** is selected, the output can be expressed as degrees Celsius, Fahrenheit or Kelvin. The exact conversion (linear scaling) depends on the base wave length and is shown in the upper part of the dialog.

Digital I/O Settings

These settings are part of the measurement point setup dialog.



Digital inputs and outputs use the measurement values 0 and 1 to represent active and inactive. You can use the **state name** fields to assign a meaningful description to these values.

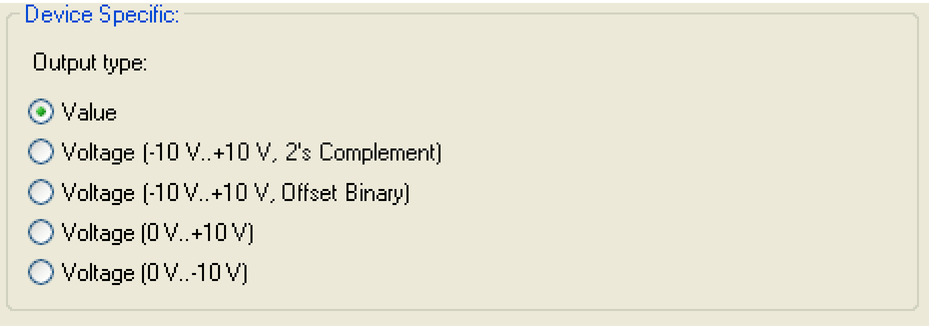
For digital outputs, a **source** measurement point can be specified. Every time the value of this point changes, it is evaluated. If found to be less than 0.5, the digital output will be made inactive (0) otherwise, the digital output will be made active (1).

An alternative way to set or clear digital outputs is to use triggers that set the output value in response to a specific (set of) events. If this approach is used, the **source** field can be left blank.

Analog Output Settings

These settings are part of the measurement point setup dialog.

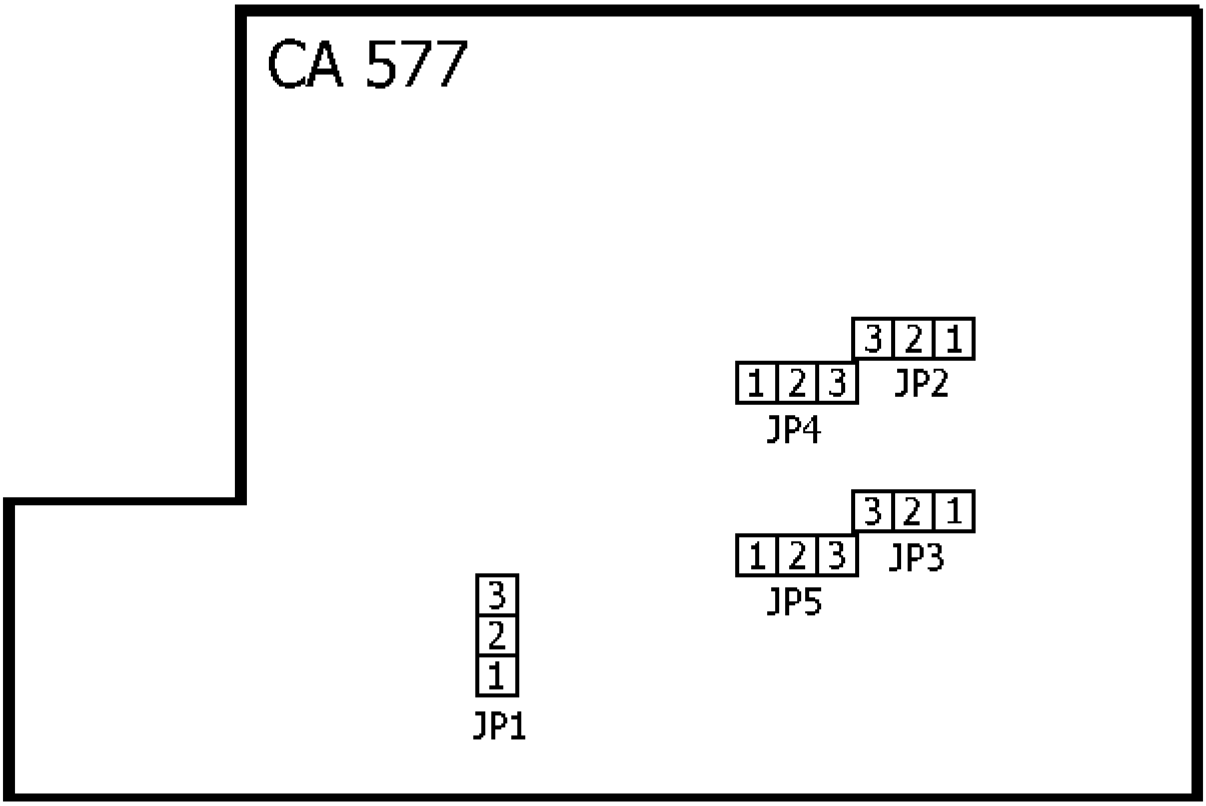
Settings specific to Autolog/Unilog 2xxx



You can select whether the **analog output value** in the software should be presented as a raw value (0...255 for 8-bit outputs or 0...16383 for 14-bit outputs) or as an output voltage.

For 14-bit outputs, there are 4 different ways in which the raw value can be converted to an output voltage. **The setting you choose here should match the one configured by jumpers on the card.**

If you choose one of the Voltage ranges on the 14-bit output, please refer to the table and picture below to set the jumpers on the CA-577 analog output card.

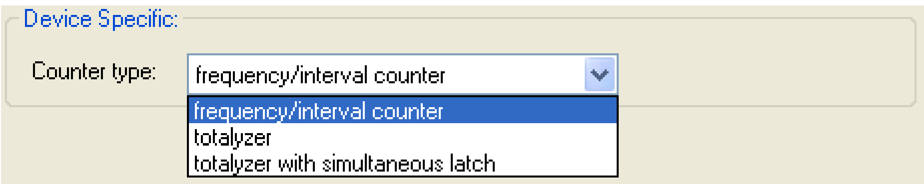


Jumper	-10 - +10 V 2' complement	-10 - +10 V offset binary	0 - +10 V	0 - -10 V
JP1	2-3	2-1	2-1	2-1
JP2	2-1	2-1	2-1	2-3
JP3	2-1	2-1	2-1	2-3
JP4	2-3	2-3	2-1	2-1
JP5	2-3	2-3	2-1	2-1

Counter Settings

These settings are part of the measurement point setup dialog.

Settings specific to Autolog/Unilog 2xxx



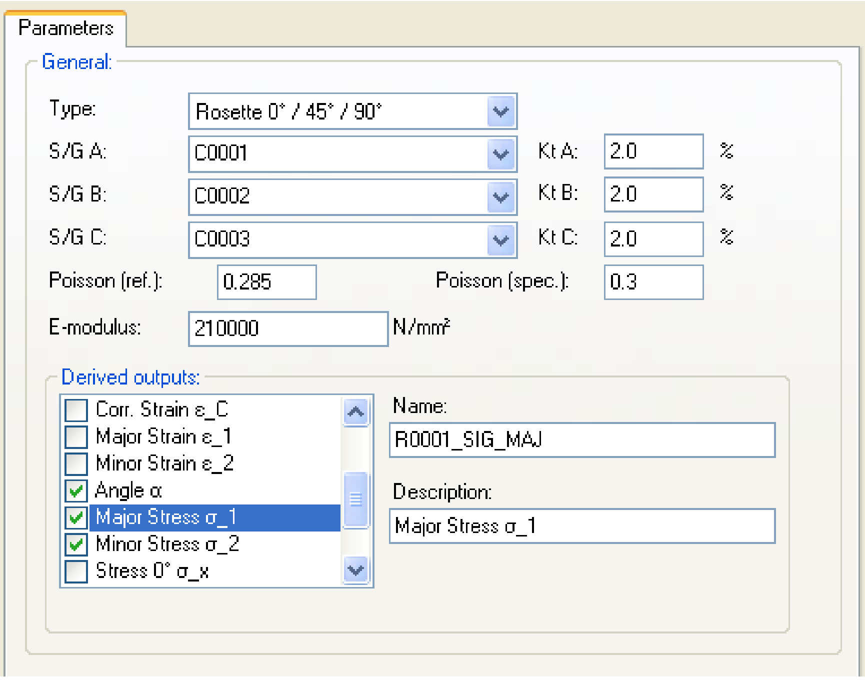
The counters can be configured in two ways: **frequency/interval or totalyzer**. Jumpers on the card do the actual configuration; the selected counter type should match the card's configuration.

Frequency/interval is used to either measure the frequency in which pulses arrive, or the interval between the pulses, depending on the hardware setting.

The totalyzer is used to keep a running count of the number of pulses, using a 16-bit counter. If the count goes over 65535 it rolls back to 0. For this type of counter, you can choose a special mode 'totalyzer with latch'. Choose this option to speed up measurement when a large number of counters are measured in a single measurement group, together with only a small amount (or none) of other types of channels. **When in doubt, choose** the normal **totalyzer**, without latch.

Rosette Settings

These settings are part of the measurement point setup dialog.



The rosette channel calculates strains and stresses for rosettes made from two or three strain gauges.

First you must specify the type of rosette, determined by the amount and placing of the individual strain gauges:

Type 0° / 90°: Two strain gauges, S/G A for 0° and S/G B for 90°.
Type 0° / 45° / 90°: Three strain gauges, S/G A for 0°, S/G B for 45° and S/G B for 90°.
Type 0° / 60° / 120°: Three strain gauges, S/G A for 0°, S/G B for 60° and S/G B for 120°.

You can select the individual inputs for the strain gauge from the list boxes S/G A, S/G B and S/G C or type the channel names yourself.

Beware: only measurement points that display their value in μm/m (strain) are valid inputs for rosettes. Do not set the individual strain gauge inputs to result type 'stress'. If you need the stress values for the individual strain gauges, use the available rosette outputs instead.

The constants Poisson (ref.), Poisson (spec.), Kt (transverse sensitivity) and E-modulus can also be entered. Note that the Poisson (spec.) and E-modulus are only used for stress calculations and are not needed or used for strain results.

The rosette channel itself does not generate any output value. To have the calculations stored you will need to create one or more rosette output channels. From the list box under '**derived outputs**', you can check which outputs you want to see.
For each output you can set a specific name and description. If you use the '*' sign anywhere in the output name, it will be replaced by the name of the rosette channel.

To remove an output channel, simply uncheck it in the list box.

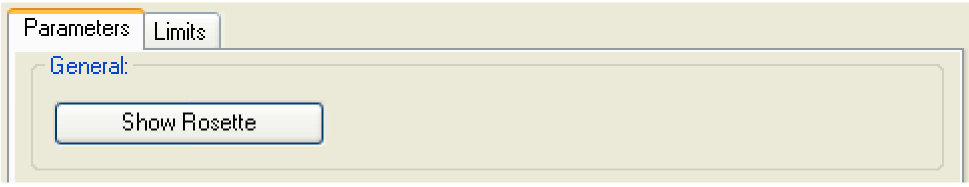
Depending on the type of rosette, you can create the following output channels:

Stress A	Stress for strain gauge A
Stress B	Stress for strain gauge B
Stress C	Stress for strain gauge C
Corr. Strain A	Strain for strain gauge A after transverse sensitivity correction.
Corr. Strain B	Strain for strain gauge B after transverse sensitivity correction.
Corr. Strain C	Strain for strain gauge C after transverse sensitivity correction.
Major Strain	Major strain, angle Alpha to strain A.
Minor Strain	Minor strain, perpendicular to the major strain.
Alpha	Angle in degrees (-90 … 90) between Strain A and Major Strain.
Major Stress	Major stress, angle Alpha to strain A.
Minor Stress	Minor stress, perpendicular to the major stress.
Stress 0°	Stress in the direction of strain A
Stress 90°	Stress perpendicular to strain A
Shear Stress	Shear stress belonging to Stress 0° and Stress 90°
Equivalent Stress	Equivalent uniaxial stress for the major/minor stresses (von Mises).

For further information on the rosette outputs and the corresponding calculations, refer to Appendix A.

Rosette Output Settings

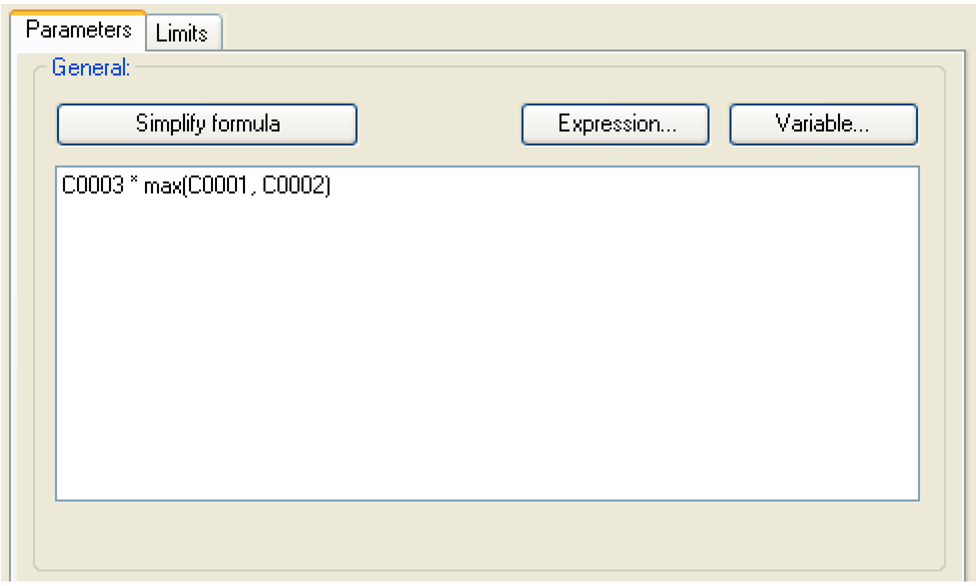
These settings are part of the measurement point setup dialog.



The individual rosette outputs do no have any specific settings, all settings are controlled by the rosette item that the output is linked to.

Virtual Settings

These settings are part of the measurement point setup dialog.



A virtual point can combine the results of any number of other measurement points (including other virtual points) using a user specified formula.
If you add a virtual point to one or more measurement groups, it will be evaluated for every measurement (measurement mode). If you do not add a virtual point to any measurement groups, it will automatically be evaluated as soon as the value of any of the channels used in its formula changes (automatic mode).

To create a formula for the virtual channel, you can use **'Expression'** button to select from the available functions and insert it into the current formula. You can also directly type the formula if you wish.

To refer to an existing channel, you must type its name, or click on the '**Variable'** button to select one from a list. For functions like 'MAX' or 'AVG', you can also select multiple variables from the list to use as parameters.

To refer to the previous value of a channel, type the channel name followed by a backquote (`). You can use this feature to monitor changes in a value. For example: **M1 - M1`** will calculate the difference between the current and previous value of measurement point M1.

Some example formulas, where M1 and M2 are the names of two existing channels:

Formula	**Explanation**
2 * M1	Multiply M1 by a factor 2
max(M1, M2)	Determine the maximum of channel M1 and M2
t_max(M1, 10)	Determine the maximum of every 10 consecutive values of M1
(M1 + 2) * M2	Multiply M1+2 by M2
M1 - M1`	Calculates the difference between the current and previous value of M1

Press the **'Simplify formula'** button to simplify the current formula. This will remove all extraneous brackets and evaluate constant expressions. For example, the formula '**SQRT(4) +1**' will reduce to '**3**'.

Functions for Virtual Points

This list shows all functions that can be used in the formulas of virtual points with a short explanation. For all functions based on boolean logic, the result is **1** for **true** and **0** for **false**.

-	Subtraction, for example 3-2 = 1
+	Addition, for example 3+2 = 5
*	Multiplication, for example 3*2 = 6
/	Division, for example 3/2 = 1.5
^	Power, for example 3^2 = 9
<	Less than, for example 3 < 2 = 0 (false), 2 < 3 = 1 (true)
>	Greater than, for example 2 > 3 = 0 (false), 3 > 2 = 1 (true)
<=	Less than or equals
>=	Greater than or equals
=	Equals, for example '2 = 3' = 0 (false), '2 = 2' = 1 (true)
<>	Does not equal, for example '2 <> 3' = 1 (true), '2 <> 2' = 0 (false)
<<	Bit shift left. For integers: 'a << b' = 'a * (2^b)'
>>	Bit shift right. For integers: 'a >> b' = 'a / (2^b)'
ABS	Absolute value, for example ABS(3) = 3, ABS(-2) = 2.
ACOS	Arccosine in radians, output range 0 ... PI.
AND	Logical and, for example 1 AND 1 = 1 (true), 1 AND 0 = 0 (false)
ASIN	Arcsine in radians, output range -PI/2 ... PI/2.
ATAN	Arctangent in radians, output range -PI/2 ... PI/2.
ATAN2	Arctangent in radians, output range -PI/2 ... PI/2, for example ATAN2(a, b) = arctangent of a/b

AVG	Average of any number of values, for example AVG(1) = 1, AVG(1, 2, 3, 4) = 2.5
BIT	Value of a specific bit in a number, eg BIT(5,0) = 1, BIT(5,1) = 0. The value 5 has bitpattern 000101, therefore bit 0 (least significant bit) equals 1 and bit 1 equals 0
BITAND	Bitwise and of two values, eg 5 BITAND 3 = 1. Bitpattern for 5 = 000101, for 3 = 000011, therefore only bit 0 is 1 in both cases.
BITNOT	Bitwise negation of a value, using 32 bits (!)
BITOR	Bitwise or of two values, eg 5 BITOR 3 = 7. Bitpattern for 5 = 000101, for 3 = 000011, therefore the output bit pattern is 000111 (equals 7).
BYTE	Construct a byte from a bitpattern, for example BYTE(1, 0, 0) = 4, BYTE(1, 0, 1) = 5.
CEIL	Round a number up, for example CEIL(2.7) = 3, CEIL(2.1) = 3.
COS	Cosine (radians)
COSH	Hyperbolic Cosine (radians)
DATE(0)	Number of days since 1/1/1980 for current date
DATE(1)	Day of month (1 ... 31) for current date
DATE(2)	Month (1 ... 12) for current date
DATE(3)	Year (for example 2000) for current date
DATE(4)	Daynumber in current year (1 ... 366) for current date
DATE(5)	Day of week (0 = Sunday, 1 = Monday, 2 = Tuesday, ..., 6 = Saturday)
EXP	Exponential, EXP(a) = E ^ a
FLOOR	Round a number down, for example FLOOR(2.7) = 2, FLOOR(2.1) = 2.
IF()	if-then selection. For example, 'IF(a, b, c) should be read as 'if a is nonzero then b else c'. Therefore 'IF(0, 3, 2)' = 2, 'IF(1, 3, 2)' = 3.
LOG	Natural logarithm (base E)
LOG10	Logarithm (base 10)
MAX	Maximum of any number of values, for example MAX(1) = 1, MAX(1, 4, 3, 2) = 4
MIN	Minimum of any number of values, for example MIN(1) = 1, MIN(1, 4, 3, 2) = 1
MOD	Modulo (remainder after division), for example 7 MOD 3 = 1 (remainder of 7 after division by 3 equals 1)
NOT	Logical not, for example 'NOT 0' = 1 (true), 'NOT 5' = 0 (false)
OR	Logical or, for example 1 OR 1 = 1 (true), 1 OR 0 = 1 (true)
RAND()	Random number between 0 and 1.
ROUND	Round a number to the nearest integer, for example ROUND(2.7) = 3, ROUND(2.1) = 2
SIN	Sine (radians)
SINH	Hyperbolic sine (radians)
SQRT	Square root, for example SQRT(4) = 2
TAN	Tangent (radians)
TANH	Hyperbolic tangent (radians)
TIME(0)	Current time in milliseconds since midnight (0 ... 86400000)
TIME(1)	Milliseconds (0 ... 999) for current time
TIME(2)	Seconds (0 ... 59) for current time
TIME(3)	Minutes (0 ... 59) for current time
TIME(4)	Hours (0 ... 23) for current time
XOR	Exclusive or, eg 5 XOR 3 = 6. Bitpattern for 5 = 000101, for 3 = 000011, therefore the output bit pattern is 000110 (equals 6).
FILTER	Configurable Filter function
DELTA(v)	Difference between current and previous value of the expression 'v'. Can also be written using the backquote (`) operator as: v - v`
DELTA_T(v)	Differential of expression 'v' over time. Is equal to 'DELTA(v) / DELTA_T(v)'.
COUNTER	The function COUNTER(var, highlevel, lowlevel, interval) will count the number of cycles that input 'var' makes during each 'interval' seconds. A cycle is counted when 'var' drops below 'lowlevel' and then goes over 'highlevel'. A new value (count) is produced each time the interval expires.
COUNTERF	The function COUNTERF(var, highlevel, lowlevel, cyclecount [, interval]) will calculate the frequency (in Hz) of the cycles that input 'var' makes. A cycle is counted when 'var' drops below 'lowlevel' and then goes over 'highlevel'. After

	each 'cyclecount' cycles, the function produces a new frequency-value. The optional 'interval' parameter puts a limit on the amount of time to wait for the required number of cycles; a value of 0 is produced when the interval expires before 'cyclecount' cycles are counted.
STOPWATCH(v)	The function STOPWATCH(v) is a special function that shows a running count of seconds passed since it was triggered. The parameter 'v' can be between 1 and 4, allowing for four different stopwatches. For more information, refer to the stopwatch topic.

The following functions are special, because they are based on a number of consecutive measurements from a single source. Their output changes each time this number of measurements has been made and remains constant in between. The number of measurements specified in the function must a constant between 1 and 100.

T_AVG	Average over a number of measurements. For example: T_AVG(CHN, 5) will produce the average for each group of 5 measurements for CHN.
T_MAX	Maximum over a number of measurements. For example: T_MAX(CHN, 5) will produce the maximum value for each group of 5 measurements for CHN.
T_MIN	Minimum over a number of measurements. For example: T_MIN(CHN, 5) will produce the minimum value for each group of 5 measurements for CHN.

The following functions can be used to monitor different aspects of the PC and the archiving of data.

CPU(0)	Monitors CPU load in %. In a multiprocessor system this is the average load of all CPUs. Each use of this function shows the average CPU load during the time since the previous use.
CPU(1)	Monitors CPU temperature on Kontron mainboards using the JIDA API.
DISKINFO	Show disk usage information. The first parameter specifies the disk to check: 1 = A:, 2 = B:, 3 = C:, etc. The second parameter is 0 for free space in bytes, 1 for total space in bytes. Examples: DISKINFO(3,0) = Free space in bytes on C: DISKINFO(5,1) = Total space in bytes on E:
ARCHIVING	Shows statistics on measurement data stored to disk. This function shows the amount of data that was actually written to the measurement archive of the configuration. The first parameter specifies the device number: 0 = total over all devices in the configuration, 1 and up = a specific device (follows the order as visible in the overview tree). The second parameter is 0 for the number of measurement points for which data was stored, or 1 for the number of measurement values stored per second. Examples: ARCHIVING(0, 0) = Total number of measurement points for which data gets stored to the archives. ARCHIVING(2, 1) = Number of measurement values per second stored to the archives for device #2. Note that there will always be a delay visible when using this function. This is caused by the fact that it always averages over a period of 2 seconds, and the fact that measurement data gets stored to the archive in blocks after some delay to improve efficiency.

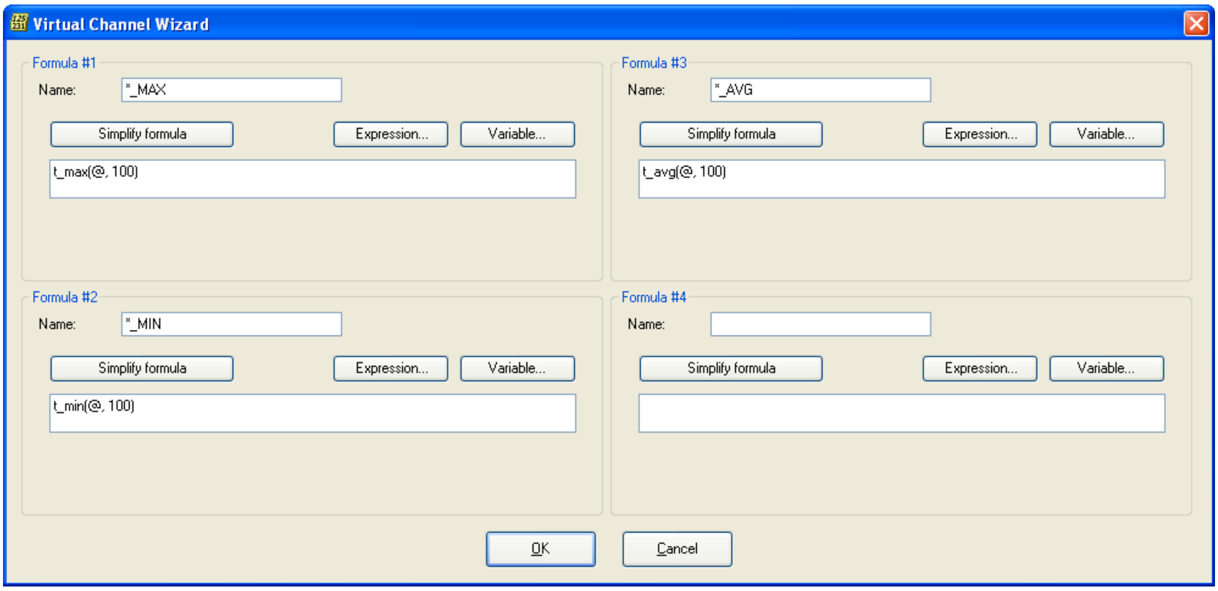
The following functions can be used to retrieve information from a GPS receiver. A compatible GPS receiver should have the following characteristics:

- Accessible through a (virtual) COM-port in Windows.
- Uses NMEA standard communication (baud rate: 4800).
- Continually sends unsollicited NMEA-messages. In particular, the following message types are parsed by Autosoft 3000: $GPRMC (recommended minimum information, required), $GPGSV (satellites in view, optional), $GPGSA (positional accuracy, optional) and $GPGGA (height information, optional).

Information is retrieved using 'GPS(port, parameter)', where 'port' is the COM-port number and 'parameter' signifies the specific type of information to retrieve. The following parameters are available:

GPS(port,0)	Current state of the COM-port, 1 = OK, 0 = inaccessible
GPS(port,1)	Number of satellites used to determine the position (0 = no fix)
GPS(port,2)	Latitude, as a floating point value in degrees. Positive values correspond to the Northern hemisphere, negative values to the Southern hemisphere.
GPS(port,3)	Longitude, as a floating point value in degrees. Positive values correspond to the Eastern hemisphere, negative values to the Western hemisphere.
GPS(port,4)	Current speed in m/s, if the unit is moving.
GPS(port,5)	Current true bearing in degrees, if the unit is moving.
GPS(port,6)	Time difference in msec between UTC time received from satellites and current PC time, positive numbers indicate the PC is ahead in time.
GPS(port,7)	UTC Time in milliseconds since midnight, as received from satellites.
GPS(port,8)	Days since 1/1/1980, as received from satellites.
GPS(port,9)	Day of month (1..31), as received from satellites.
GPS(port,10)	Month (1..12), as received from satellites.
GPS(port,11)	Year (2000..2099), as received from satellites.
GPS(port,12)	Position dilution of precision, indicates accuracy of position, lower values mean higher precision. Rough guide: <3 = good, <10 = ok, >10 = bad, 100 = no fix.
GPS(port,13)	Horizontal dilution of precision, indicates accuracy of position, lower values mean higher precision. Rough guide: <3 = good, <10 = ok, >10 = bad, 100 = no fix.
GPS(port,14)	Vertical dilution of precision, indicates accuracy of position, lower values mean higher precision. Rough guide: <3 = good, <10 = ok, >10 = bad, 100 = no fix.
GPS(port,15)	Number of satellites visible to the unit. The unit does not necessarily have a fix on all visible satellites.
GPS(port,16)	Height in meters above/below mean sea level.
GPS(port,17)	Geoidal separation (difference between WGS-84 earth ellipsoid and mean sea level), in meters.
GPSDIST(lat1, long1, lat2, long2)	Calculate the distance between 2 points (latitude, longitude) in meters.

Virtual Point Wizard



The virtual point wizard helps you to create a series of virtual points that all perform the same type of function(s) on a list of inputs. When you open the Virtual Point Wizard from the "Insert" menu, you can first select the list of points that the virtual function should operate on.

Then you can define up to four different functions that operate on all of the selected inputs. The example creates three different virtual points for each input: one for a maximum, one for a minimum, and one for an average over every 100 values of the input.

In the formula name, the '*' is replaced by the name of the input variable. So with an input called 'VAR1', the virtual point for the first formula in this example would be called 'VAR1_MAX'. In the formula text itself, the '@' character is replaced by the input name, so the actual formula would be 't_max(VAR1, 100)'.

To create only one or two functions per input, simply remove the formula from the other fields.

Filter

The virtual function filter has the following parameters:

filter(input, class, type, window, order, sample rate, cutoff frequency, bandwidth, ripple, transition)

where:
input = incoming signal to filter
class = filter class, 1 = FIR, 2 = Butterworth, 3 = Chebyshev, 4 = Chebyshev type II (inverse), 5 = Elliptic
type = filter type, 1 = low-pass, 2 = high-pass, 3 = band-pass, 4 = band-stop
window = FIR window type, 0 = rectangular, 1 = Hamming, 2 = Hann, 3 = Blackman. Unused for IIR-type filters (Butterworth, Chebyshev, etc.)
order = digital order (number of coefficients) for FIR, or number of poles for IIR-type filters (Butterworth, Chebyshev, etc.)
sample rate = sample rate (in Hz) of the incoming signal, used as a reference for the cut-off frequency
cut-off frequency = cut-off point (in Hz) for the filter, or mid-point for band-pass/band-stop
bandwidth = band (in Hz) for band-pass/band-stop only.
ripple = Pass-band ripple for Chebyshev and Elliptic filters, or stop-band attenuation for Chebyshev type II, in dB. Unused for other filter classes.
transition = Transition band for Elliptic filters (in Hz). Unused for other filter classes.

The best choice of filter and parameters depends on the application.

Filter Selection

There are many different types of digital filters; choosing the right one for your application requires some knowledge of the available options and the signal you want to filter.

FIR vs. IIR

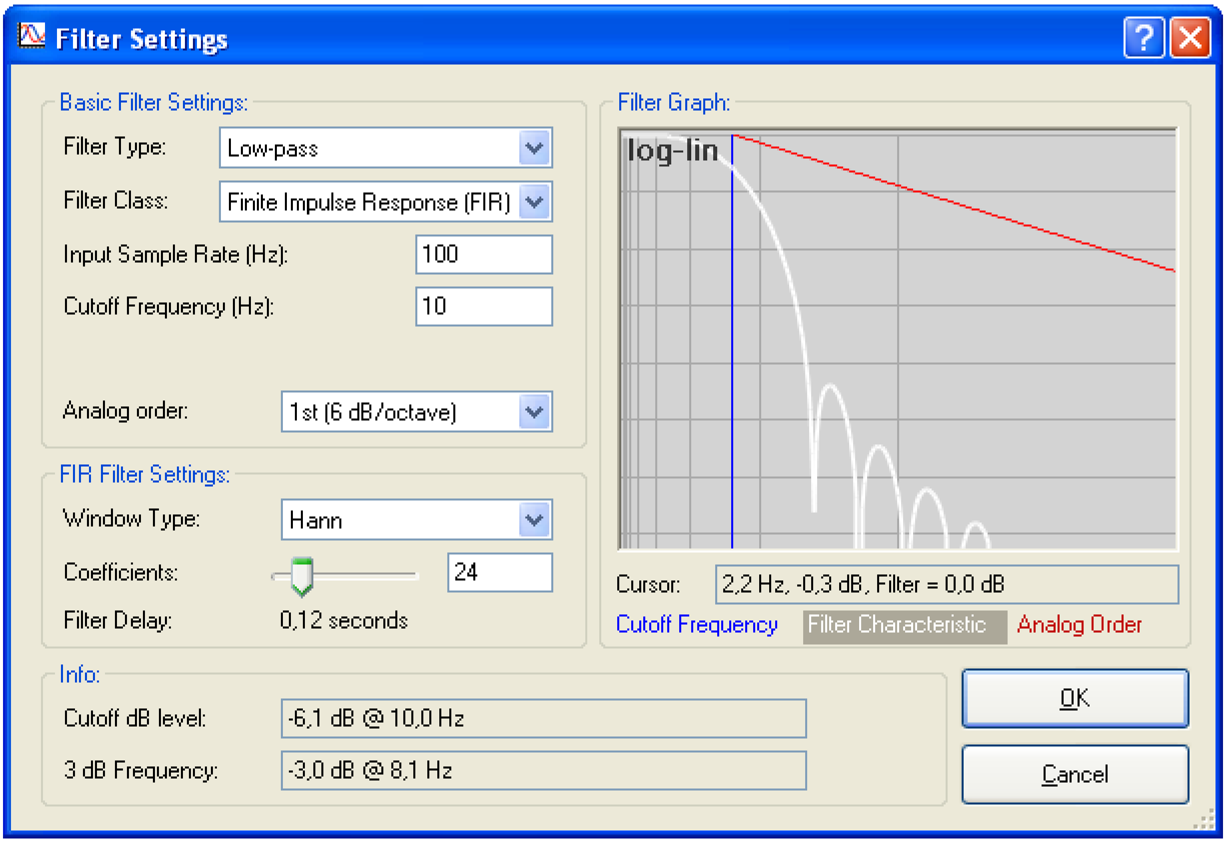
The most important choice to make is between FIR (Finite Impulse Response) and IIR (Infinite Impulse Response) filters. IIR filters include digitale equivalents of well-known analog filters, like the Butterworth filter. In digital filtering however, FIR filters have several characteristics that make them preferable to IIR filters: they have a predictable time delay and are well-behaved (numerically stable). This is because FIR filters use a fixed (finite) number of unfiltered values to calculate each filtered output value. IIR filters use not only unfiltered values, but also previously calculated filtered values to determine each output value. The consequence is that filtered value are indirectly calculated from a potentially infinite number of previous input values, which can have negative influence on the numerical stability of the filter.
Conclusion: because FIR filters are well-behaved, they should be your first choice. But there are two reasons why IIR may be the better choice for your application: delay and complexity. Depending on the desired filter characteristic, a FIR filter may need a lot of input values for each filtered value, which could mean a lot of calculations for each sample and a long time delay on the filtered signal. IIR filters can offer shorter time delays and less calculations in these cases. The price you pay is that the time delay is no longer fixed and that numerical stability is no longer guaranteed.

Available Filters

The following filter classes are supported:
- FIR (Finite Impulse Response)
- Butterworth (IIR, comparable to analog equivalent)
- Chebyshev type I (IIR, with configurable ripple in the pass-band)
- Chebyshev type II (IIR, with configurable stop-band attenuation)
- Elliptic (IIR, generalized form of the other IIR filter types)

FIR Filter

If you are unsure about the type of filter to use, please read the section on filter selection first.



Basic filter settings

Using the filter configuration dialog you can visualize the effects of a selected filter and its associated parameters. The basic filter settings at the top left are available for every type of filter, as is the graph on the right. The graph shows the level of suppression (in dB) for a range of frequencies (the filter characteristic line in white), you can use the mouse cursor to hover over frequencies and show the corresponding dB value. The analog order (shown in red) can be used as a reference for the quality of the filter, but has no influence on the filter itself. You can click on the graph to switch between a log-log and a log-lin representation of the frequency spectrum. The info window at the bottom left shows the suppression at the cutoff frequency as well as the frequency at which the filter reaches 3 dB suppression.

FIR filter settings

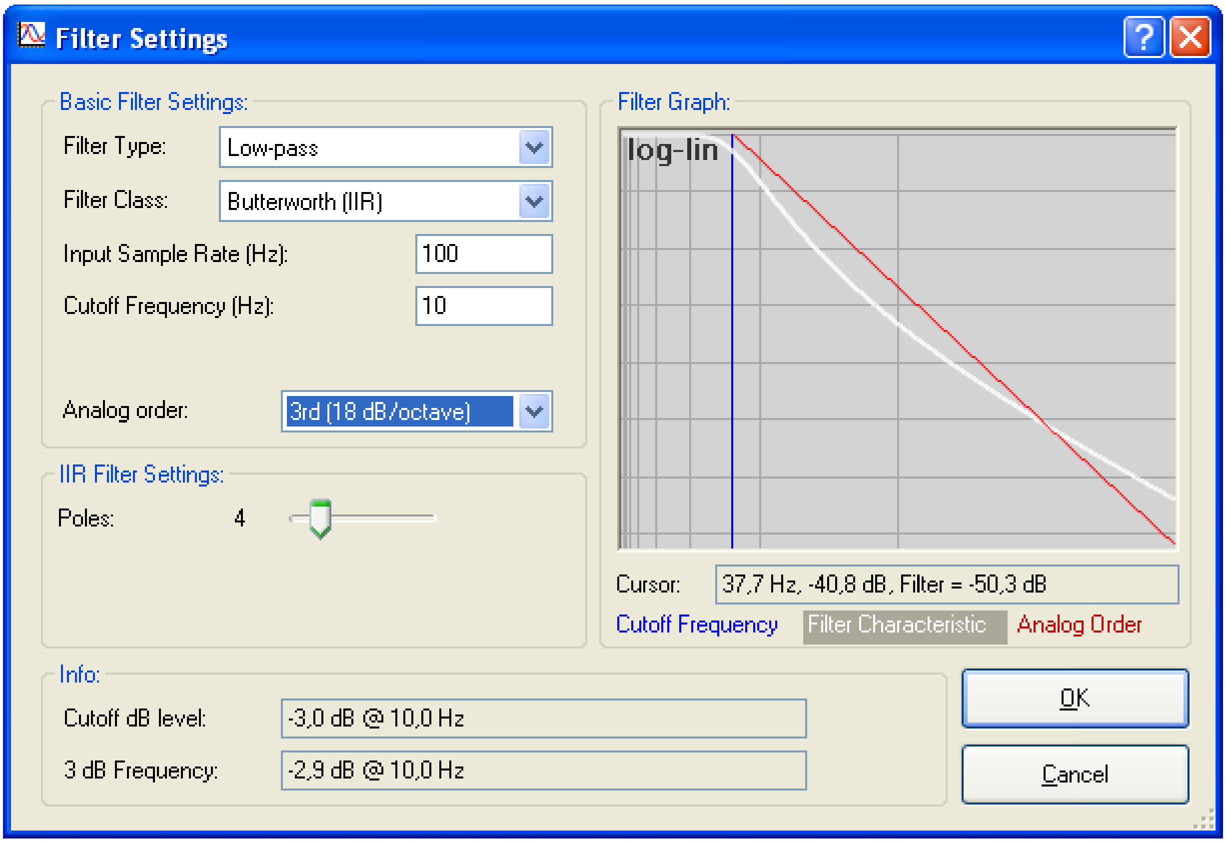
The FIR (Finite Impulse Response) filter class is controlled by two important parameters: the window type and the number of coefficients. The window type influences the characteristic of the filter; the following choices are available:

- **Rectangular**: the most basic window, available for reference or testing, but not a good choice for most practical applications.
- **Hamming**: has a small transition band (short distance between the pass-band and the stop-band), but shows relatively big ripples in the stop-band.
- **Blackman**: has a large transition band, but extremely good suppression in the stop-band
- **Hann**: middle-ground between Hamming and Blackman window types, the default choice.

The number of coefficients determines how well the filter performs: more coefficients result in a steeper filter. The cost for this is two-fold: a higher time-delay between input and filtered output (shown in the dialog) and more computations (the number of coefficients corresponds to the number of multiplications and additions that need to be performed to calculate each individual filtered value).

For FIR filters, when you select a different analog order (the red reference line in the graph), the number of coefficients is automatically adapted to approach this line. This is just an estimation and can be changed manually to any other suitable value.

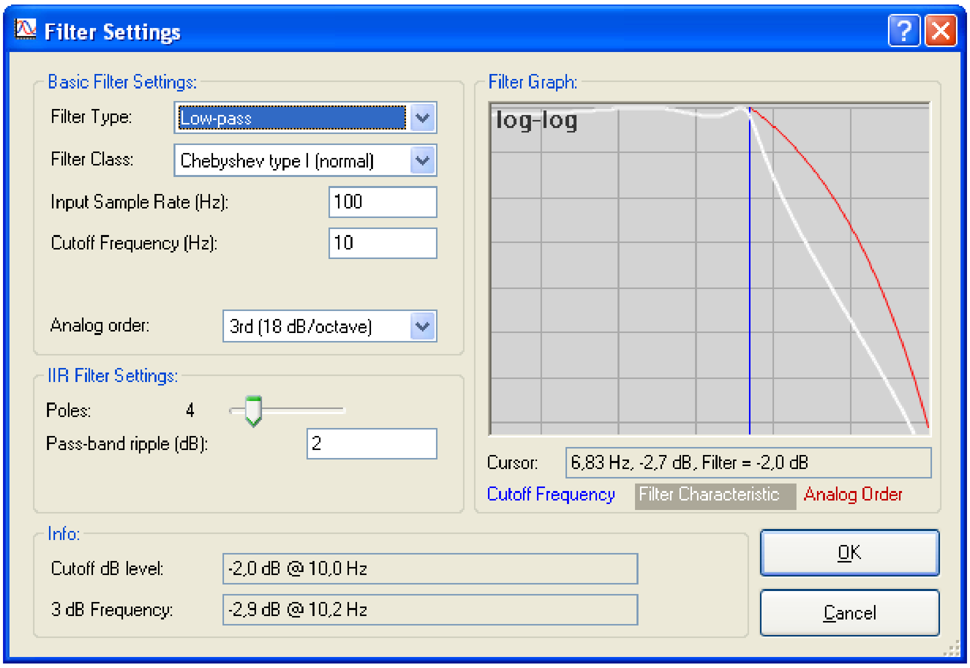
Butterworth Filter



The Butterworth filter is the most well-known filter in the class of IIR (Infinite Impulse Reponse) filters, and the easiest to use. Its characteristics mimic the analog equivalent; most important features: no ripples in pass-band or stop-band, but a relatively large transition band. The steepness of the filter can be influenced by the number of poles, more poles mean a steeper filter. The negative side: more poles also mean a bigger chance of numerical instability and longer delay times between filtered and unfiltered signal. As a rule, you should try to keep the number poles as low as possible, preferably 4 or less.

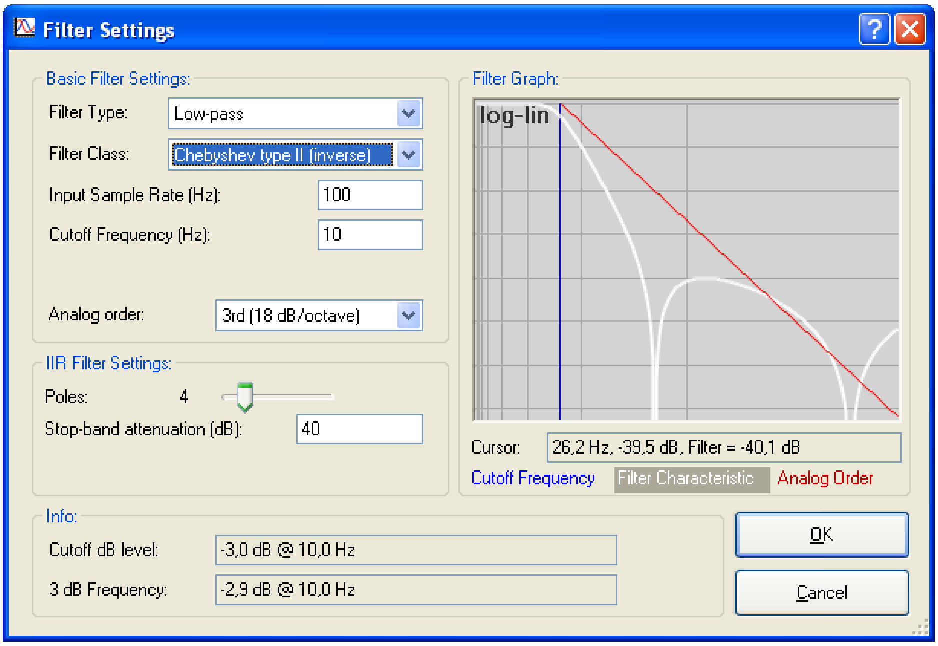
Chebyshev Filters

Chebyshev Type I Filter



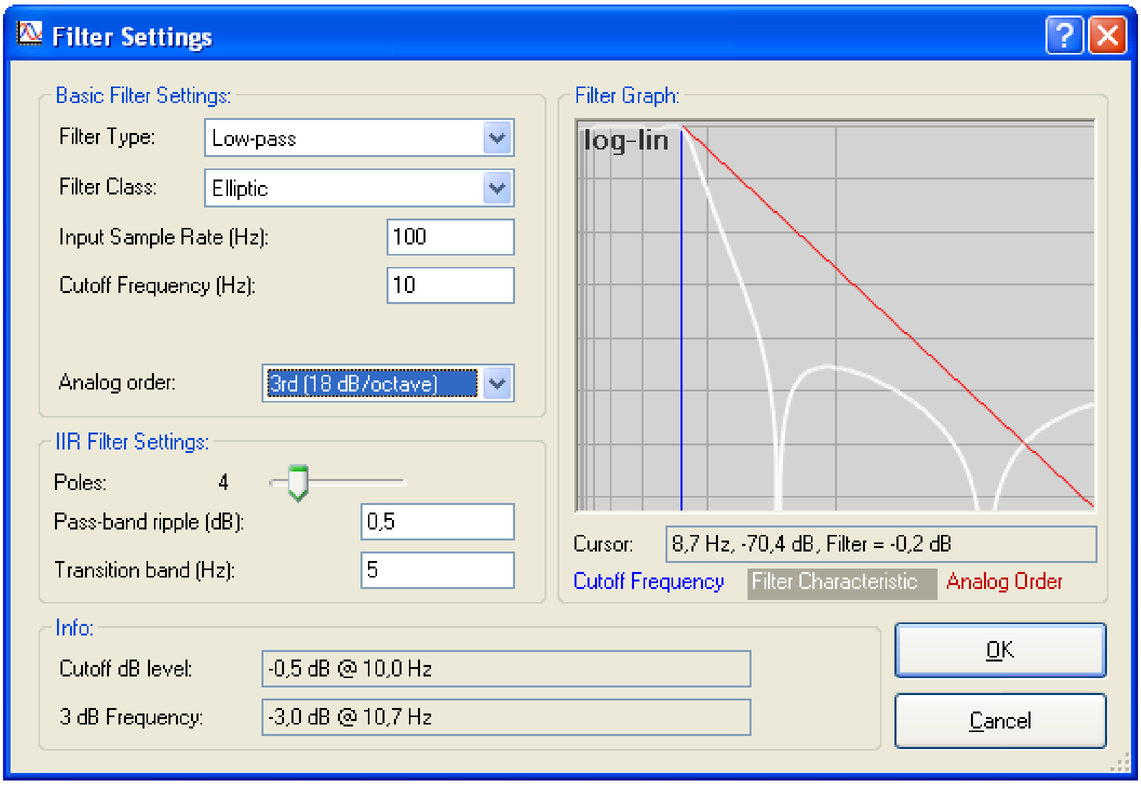
The Chebyshev filter belongs to the class of IIR (Infinite Impulse Reponse) filters, has a relatively short transition band and good suppression (without ripples) in the stop-band. It has a ripple the pass-band; the magnitude of this can be configured. A default value often used for this pass-band ripple is 0.5 dB. The lower the pass-band ripple, the larger the transition band will become, at 0 dB the characteristic is similar to the Butterworth filter.
The steepness of the filter can further be influenced by the number of poles, more poles mean a steeper filter. The negative side: more poles also mean a bigger chance of numerical instability and longer delay times between filtered and unfiltered signal. As a rule, you should try to keep the number poles as low as possible, preferably 4 or less.

Chebyshev Type II Filter



The Chebyshev type II (or Inverse Chebyshev) filter belongs to the class of IIR (Infinite Impulse Reponse) filters, has a relatively short transition band, no ripples in the pass-band and a configurable suppression (with ripples) in the stop-band.
The steepness of the filter can further be influenced by the number of poles, more poles mean a steeper filter. The negative side: more poles also mean a bigger chance of numerical instability and longer delay times between filtered and unfiltered signal. As a rule, you should try to keep the number poles as low as possible, preferably 4 or less.

Elliptic Filter

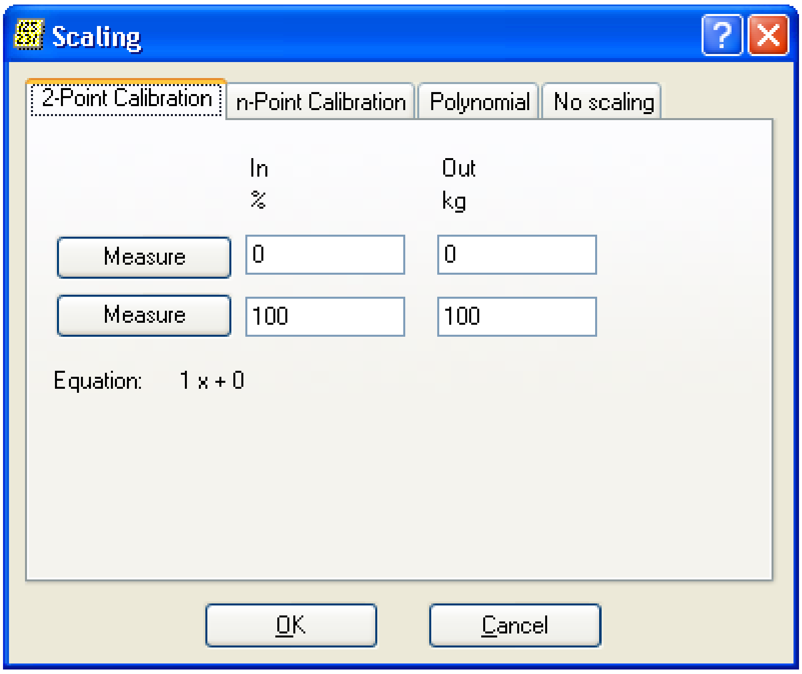


Elliptic Filter

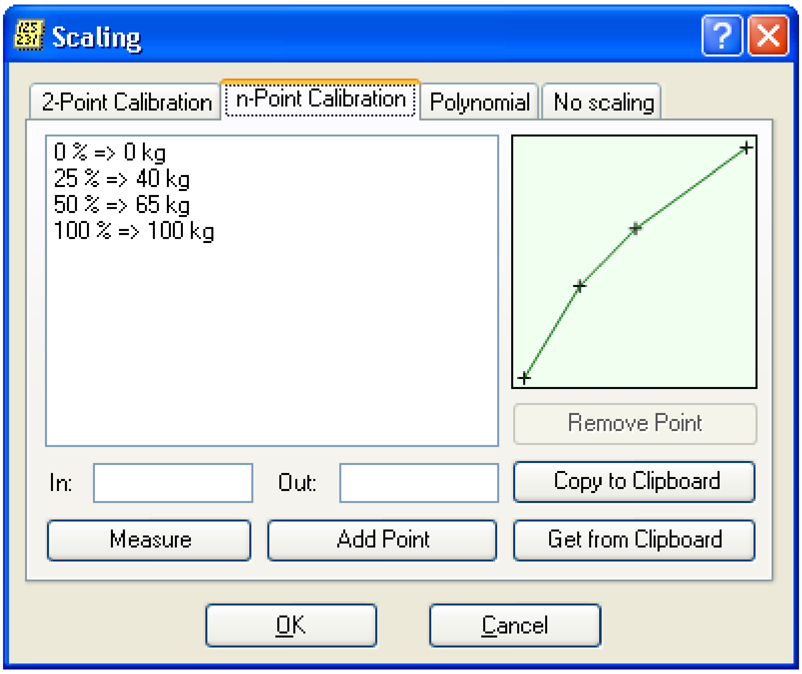
The Elliptic filter belongs to the class of IIR (Infinite Impulse Reponse) filters, and is a generalized form of the Butterworth and Chebyshev type filters. It has a configurable ripple in both pass-band and stop-band, and a very short transition band, depending on the amount of ripple. You can configure the pass-band ripple in dB and the transition band in Hz (estimation), the stop-band ripple will follow from these values.
The steepness of the filter can further be influenced by the number of poles, more poles mean a steeper filter. The negative side: more poles also mean a bigger chance of numerical instability and longer delay times between filtered and unfiltered signal. As a rule, you should try to keep the number poles as low as possible, preferably 4 or less.

Scaling Configuration

To configure a polynomial or n-point scaling, first set the appropriate output units in the measurement point setup.

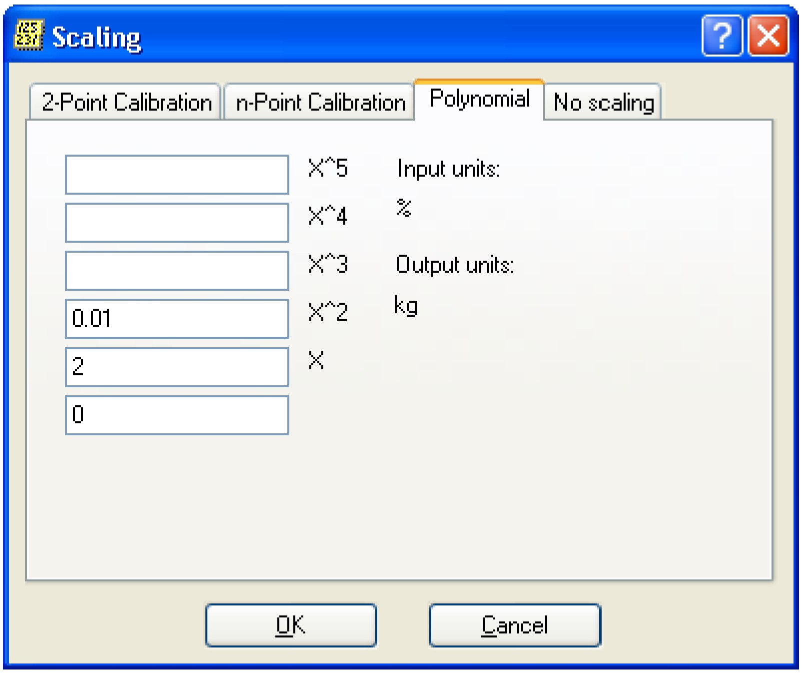


For 2-point calibration, you can set input and output values for 2 points and the software will apply linear scaling to the input to match the specified output values, shown by the equation below the settings. Use the 'measure' buttons to obtain a current input value for the measurement point. The 2-point calibration setting is a special case of the n-point calibration:



In the n-point calibration window, you can use an arbitrary number of interpolation points to scale your input value. Between each point, linear interpolation is used. To import or export a conversion table,

you can use the '**Copy to Clipboard**' and '**Get from Clipboard**' buttons. Imported tables should be in text format, two columns separated by a <tab> character. The first column defines input values, the second column the associated output values.
The dialog will always sort the interpolation points by ascending input value. If you add a new interpolation point with an input value that is already used, the old point will be replaced.

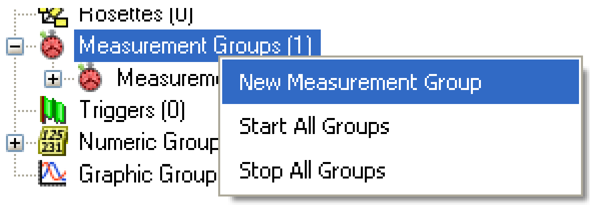


Instead of linear interpolation, you can also define a polynomial to map input to output values. Polynomials up to 5th order are allowed, leaving the higher order fields blank results in lower order polynomial calculations.

Measurement Groups

Measurement groups are used to perform the actual measurements. A measurement group can contain any number of measurement points, which can be collectively be measured, using the same measurement speed for all points.

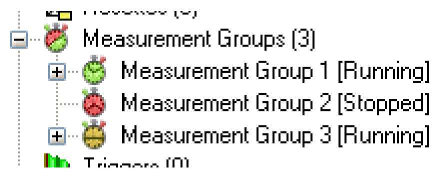
To create a new measurement group, right-click the 'Measurement Groups' item in the overview tree and select '**New Measurement Group**' from the menu.



Alternatively, drag and drop any selection of measurement points or devices/cards containing measurement points to the 'Measurement Groups' item to create a new group to which the points will immediately be added.

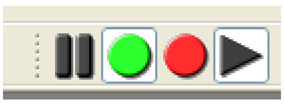
To perform a series of measurements, a measurement group must be started after creating it. There are several visual cues showing which measurement groups are currently running and producing measurement values.

First, the color of the icons shown in the overview tree:



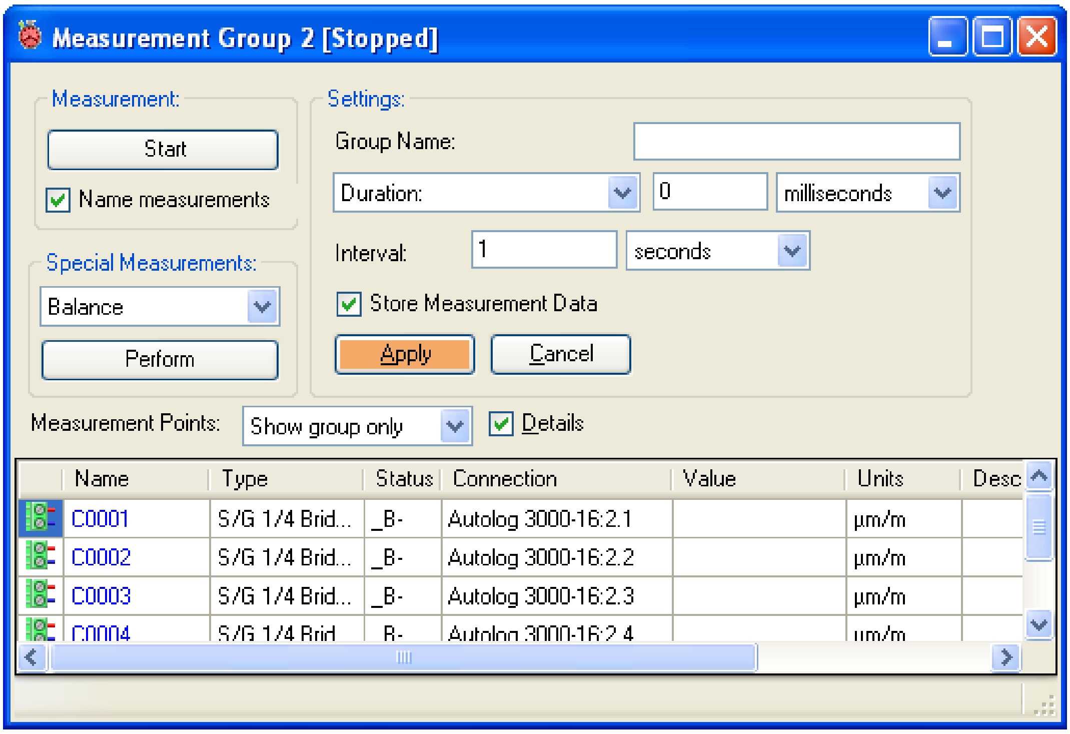
In this example Measurement Group 1 is running (green icon), Measurement Group 2 is stopped (red icon). Although Measurement Group 3 also shows the text '[Running]', it is configured not to store any measurement values to disk, and therefore shows a yellow icon.

Similar information can be found on the top-right of the Autosoft 3000 window, were a toolbar allows quick access to start/stop measurement groups:



From left to right this example shows the **'pause'** button(2 vertical bars), a green circle indicating group 1 is running and storing data, a red circle indicating group 2 is stopped and a black triangle indicating that although group 3 runs, it does not store data.
Move and hold the mouse over these buttons to see which group is which.
The **'pause'** button can be pressed to temporarily suppress all data storage. When the configuration is **'paused'**, all measurements are performed normally, but no measurement data gets written to the archives. This can be used to test the configuration. As long as the configuration is paused, the pause button will blink black/red.

Measurement Group Settings



To show the settings window for the measurement group, double-click the group item in the overview tree on the left. Closing the window (X) will not remove the group, but only hide its settings.

To add measurement points to the group, you can drag and drop any item containing measurement points to the white area at the bottom of the window. Alternatively, you can click the **'Show group only'** box and change it the read **'Show all points'**. Then, you can use the check marks that will appear in from of each measurement point to determine which points should be included in this group.

The title bar shows the current state of the group, both in text and in the form of a colored icon.

The button top-left is using to start/stop the measurement, settings can be made on the right side. Be aware that changes to the settings are not applied immediately, but only after clicking the **'apply'** button, which will flash orange to make you aware of this.

The **'special measurement'** section allows you to perform special commands like balance, tare, reset balance, reset tare and shunt measurements. Each special measurement is performed only once, after clicking the **'Perform'** button. Balance and tare measurements are only performed on measurement points that have the **'auto balance'** or **'auto tare'** checked, shunt measurements are only performed on measurement points that support this type of measurement.
When **'detail'** information on the measurement points is checked, the **'status'** column shows which points have 'auto balance' (B) and 'auto tare' (T) active.

You can use the **'group name'** field to choose your own name for the measurement group.

Below the group name, you can choose the type of measurement to perform.

Type	Meaning
Unlimited	The measurement will not stop automatically, and can only be stopped by the user.
Number of Measurements:	The measurement runs until the specified number of measurements has been made.
Duration:	The measurement runs for a specified length of time.
Loadstep	A loadstep measurement is a single measurement. A loadstep measurement is comparable to a 'number of measurements' measurement with the number set to 1.

The **measurement interval** can be specified if the measurement is not a loadstep. You can enter any interval with a minimum of 0.001 seconds (1 millisecond). If the hardware cannot keep up with the specified measurement interval, the actual scanning speed will be as fast as possible.
For Autolog 3000, be aware that there is a limited number of measurement intervals supported. If you choose an unsupported interval, the Autolog 3000 will actually measure at the next higher supported speed. The following intervals are supported by Autolog 3000:

0.2 milliseconds (5 kHz), only for CA3520/CA2CF measurement card
0.5 milliseconds (2 kHz), only for CA3520/CA2CF measurement card
1 millisecond (1 kHz)
2 milliseconds (500 Hz)
4 milliseconds (250 Hz)
10 milliseconds (100 Hz)
20 milliseconds (50 Hz)
40 milliseconds (25 Hz)
100 milliseconds (10 Hz)
200 milliseconds (5 Hz)
1000 milliseconds (1 Hz)
any multiple of 1000 milliseconds (integer number of seconds)

The **'store measurement data'** check box determines whether or not measurement data produced by this measurement group gets stored in the archive. **If you uncheck this box, all measured data produced by this group will only be visible online and can not be retrieved or used outside of Autosoft 3000 afterwards!**

To start/stop measurement at a specific time or after an event, use triggers.

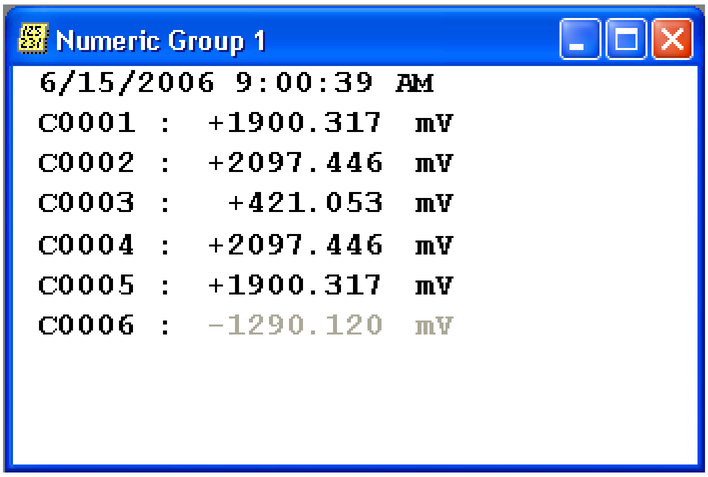
Numeric Groups

Numeric groups show measurement data in numerical form. Any selection of measurement points can be dragged to the numeric group window, where the most recent measurement value(s) can be shown and printed.

To create a new numeric group, right-click the 'Numeric Groups' item in the overview tree and select '**New Numeric Group**' from the menu.

Alternatively, drag and drop any selection of measurement points or devices/cards containing measurement points to the 'Numeric Groups' item to create a new group to which the points will immediately be added.

To show the display window for the numeric group, double-click the group item in the overview tree on the left. Closing the window (X) will not remove the group, but only hide its display.

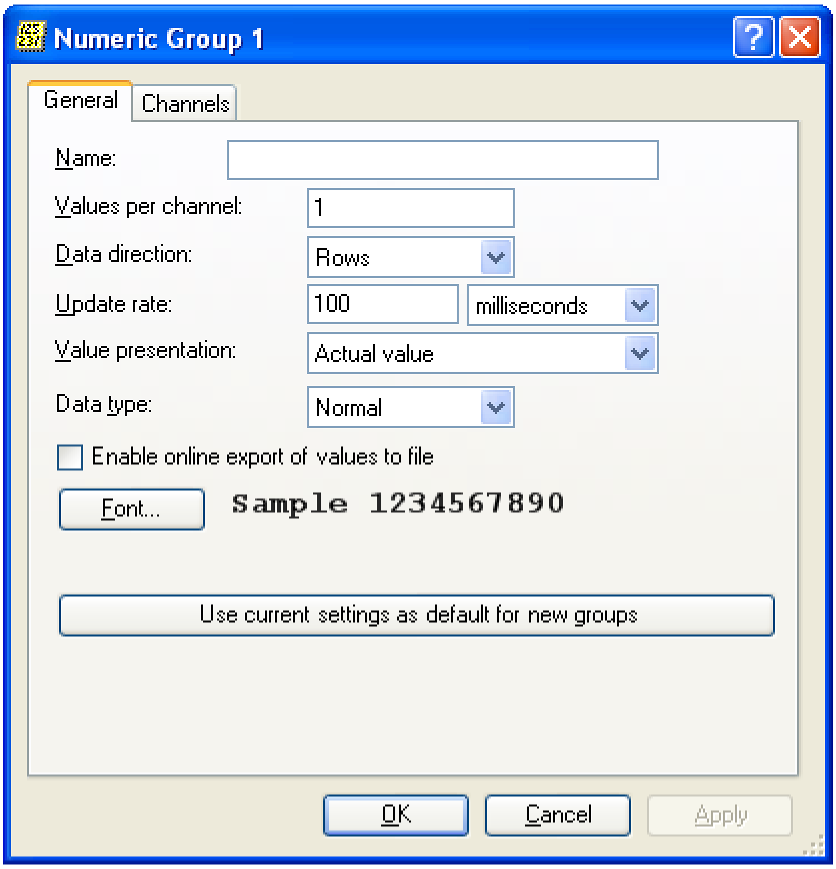


The display shows the time at which the newest measurement value was received. Values that are displayed in gray (C0006 in the example above) indicate old data, i.e. measurement values that were not updated recently.

To rearrange the order in which measurement points are displayed, you can drag them to a new place. To make selections of measurement points: click to select a single one, shift-click to select a range or ctrl-click to add points to your selection.

Numeric Group General Settings

To access the settings of a numeric group, right-click anywhere in its display window and select **'Properties'** from the context menu.



You can set a **name** for the numeric group, which is used to identify it throughout the program.

The **values per channel** setting determines how many of the most recent values for each measurement point are shown. If multiple values are to be shown, the **data direction** determines whether they are shown in rows or in columns.

The **update rate** determines the maximum number of values sent to the numeric group for high speed measurements. Because the numerical presentation only shows one or a few values per channel, it is usually not necessary to set this update rate higher than the default value.

The **value presentation** allows you to show the measurement value as a percentage of the full scale of the measurement point. The meaning of 'full scale' is determined by the minimum and maximum value for each measurement point, as defined in the measurement point setup. The minimum value corresponds with 0%, the maximum value with 100% of full scale.

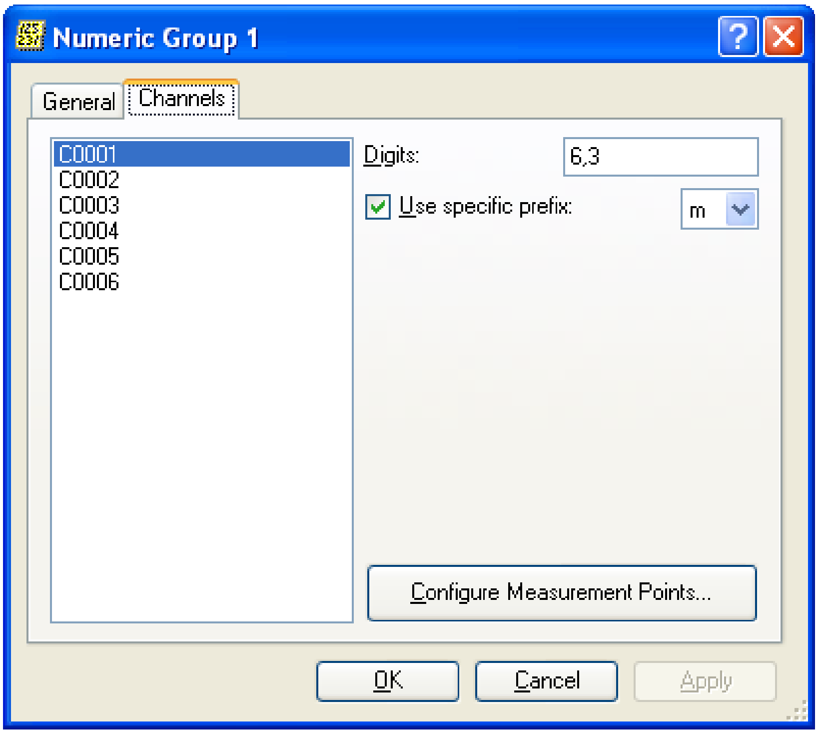
To show the results of special measurements, like balance, tare and shunt, change the **'data type'** setting to the appropriate choice.

To store all values received by the numeric group to file, check the **'Enable online export of values to file'** setting. This will enable the File Output tab.

You can set the font used for numeric presentation by pressing the **'font'** button.

Numeric Group Channel Settings

To access the channel settings of a numeric group, right-click on any channel in its display window and select **'Properties'** from the context menu.



Using the channel settings you can change the presentation format of individual channels for this specific numeric group. The default presentation settings are copied from the measurement point setup.

Using the **digits** fields, you can specify the number of digits used for the presentation of the selected points. For example '6.3' sets a maximum of 6 pre-comma and exactly 3 post-comma digits.

You can also specify which **metric prefix** should be used to display the value, or allow the software to select the best prefix based on the current measurement value.

Use the **Configure Measurement Points...** to access the measurement point setup for the selected channel(s).

Numeric Group File Output

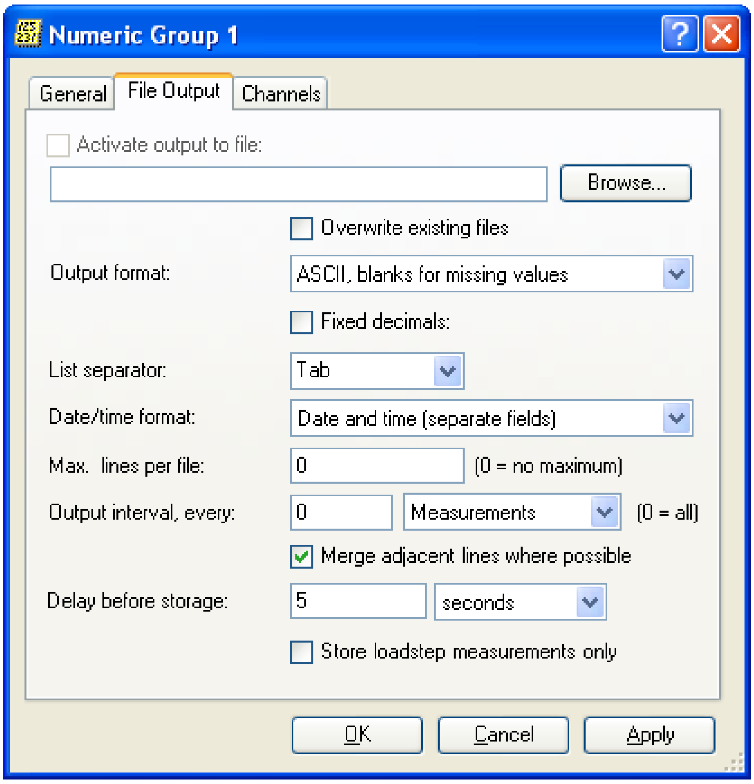
First of all a general tip regarding this function:
The direct storage of measurement data in an ASCII file should not be the default way of working. When needed, it should only be activated in parallel to storage in the Autosoft 3000 archive (see Measurement Group Settings). Please beware of the following when storing directly to ASCII:

1. The files can become very large very quickly, when the measurement speed is high.
2. The data can not be restored to the graphic groups of Autosoft 3000 for later analysis. This is only possible for measurement data stored in the Autosoft 3000 archive.

To access the settings of a numeric group, right-click anywhere in its display window and select **'Properties'** from the context menu. The 'File Output' tab becomes visible after you check the **'Enable online export of values to file'** setting on the General tab.

After activating the online export function, the icon of the numeric group changes to a floppy disk symbol, showing whether or not storage is active (red/green).

To activate output to file, first select the file name to use, or use the **'browse'** button to find a suitable location. Then select the type of file(s) you wish to generate:



If the **'overwrite existing files'** is not checked, a sequence number will automatically be appended to the file name and increased to make sure no existing file gets overwritten.

The **output format** is either ASCII, DIAdem or a list-based format. ASCII files are tab-seperated by default, making it easy to read them in e.g. Excel. DIAdem output consists of 2 files: 1 ASCII file (extension .DAT) describing the format and 1 binary file (extension .R32) containing the measurement values. List files contain a single line for each measurement value, and are not suitable for high volumes of measurement data.

You can the '**fixed decimals**' check box to specify the exact number of post-comma digits to show in ASCII-based export formats. Leave this unchecked to get the maximum amount of information.

If you set a **'max. lines per file'** the export will create multiple files with increasing sequence numbers as needed to make sure each file does not contain more than the specified amount of lines.

The **'output interval'** can be used to reduce the amount of data retrieved by e.g. only exporting 1 value out of every 10, or exporting 1 value per minute.

If you check the **'merge adjacent lines where possible'** setting, the export routine will try to merge as many values as possible onto a single line, even if they do not have the same time stamp. This helps to combine data from different sources (devices) that do not supply data for the same measurement at the exact same time.

The **delay before storage** setting ensures that data is buffered before it is written to disk, which is needed to synchronize data from different devices. For measurements that can take a long time to perform (like IDVM scans on an Autolog 20xx/Unilog 2500 device), a longer delay time may sometimes be necessary to synchronize with other data, for example from virtual measurement points.

The **store loadstep measurements only** setting will only store information associated to loadstep measurements. Using this setting and activating the file output will also limit the values shown in the numeric group to the ones produced by loadsteps only.

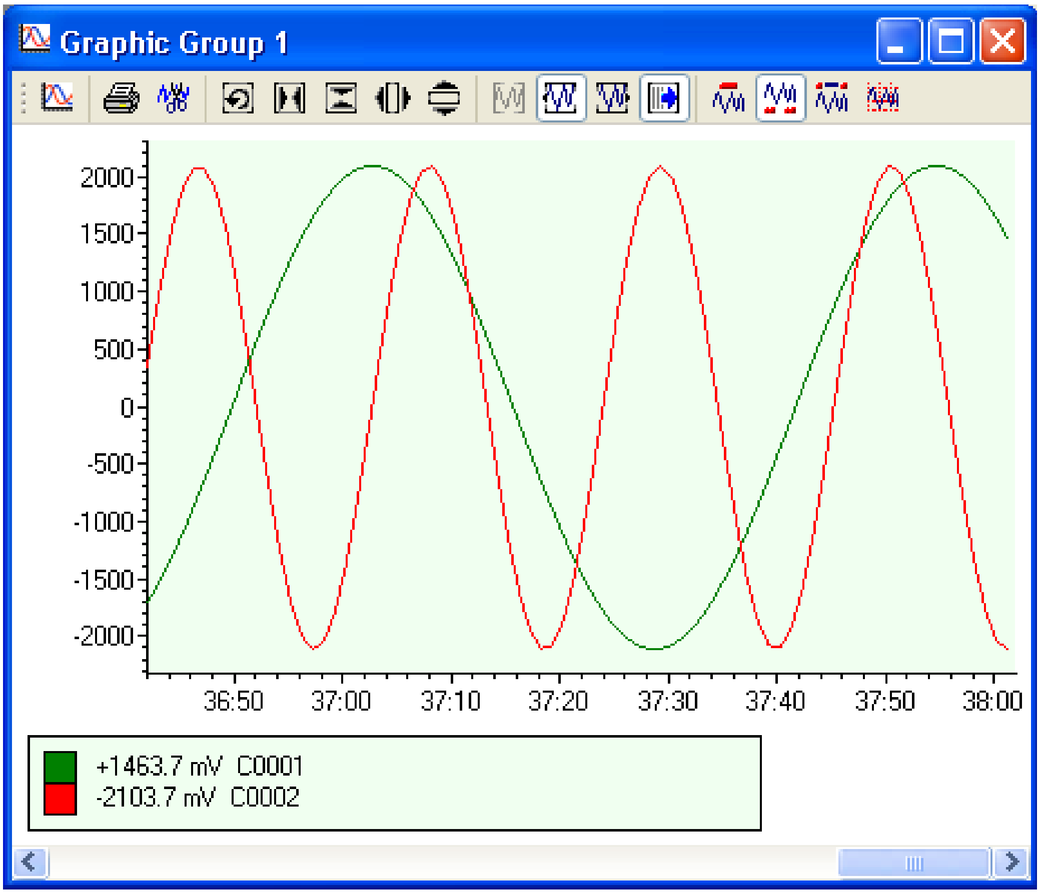
Graphic Groups

Graphic groups show measurement data in graphical form. Any selection of measurement points can be dragged to the graphical group window, where the measurement values are shown as a function of time, or as an X-Y plot using any measurement point as X-axis.

To create a new graphic group, right-click the 'Graphic Groups' item in the overview tree and select '**New Graphic Group**' from the menu.

Alternatively, drag and drop any selection of measurement points or devices/cards containing measurement points to the 'Graphic Groups' item to create a new group to which the points will immediately be added.

To show the display window for the graphic group, double-click the group item in the overview tree on the left. Closing the window (X) will not remove the group, but only hide its display.



By default, the graphic group shows measurement points in a Y-t plot (measurement values against time). The legend at the bottom shows which axis the points are assigned to (both points are assigned to the left Y-axis in this example).

To rearrange the assignment of the measurement points, you can click to select them in the legend and drag them to a new location (right side of graph to assign to right axis, x-axis to assign to x-axis).

The contents of the graphic display can be manipulated using the mouse in a number of ways.

To zoom in and out or move the graph contents, the following options are available:

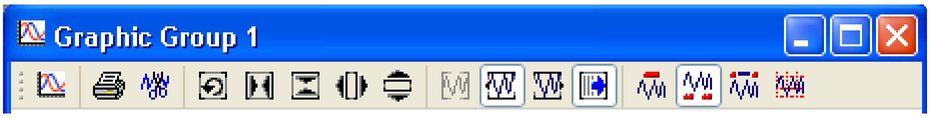
- Click and drag in the graph itself to select a region to zoom in to.
- Hover the mouse over any location in the graph and use the scroll wheel on the mouse to zoom in and out.
- Hover the mouse over any axis and use the scroll wheel on the mouse to zoom in and out for a single axis.
- Click and drag any axis to move the graph in that direction.
- Click the middle mouse button (or scroll wheel) in the graph to show a 'move'-cursor to drag the graph around.

To show (cursor)-information (best used when the graphic is not scrolling automatically):

- Hover the mouse over any location in the graph and wait for an information window to appear, which shows the current location.
- Click once in the graph and move the mouse to a new location to get info on a region. Wait for the information window to appear, it now shows the difference between current and click-location.

Click on any measurement point in the legend to select it, then move the mouse over the graph. The cursor will now follow the measurement data for the selected point. Click on any unused area in the legend to deselect the measurement point and return to normal cursor movement.

Graphic Group Toolbar



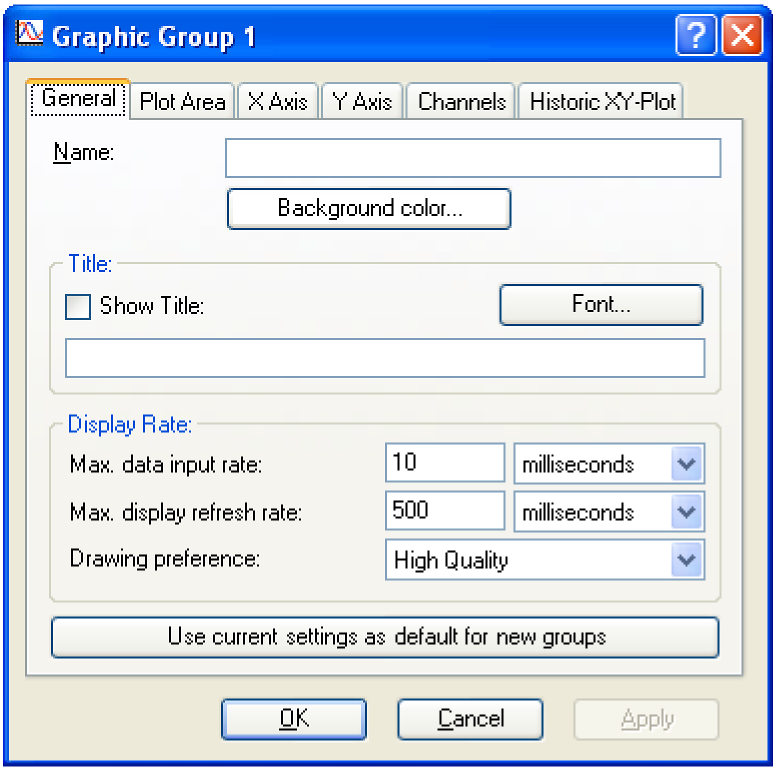
The toolbar gives you quick access to common tasks, from left to right the buttons are:

Properties	Shows the settings for this graphic group
Print	Prints the current window contents
Clear Graph	Removes all current data points from the graph
Zoom Back	Return to previous zoom state
Zoom In/Out	Zoom in and out in x and y-direction
Autoscale X	Automatic scaling of X-axis (only for X-Y plots)
Autoscale Y	Automatic scaling of left and right Y-axis
Moving X-axis	Automatic scrolling of graph (Y-t plot) to follow current time
Show Title	Show/hide title of graph
Show Legend	Show/hide the legend
Show Date/Time	Show/hide the date/time information at the top left and right
Show Grid	Show/hide grid

More settings can be accessed by right-clicking in the graphic display itself and choosing the appropriate item from the context menu.

Graphic Group General Settings

To access the general settings of a graphic group, right-click anywhere in its display window and select **'Properties'** from the context menu.



You can set a **name** for the numeric group, which is used to identify it throughout the program.

The **'background color'** is the color used for all areas of the display except the plot and legend.

The **title** can displayed at the top of the graphic display so it will show up in printouts and copies of the graph (unlike the **name** of the graphic group).

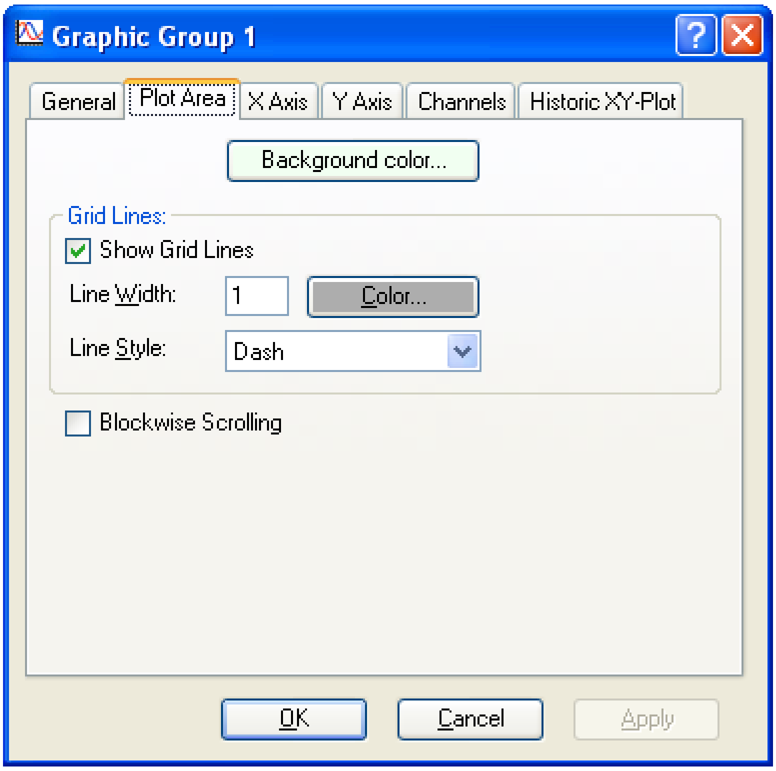
The **display rate** settings influence the way on-line measurement data is processed and displayed. The **'max. data input rate'** limits the amount of measurement data that is displayed on-line for high-speed measurements. Because the graphic group can only show a limited number of data points (5000 for each measurement point), accepting all data from a high-speed measurement causes older data to be flushed out of the display very quickly.
When scrolling back in history, all necessary data will be retrieved from the archives, not limited by the 'max.data input rate' setting.

The **'max. display refresh rate'** determines how often the graph tries to update itself when moving x-axis is active. High speeds (e.g. 20 milliseconds) will show smooth scrolling, but can cause a considerable CPU-load.

Using the **'drawing preference'** you can influence the quality of the line drawings, again at the cost of a higher CPU-load.

Graphic Group Plot Settings

To access the plot area settings of a graphic group, right-click on the plot area in its display window and select **'Properties'** from the context menu.



Here, you can select the **background color** for the plot area.

Also, you can choose whether or not to **show grid lines** and determine the **line width, style and color** to use for the grid.

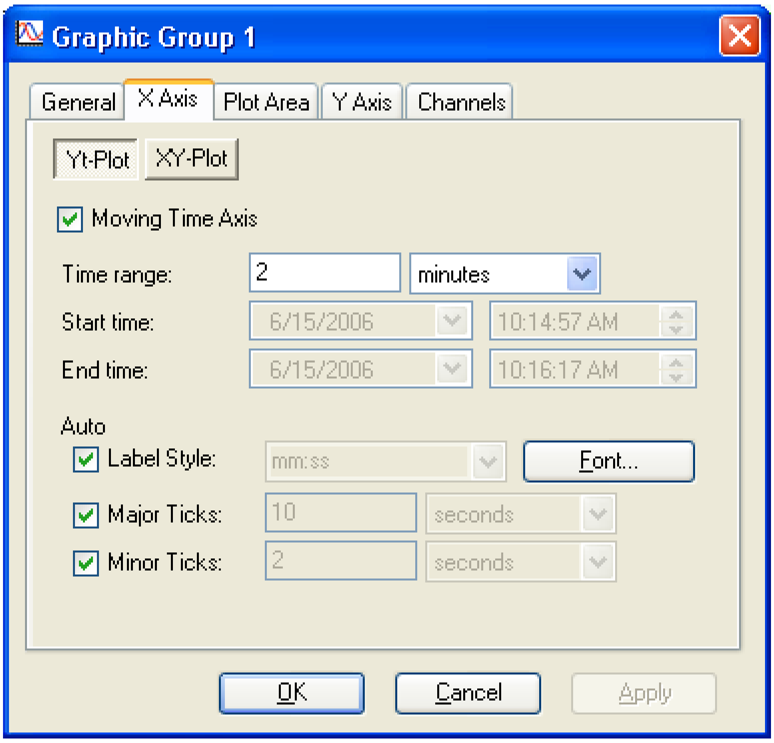
Blockwise scrolling will make the graph move along the time axis in steps of about half the width of the full graph, instead of the continuous scrolling used by default.

Graphic Group X-Axis Settings

To access the x-axis settings of a graphic group, right-click on the x-axis in its display window and select **'Properties'** from the context menu.

The most important choice to make is between XY-plot and Yt-plot. The remaining settings depend on this choice.

Settings for Yt-plot



The **'moving time axis'** determines whether or not the graphic display will scroll automatically to keep up with the current time of day.

The **'time range'** determines the scale of the time axis. When 'moving time axis' is not checked, you can set a specific time range to display.

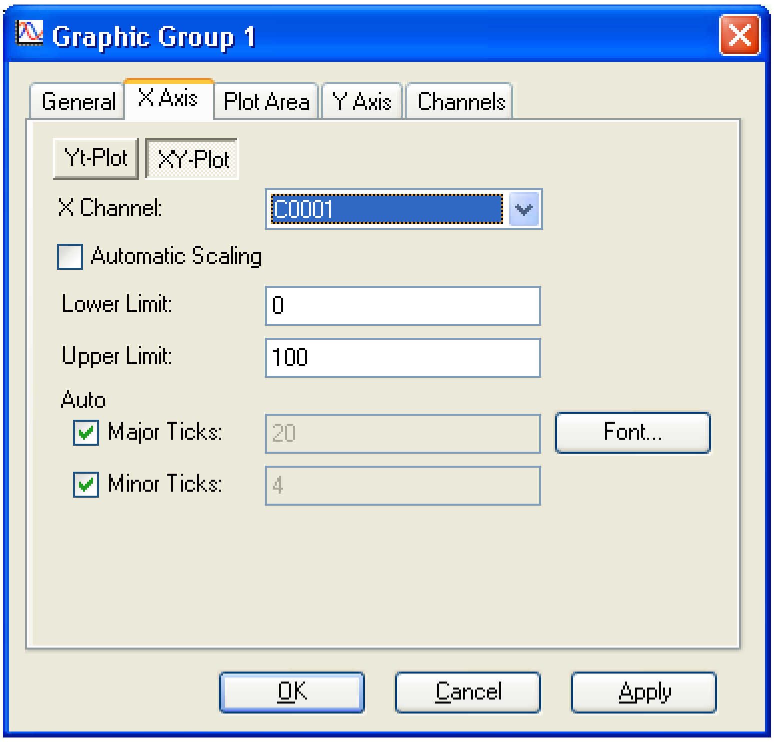
The **'label style'** can be chosen automatically by the software to fit the time range, or you can set it manually. To set your own label style, the following symbols are available:

yyyy	year
MM	month (1-12)
dd	day of month
HH	hour (0-23)
mm	minutes
ss	seconds
fff	milliseconds

The **'major ticks'** and **'minor ticks'** distances are chosen automatically by default, to accommodate for the space used by the labels. When you uncheck the **'auto'** setting you can define your own spacing.

The **'font'** can be used to select the font used for the axis-related labels. This font applies to all axes.

Settings for XY-plot



The **'X Channel'** box defines which measurement point is used for the X-axis in the graph.

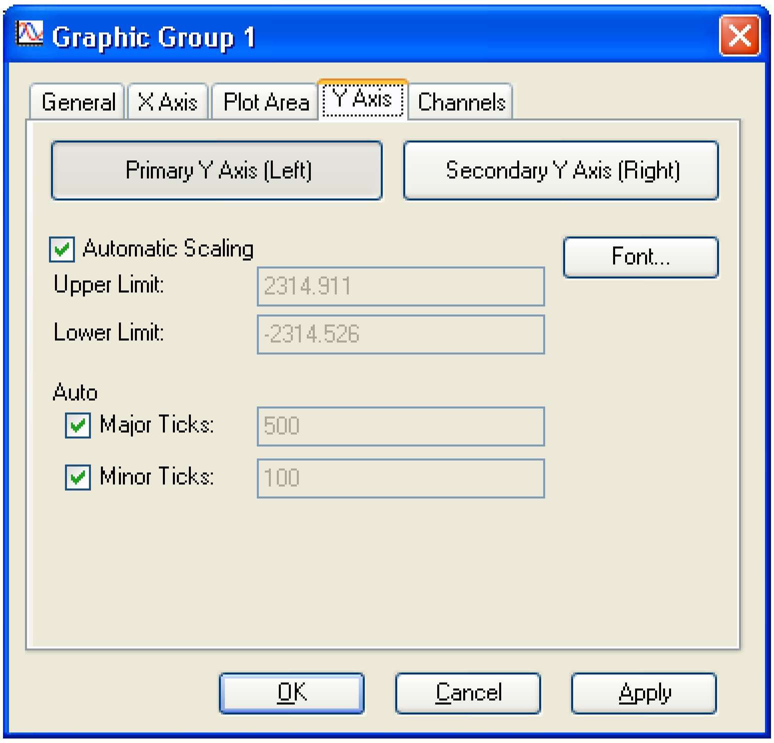
If you check **'Automatic Scaling'**, the X-axis will be scaled to match the values of this channel, otherwise you can set your own limits for the scale.

The **'major ticks'** and **'minor ticks'** distances are chosen automatically by default, to accommodate for the space used by the labels. When you uncheck the **'auto'** setting you can define your own spacing.

The **'font'** can be used to select the font used for the axis-related labels. This font applies to all axes.

Graphic Group Y-Axis Settings

To access the y-axis settings of a graphic group, right-click on the y-axis in its display window and select **'Properties'** from the context menu.



All settings apply to either the primary (left) or the secondary (right) y-axis, depending on the choice made by clicking the buttons at the top.

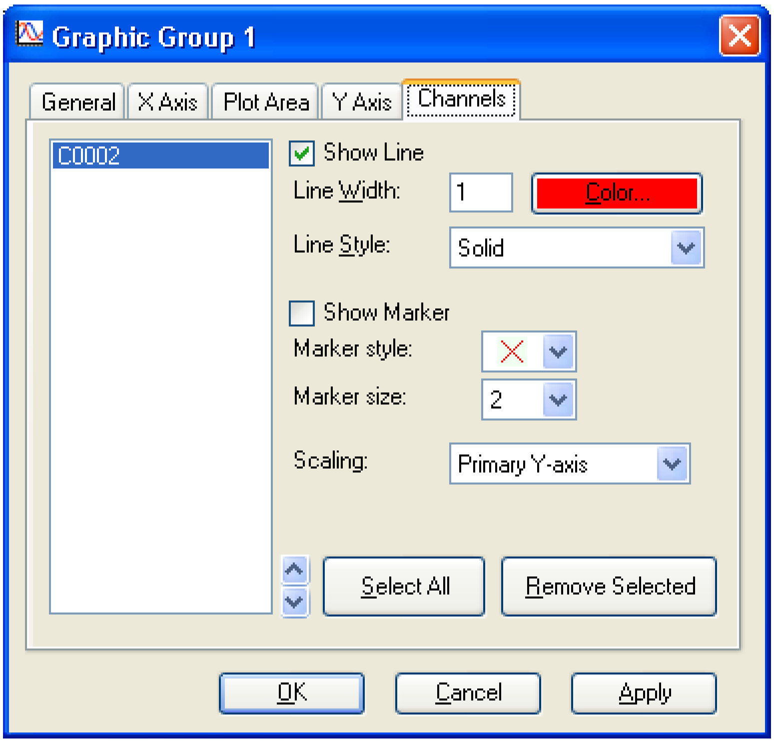
If you check **'Automatic Scaling'**, the X-axis will be scaled to match the values of this channel, otherwise you can set your own limits for the scale.

The **'major ticks'** and **'minor ticks'** distances are chosen automatically by default, to accommodate for the space used by the labels. When you uncheck the **'auto'** setting you can define your own spacing.

The **'font'** can be used to select the font used for the axis-related labels. This font applies to all axes.

Graphic Group Channel Settings

To access the channel settings of a graphic group, right-click on any channel in the legend and select **'Properties'** from the context menu.

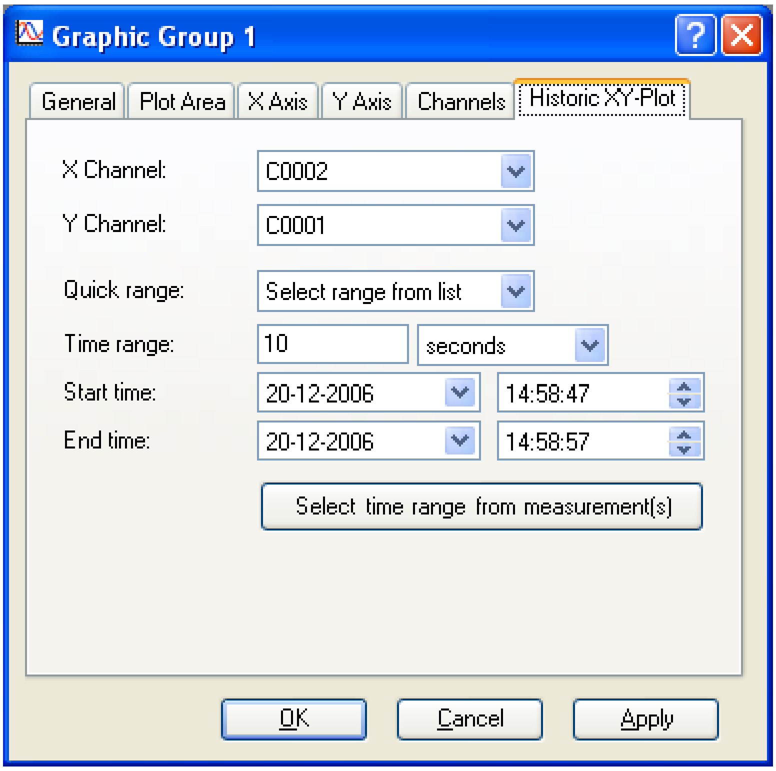


For each measurement point, you can select whether or not to **show a line** connecting the measurement points and whether or not to **show a marker** for each individual measurement point. For the line, you can set the line **width, style and color**, for markers, you can set the **style** and **size**.

The **scaling** box allows you to assign the measurement point to either the primary or the secondary y-axis. You can also scale the measurement point independent of the y-axes. If you choose 'auto scale' the graph for this point will be scaled automatically, independent of other points in the graph. If you choose 'full scale', the graph for this point with be scaled to fit full scale of the measurement point, as defined by the minimum and maximum value in the measurement point setup.

Graphic Group Historic XY-Plot

To access the historic xy-plot settings of a graphic group, right-click on any channel in the legend and select **'Properties'** from the context menu, then click on the tab **'Historic XY-Plot'.**



Using the historic XY-plot tab, you can retrieve a specific measurement or time range from your archived measurement data and show it as a simple XY-plot. For this, you can choose a single channel to use as your X-axis and another channel to use as your Y-axis.

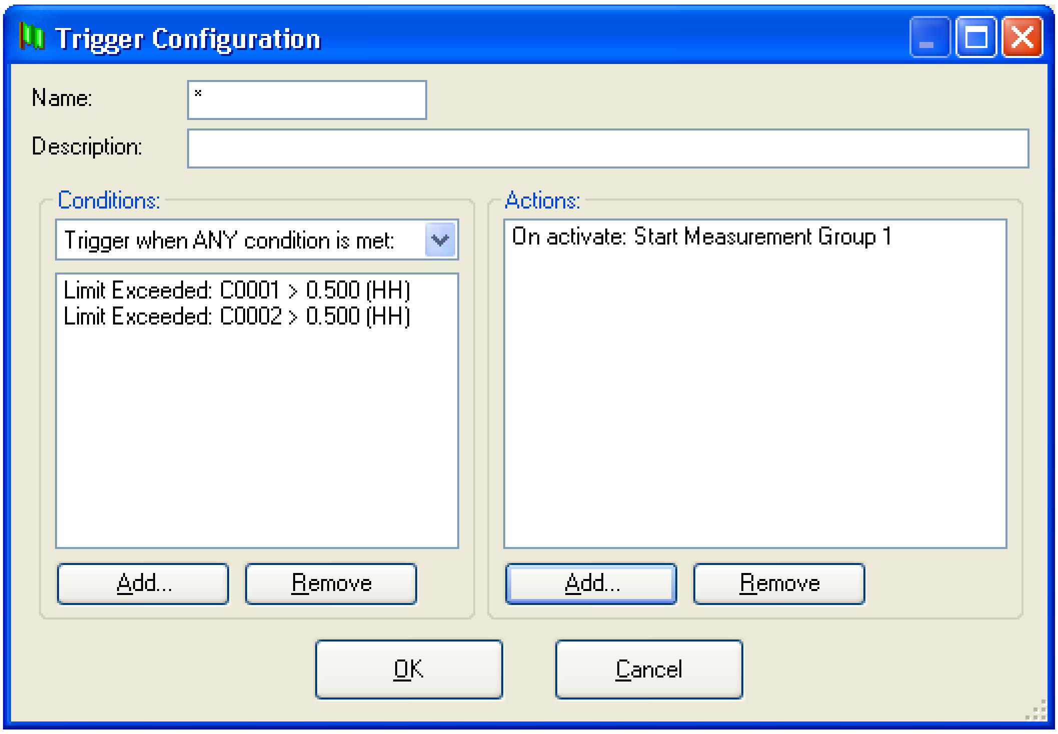
To determine the time range for which you want to see data, you can either use the **'quick range'** box to select recent data, specify the exact time range, or use the **'Select time range from measurement(s)'** button to select a specific historic measurement.

Triggers

Triggers can be used to perform specific actions in response to events. Events can include activating a new configuration, exceedance of a limit or the start of a measurement group. Actions can that can be performed include starting/stopping measurement groups, starting/stopped data storage on a measurement group, changing the measurement speed or setting an output. Actions can be performed immediately after the trigger event occurs, or with some delay, or repeatedly for the duration of the event.

To create a new trigger, right-click the 'Triggers' item in the overview tree and select '**New Triggers**' from the menu.

Trigger Configuration



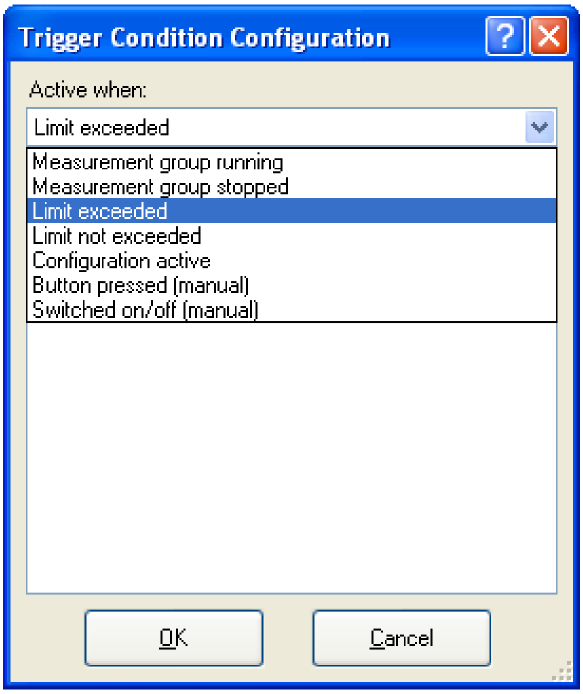
Each trigger has a unique **name** and an optional **description** for documentation purposes. The **name** will be generated automatically if the '*' default value is not changed.

A trigger consists of two parts: the **conditions**, on the left side of the dialog, determine which conditions trigger it, the **actions**, on the right side of the dialog, determine what happens when the trigger occurs.

To add conditions to the trigger, click the **'add'** button on the left side. If multiple conditions are added, you can choose whether the trigger should activate when ANY or when ALL conditions are met.

To add actions to perform, click the **'add'** button on the right side. You can add as many actions as you need. Be aware that, for example, starting a measurement group when a trigger activates does not imply it will stop when the trigger deactivates. You must manually add a second action to the trigger to make this happen.

Trigger Conditions

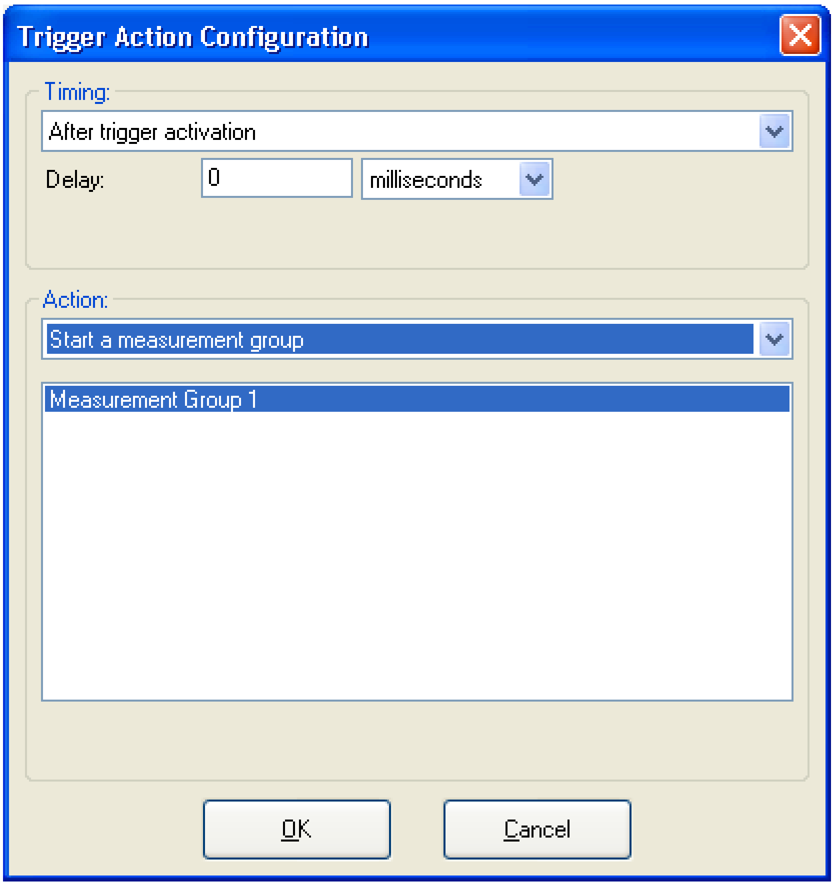


To configure a trigger condition, first select the **'Active when'** criterion. For 'measurement group running/stopped', select a measurement group from the list. For 'limit exceeded/not exceeded', select a limit from the list. If the list shows no limits after selecting this option, make sure you first configure the limit in the measurement point setup.

The '**Configuration active**' option can be used to trigger events after the configuration loads. This way you can, for instance, automatically start a measurement group after you open a saved configuration.

The '**Button pressed**' and '**Switched on/off'** can be used to create manually activated triggers, using a button or switch shown in a separate trigger window (which can be opened using 'Show Trigger Window' in the 'Windows' menu). A button will activate the trigger for as long as it is pressed, and deactivate as soon as it is released. A switch will keep the trigger activated after the first click, and deactivate after the second click.

Trigger Actions



To configure an action that a trigger should perform, first choose when the action should be performed (**'timing'**):

- **After trigger activation**: Perform the action after the conditions for the trigger are met, immediately, or after a configurable delay. If you set a delay, beware that the action will only be performed if the trigger remains activate for the duration of the delay.
- **After trigger deactivation:** Perform the action after the conditions for the trigger are no longer met. If you set a delay, beware that the action will only be performed if the trigger remains inactivate for the duration of the delay.
- **Repeat after activation:** Perform the action after the conditions for the trigger are met, and keep repeating it as long as the trigger remains active.
- **Repeat after activation, absolute start time:** Perform the action after the conditions for the trigger are met, and keep repeating it as long as the trigger remains active. You can use the absolute start time to perform actions at a specific time of day, for instance, every day midnight.

Next, choose what action to perform:

- **Start/stop a measurement group:** Starts or stops the specified measurement group. If the group is already started (or stopped) when the trigger event occurs, no action is performed.
- **Start/stop logging on a measurement group:** Switches data storage on a measurement group on or off. See also the 'store measurement data' option in the measurement group settings.
- **Change measurement speed**: Change the speed of the specified measurement group. This way, you can influence the amount of data collected based on events.
- **Set output**: Set an output point to a specified value, for instance to generate an external alarm.
- **Execute command:** To execute any external file or application in Windows; use the 'Browse...' button to select the application.

- **Balance/tare measurement:** Performs a balance or tare measurement on the specified measurement group.
- **Balance/tare values to zero:** Resets the balance or tare values for point in the specified measurement group.
- **Shunt measurement:** Performs a shunt measurement for the specified measurement group.
- **Clean-up Archives:** If there is a risk that the harddisk runs out of space, there is an option to monitor that (see virtual point functions), and clean up the archive automatically. It is also possible to create a ring buffer that will keep data for the last n days/hours/minutes. **Important**: be careful with this function, otherwise there is a risk of losing measurement data!!
- **Clear graphics group:** The online graph of the selected graphics group is cleared. This is particularly useful for XY-graphs.
- **Start stopwatch (with reset):** Start the selected stopwatch and reset its value to 0.
- **Restart stopwatch (no reset):** Restarts the selected stopwatch, it will continue counting from where it left off before.
- **Stop stopwatch:** Stop (freeze) the stopwatch, the last value will remain unchanged until the next (re)start.
- **Export measurement data:** performs an automatic export of measurement data from the archives. By default, an extra delay of 10 seconds is added before starting this export, to make sure all data is archived to disk before the actual export takes place.

The '**Export measurement data**' option makes it possible to convert archived measurement data to ASCII or any other supported format immediately after a measurement. Typically, a trigger can be set up to act on a manual switch. The associated actions would be to start a measurement group on activation, stop that same measurement group on deactivation, and export measurement data on deactivation.

Alternatively, the export can also be performed while the measurement is running, at a specific interval, by using the 'Repeat after activation' timing for the 'Export measurement data'-action.

The settings for the export should be set up beforehand using the 'Export Measurement Data' dialog from the 'Tools' menu. Simply configure the required export settings and choose 'Close' instead of 'Export' to keep these settings for use with the automatic export function.

The automatic export function will select the appropriate time range, and if no measurement points are selected beforehand, will also choose the measurement points based on the measurement group associated to the trigger.

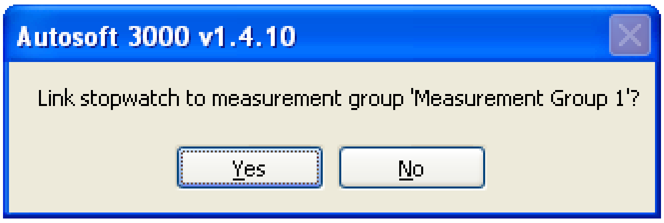
In the exported file name, a '%%'-marker will be replaced by a suitable export name (either a date/time, or the name given to the measurement). For example, when "Export to:" is set to 'C:\data\%%.txt', the actual name during the export could be 'C:\data\Measurement #1.txt', or 'C:\data\20120405-123300.txt'. Any valid date/time of the 'yyyymmdd-hhmmss' format will also be seen as a placeholder to be replaced by the actual export name.

Stopwatch

A stopwatch can be used to track the amount of time (in seconds) since a particular event happened. A common application for a stopwatch is to track how long a measurement group is running.

Another practical use is generating a relative time axis for an online graph. In a graphics group, the stopwatch channel can be used as the x-axis channel (xy-graph). If, in addition to that, the x-axis is set to auto-scale, then the graph will show all measurement values for the entire measurement, automatically compressing the graph as time goes on (see also the important note at the end of this topic).

To add a new stopwatch to a configuration, choose "New Stopwatch" from the "Insert" menu. If a measurement group already exists, Autosoft 3000 will ask if you want to link the stopwatch to this measurement group.



The stopwatch is created as a virtual point (called 'Stopwatch_1' by default) with a specific function: 'stopwatch(1)', you can create up to 4 different stopwatches this way.

The link to a measurement group is created through a trigger (called 'Stopwatch_T1' by default). This trigger has the condition 'Start Running: Measurement Group 1', so it becomes active when measurement group 1 starts, and inactive when it stops.

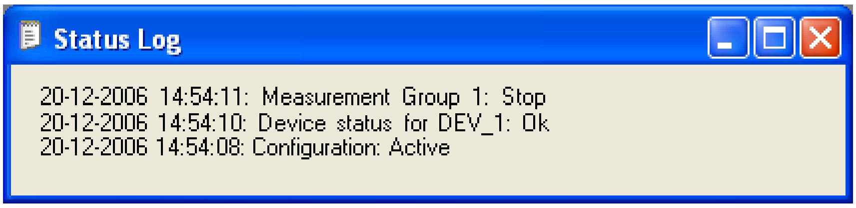
It has two associated actions: on activation the stopwatch gets reset, on deactivation the stopwatch stops.

The current value of the stopwatch(es) can be viewed using menu "Show Stopwatch Window" from the "Window" menu. A right-click in this window shows a menu where you can select the display style.

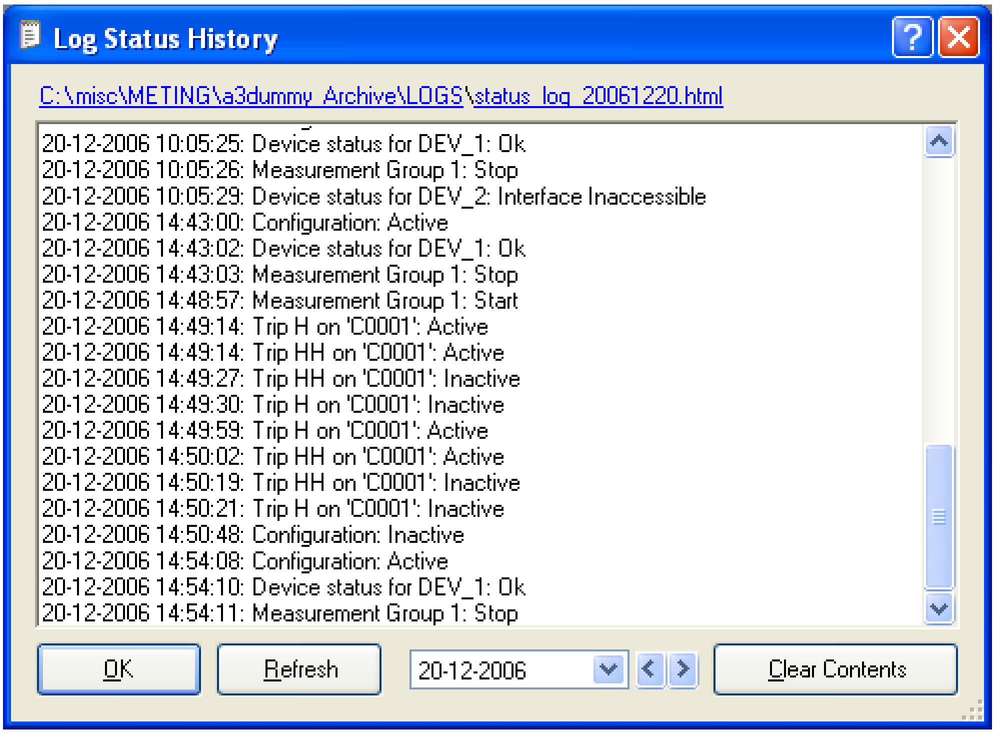
Important:
To log (archive) or show the value of the stopwatch variable in a numeric group, it must be added to the measurement group first. The measurement group determines how often the stopwatch value gets updated outside the special stopwatch window described above.

Status Log View

The status log view opens when you click on the most recent message shown in the status bar at the bottom of the main window, or double-click on the 'Status Log View' entry in the overview tree.



If you double-click on any of the messages in the status log view, the Status Log History will appear. This separate window shows the history of all events related to the current measurement configuration.



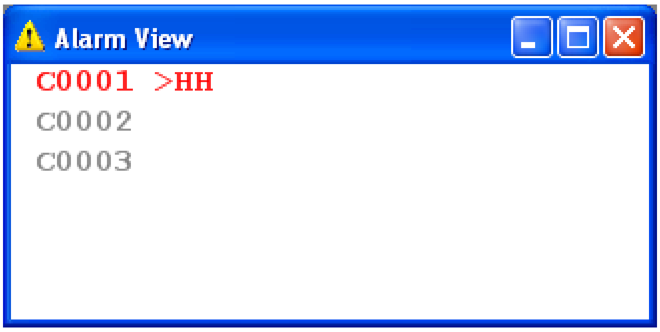
The history is stored in .html files, one file for each day. The line at the top of the window shows the exact location of the file shown in the window. Because the files use an HTML-based format, they can also be viewed in any web browser.

Using the date entry field at the bottom of the window, you can browse through the history of all available status log information.

Alarm View

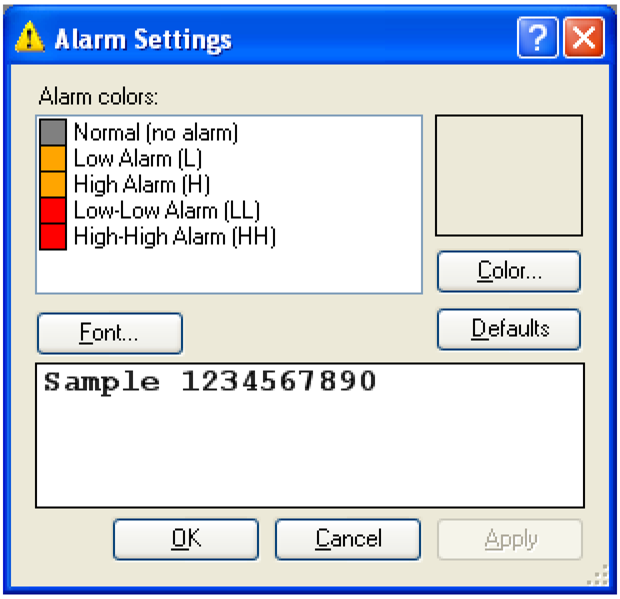
The Alarm View shows the current status of limits on measurement points. Any selection of measurement points can be dragged to the alarm view window, where any limit exceedance is shown using different colors for the different alarm levels.

To open the alarm view window, double-click on the **'Alarm View'** item in the overview tree.



Double-clicking on any of the measurement points in the view opens the limits configuration tab of the selected measurement point, allowing you to review the current settings for each limit.

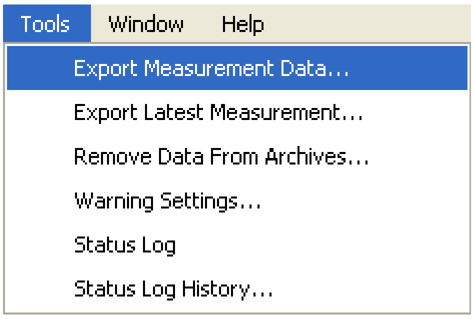
Double-clicking on an empty area of the window (no measurement point selected) opens the Alarm View configuration window.



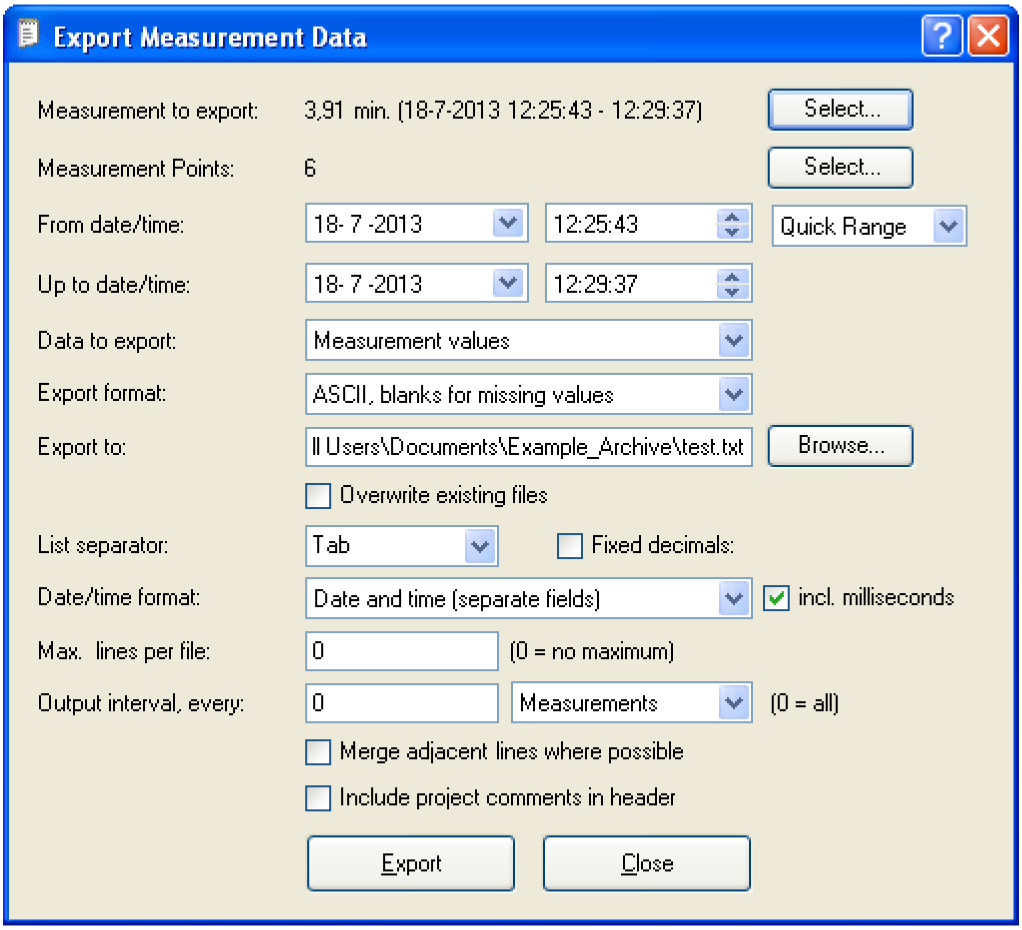
Here, you can select a color for each alarm level and a font for use in the Alarm View window. Note that the colors for the alarm levels are also used in the Graphic Groups, to display alarm levels when you select a single measurement point from the legend.

Export Measurement Data

To retrieve measurement data from the historic archive, choose "Export Measurement Data" from the menu "Tools".



A dialog will show, which allows you to select which data you want to export and what the exported file should look like.



The first line shows the period of time over which measurement data should be exported. Click the **'select'** button next to it to select a specific historic measurement.

The second line shows how many measurement points are selected for export. Click the **'select'** button next to it to make a specific selection of measurement points.

You can manually select a specific time range to export using the **'from date/time'** and **'upto date/time'** fields, or make a **'Quick Range'** selection to retrieve a recent measurement.

The **'data to export'** can be 'normal' measurement values or values from balance, tare or shunt measurements.

You can choose the file name to export to or **'browse'** for a suitable location. If the **'overwrite existing files'** is not checked, a sequence number will automatically be appended to the file name and increased to make sure no existing file gets overwritten.

The **export format** can be ASCII (to file or to clipboard), DIAdem, a list-based format or Matlab .MAT (level 4). ASCII files are tab-seperated by default, making it easy to read them in e.g. Excel. DIAdem output consists of 2 files: 1 ASCII file (extension .DAT) describing the format and 1 binary file (extension .R32) containing the measurement values. List files contain a single line for each measurement value, and are not suitable for high volumes of measurement data.
When you choose to export ASCII data directly to clipboard, you limited to a maximum of 100 measurement points and 30.000 lines of measurement data. Using 'tab' as separator, ASCII data on the clipboard can easily be pasted to applications like Excel.

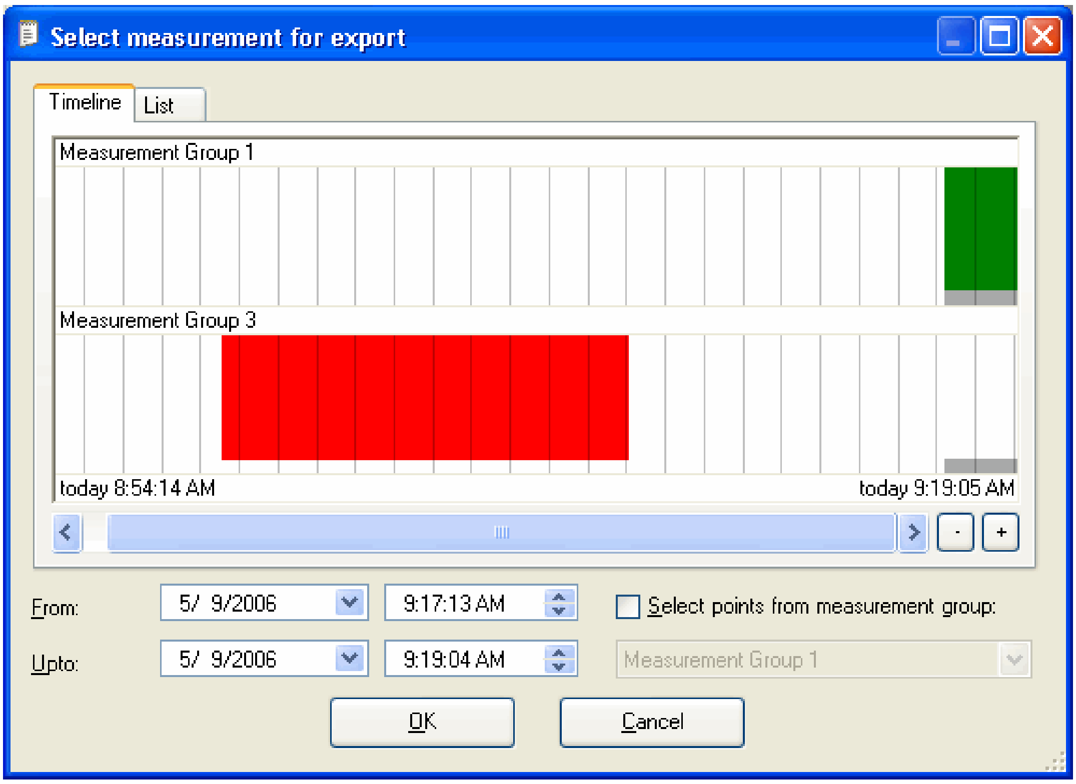
If you set a **'max. lines per file'** the export will create multiple files with increasing sequence numbers as needed to make sure each file does not contain more than the specified amount of lines.

The **'output interval'** can be used to reduce the amount of data retrieved by e.g. only exporting 1 value out of every 10, or exporting 1 value per minute.

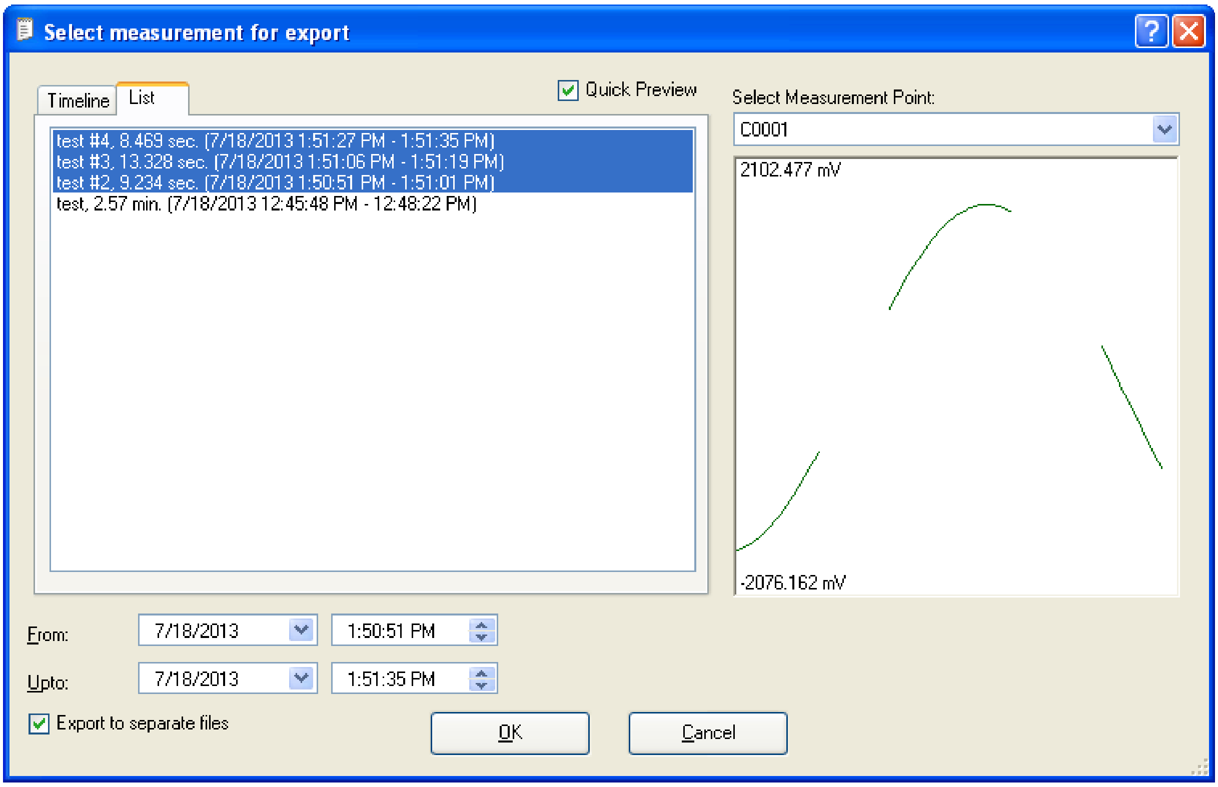
If you check the **'merge adjacent lines where possible'** setting, the export routine will try to merge as many values as possible onto a single line, even if they do not have the same time stamp. This helps to combine data from different sources (devices) that do not supply data for the same measurement at the exact same time.

If you check the '**include project comments in header**' setting, the export will add extra lines containing the configuration comments (menu 'File', 'Edit Comments...') to the exported data.

Export Measurement Selection



The **timeline** tab shows a graphic time line of measurements performed. You can select a timeframe to export by clicking on a single measurement, clicking and dragging to select a specific time range, or shift-clicking to extend a time range to include multiple measurements. Use the scroll-wheel on your mouse to quickly zoom in and out (or use the +/- buttons in the dialog if your mouse does not have a scroll-wheel).
Note that measurement ranges are shown by a colored rectangle, and ranges where a measurement group was running but not storing data are shown in a lighter color.



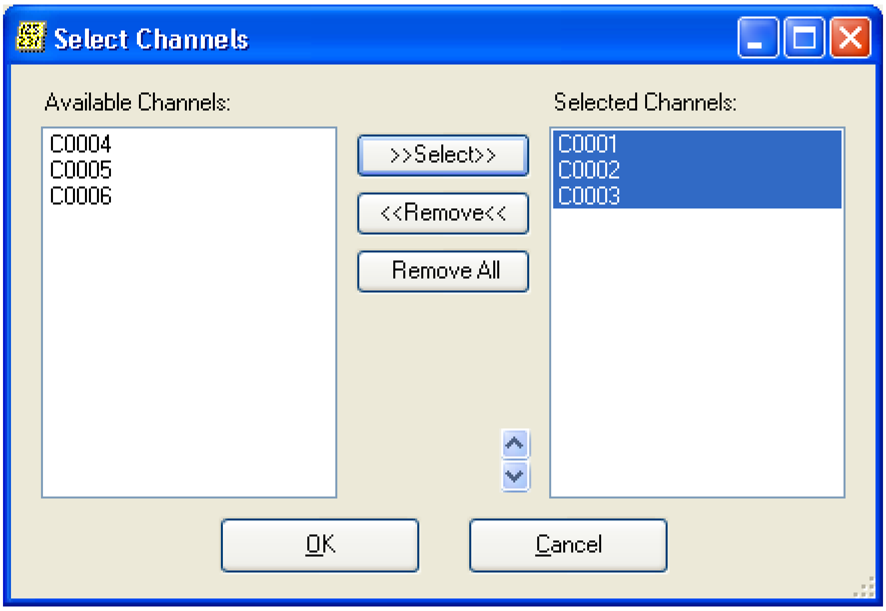
Use the **list** tab to select measurements from a list. The list shows measurements in reverse chronological order.

The **Quick Preview** shows the content of the archive for a specific measurement point, to help you locate the correct data.

When you check the **'Select points from measurement group'** this dialog will not only select the time range, but also the measurement points in the specified group for export.

You can select multiple items from the list using the shift-key. By default, this will select a time range containing all of those items, and generate a single export file for that range. Alternately, you can check '**Export to separate files**' to create multiple export files, one for each selected entry in the list.

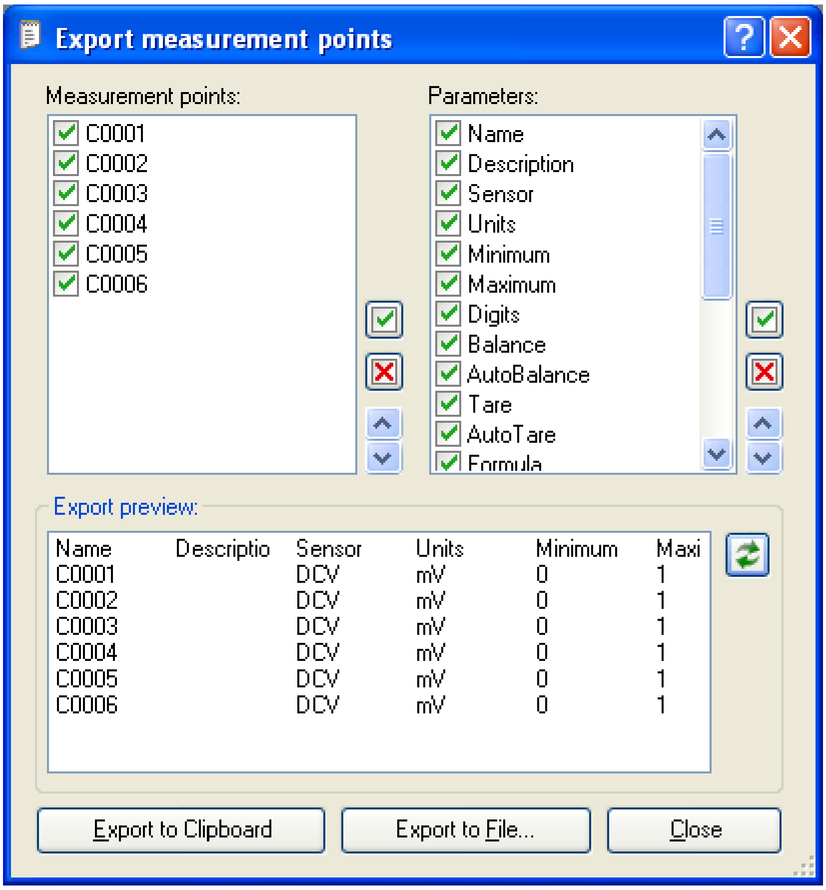
Export Point Selection



To select measurement point to export: click and mark channels on the left side and click **'>>select>>'** to add them to the list of selected channels. To change the order in which the channels appear in the export file: mark selected channels (click to mark a single channel, shift-click to mark a range and ctrl-click to mark additional channels) and use the up/down arrows (above the cancel button) to mode the channels to a new place in the list.

Export Measurement Points

To export parameters from any or all measurement points, choose **'Export Measurement Points...**' from the **'File'** menu. A dialog will appear, allowing you to select which measurement points and which parameters you want to export.



To select which measurement points you want to export, use click, shift-click and ctrl-click to select any combination of measurement points from the list, then use the buttons on the right of the list to select or deselect them. You can also move the marked measurement points to another position in the list by using the up/down-arrows on the right of the list. Similarly, you can choose the parameters to export.

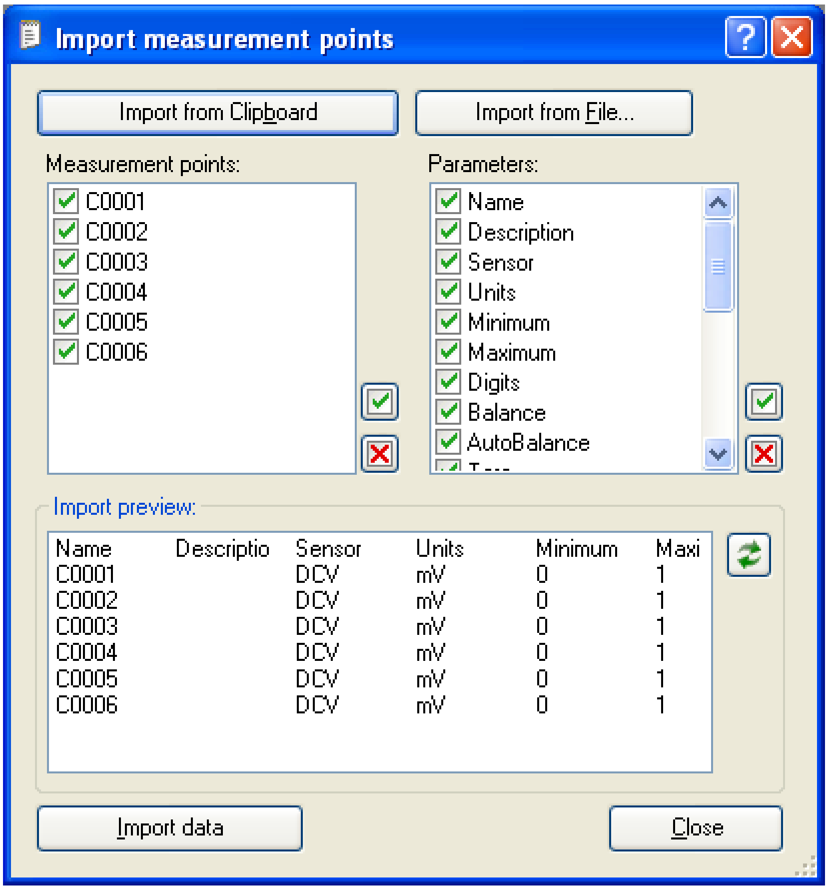
Click on the 'refresh' button (the one with the green arrows pointing left and right) to refresh the 'export preview', which shows you a rough indication of what the exported data will look like.

After making your selection, use **'Export to Clipboard'** or **'Export to File'** to export the data.

You can load this information in e.g. Excel, make modifications, and then import the data back into Autosoft 3000.

Import Measurement Points

To import parameters for measurement points, choose **'Import Measurement Points...**' from the **'File'** menu. A dialog will appear, allowing you to select which data you want to import.



First, you select the source data by clicking **'Import from Clipboard'** or **'Import from File...'**. The dialog will now show you which measurement points and parameters are available in your source data. Each parameter must be marked by a specific column header, which tells Autosoft 3000 how to interpret the data. Use the 'Export Measurement Points' function to find out how the imported data should be formatted. You can simply import all data, or make a selection of measurement points/parameters you want to import.

To select which measurement points you want to import, use click, shift-click and ctrl-click to select any combination of measurement points from the list, then use the buttons on the right of the list to select or deselect them. You can also move the marked measurement points to another position in the list by using the up/down-arrows on the right of the list. Similarly, you can choose the parameters to import.

Click on the 'refresh' button (the one with the green arrows pointing left and right) to refresh the 'import preview', which shows you a rough indication of what data will be imported.

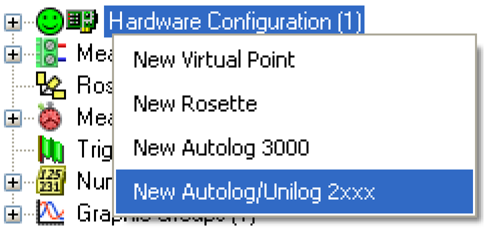
Click **'Import data'** to confirm your action and import the selected parameters into your configuration. If the import data contains improperly formatted input, invalid data, or data that is outside to allowable range for a specific parameter, a warning dialog will show the exact point where invalid data was detected.

Autolog/Unilog 2xxx

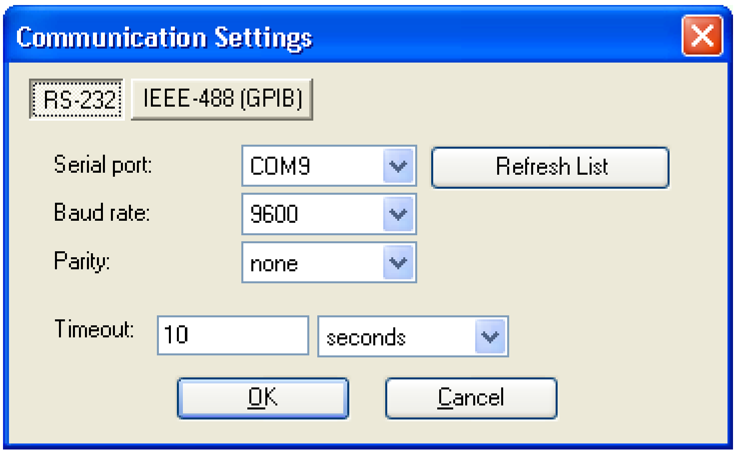
This chapter describes software functions specific to the Autolog 20xx, Autolog 2100 and Unilog 2500 devices.

Adding an Autolog/Unilog 2xxx

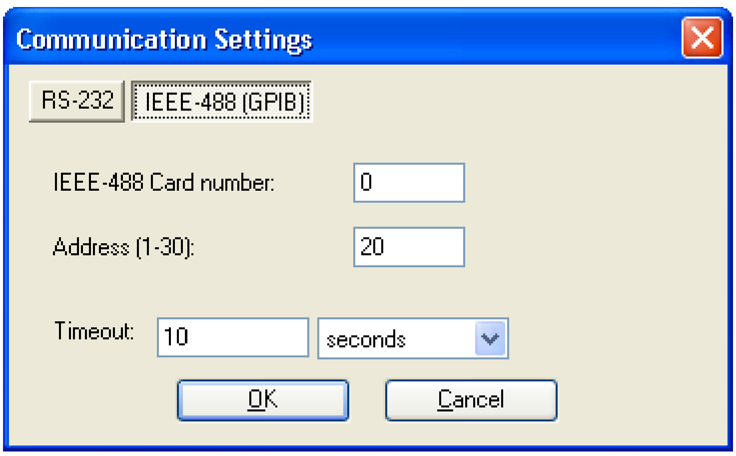
To add a new Autolog/Unilog 2xxx to your configuration, right-click the 'Hardware Configuration' item in the overview tree and select '**New Autolog/Unilog 2xxx**' from the menu.



A dialog appears where you can choose the communication settings for the device.



For RS-232 you need to select the COM-port from the list of available ones. If the COM-port you want to select is visible in the list, make sure it is not currently in use by another program. Click 'Refresh List' to check the available COM-ports again. You will also need to select the proper baudrate. Parity should be set to 'none' unless you have a very specific reason to change it.



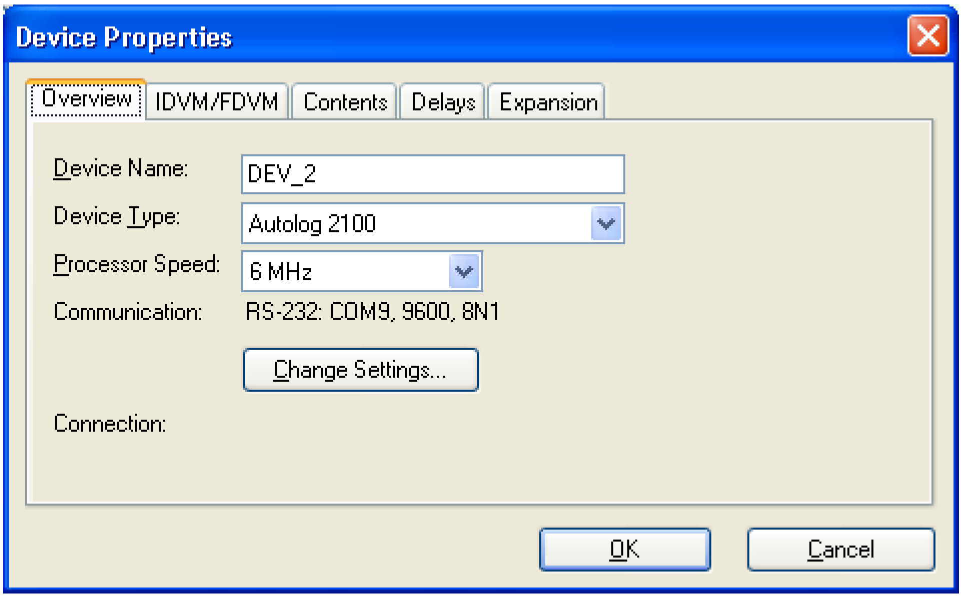
For IEEE-488 you need to select the card number and device address. We advise the use of an ISA GPIB-card from National Instruments. Cards from other manufacturers as well as PCI cards from National Instruments are not guaranteed to communicate properly with the Autolog/Unilog devices and driver. Be sure you have proper and recent drivers installed if you have trouble with the IEEE communication.

You will also need to provide a timeout time for communications with the device. The best value depends on the longest scantime of the device. If you do not know how to choose a timeout value, use 1 second for every 15 channels in the device (count channels including all extension boxes), with a minimum of 1 second. For example, if you want to scan 300 channels, use a timeout of 20 seconds.

After you press 'OK' Autosoft 3000 will attempt to connect to the device and will present the Device Properties dialog to allow detailed configuration of the device.

***Note:* The Autolog/Unilog 2xxx device should also be configured for correct type of communication. Check your hardware manual to find how to set the dipswitches inside the device to the proper position.**

Device Properties: Overview



The **overview** tab in the shows the device name, type and communication settings.

Items, which are grayed out, cannot be altered because the connected device dictates the correct setting.

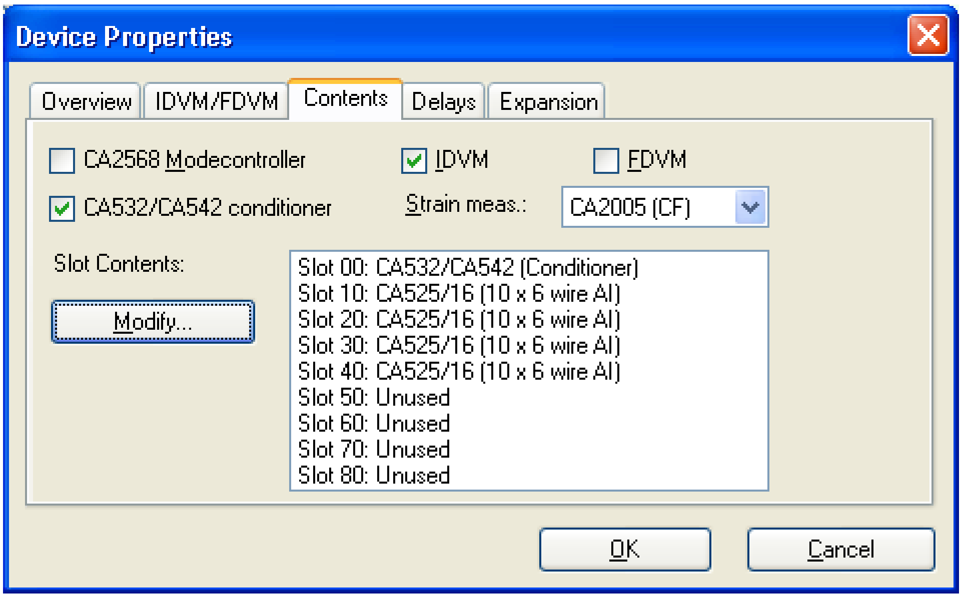
If no device is connected, you can select the device type yourself. The selected device type will influence the available choices in other parts of the configuration, so make sure you choose the correct one.

The device name can be set to anything you like, although the software will automatically impose some restrictions on the length and characters you can use.

This view also shows if the device is connected. If it is not, you can use the 'Change Settings' button to go back to the communication settings.

After you have done basic configuration of device name and type, you should proceed to the contents tab to configure the contents of the device (main unit).

Device Properties: Contents



Items, which are grayed out cannot be altered because the connected device dictates the correct setting.

The device cannot detect the 'CA532/CA542 Conditioner' option. As most devices are delivered with either CA532 or CA542, this option is checked by default. When you are unsure, check the device itself for its contents.

***Beware:* Make sure you correctly configure the availability of items like mode controller and conditioner, before configuring input/output cards and their channels. Adding or removing them later on can cause a shift in slot numbering and should therefore be avoided.**

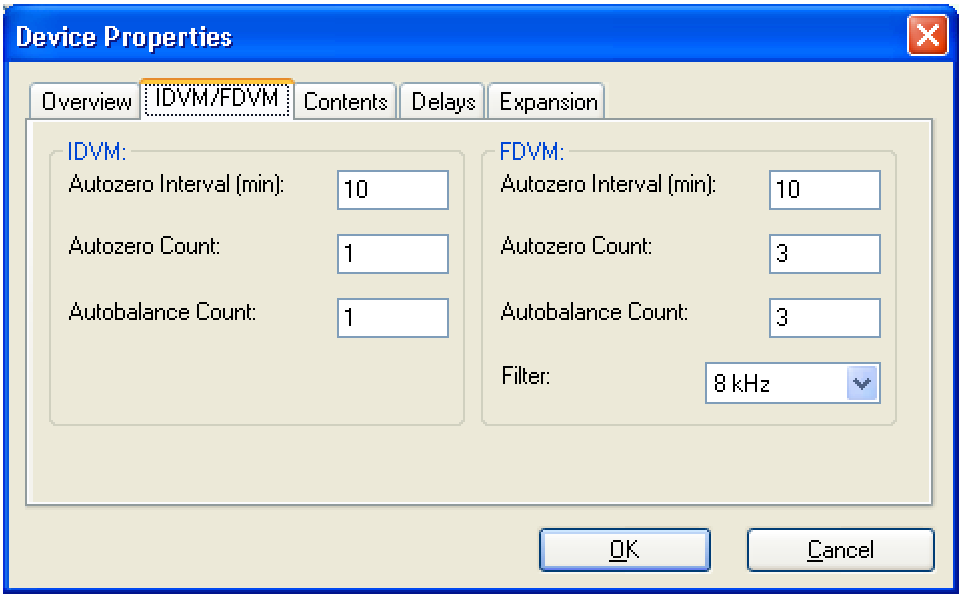
The lower part of this dialog shows the contents of the slots. If you look at the actual device from the rear, you should match this list to the device contents, with the topmost item in the list corresponding to the leftmost card in the device. Use the 'Modify' button to open the Card Selection Dialog, which allows you to add or remove input/output cards.

***Note:* If you are configuring an Autolog 20xx (for example a 2005) you must check the Expansion tab to see the correct CA-512 settings for the contents you configured. On the CA-512 card you can find 4 hex-coded rotary switches: 'A' and 'B' are used for the start adress, 'C' and 'D' for the number of channels per slot. Refer to your CA-512 hardware manual for information on how to adjust the rotary settings on the card.**

After you have configured the contents of the device (main unit) you can check the IDVM/FDVM Settings and the Delay Settings (the default settings for both of these tabs will work in most cases, except for fast measurements).

If you have one or more expansion boxes connected to your main unit, you should configure their contents using the Expansion Settings.

Device Properties: IDVM/FDVM



This tab shows the configuration options for the IDVM (Integrating Digital Volt Meter) and FDVM (Fast Digital Volt Meter).

Both DVM's drift slightly over time. Therefore, zero measurements must be made every now and then during longer measurements to determine the drift and compensate for it. The Autolog or Unilog device can do this automatically. By default, an autozero measurement is done every 10 minutes for both IDVM and FDVM.

Making an autozero measurement costs time and temporarily interrupts the normal measurement flow. If you want to avoid this interruption and you do not care too much about the possible drift (for example, because your measurement only lasts a few minutes), you can switch the autozero measurement off by setting both the interval and the count to 0.

***Note:* During continuous measurements autozero measurements are automatically switched off (Continuous means, in this context: a special measurement mode where the device measures as fast as possible, without any interruption).**

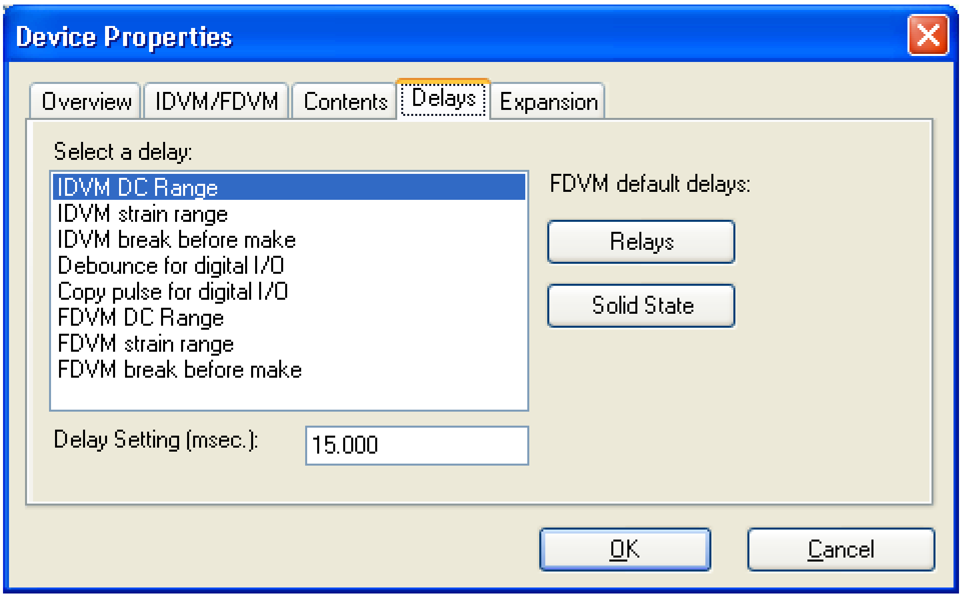
The autozero count gives the option to increase the number of zero measurements made during each autozero measurement. The measurements will be averaged to measure the zero value more accurately. This option is very useful for the FDVM, which is very sensitive to noise, but less so for the IDVM, which already integrates the signal over a period of time (about 20 msec.).

The autobalance count has the same use as the autozero count, but this time for autobalance measurements, which can be made using the Autobalance Dialog. Autobalance measurements allow you to set the input signal to zero and have the software measure. The measured value will be subtracted from all further measurements so the measured autobalance value effectively becomes the zero value for that channel.

The FDVM filter option allows for a 20 kHz, 8 kHz, 1 kHz or no filter at all to be used for FDVM measurements. The default 8 kHz filter should suffice in most situations.

***Beware:* Setting the filter to none will cause an inordinate amount of noise to be measured by the FDVM.**

Device Properties: Delays



This tab in shows a list of delay settings for the Autolog/Unilog 2xxx device. By default, these delays are configured for use with relay input cards. If you want to perform fast measurements, using an FDVM and solid state input cards (CA528), you can press the 'Solid State' button to decrease the FDVM delay times.

To modify an individual delay, first select it from the list (click on it), then enter the new value in milliseconds.

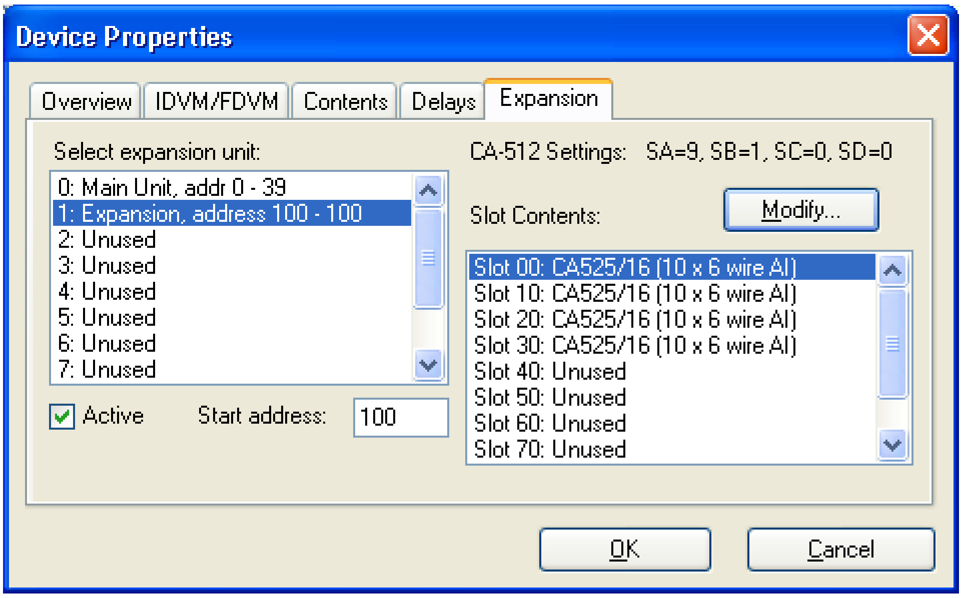
The default delay times in milliseconds for the different items are:

Item	**Relays**	**Solid State**
IDVM DC Range	15 ms	15 ms
IDVM strain range	25 ms	25 ms
IDVM break before make	0.5 ms	0.5 ms
Debounce for digital I/O	0.5 ms	0.5 ms
Copy pulse for digital I/O	10 ms	10 ms
FDVM DC Range	15 ms	0.3255 ms
FDVM strain range	25 ms	0.3255 ms
FDVM break before make	0.5 ms	0.1 ms

The Solid State default delay times assume that no filter is used. If you select another filter setting you should modify the 'FDVM DC Range' and 'FDVM Strain Range' to match the following table:

Filter Setting	**Solid State delay**
none	0.3255 ms (Solid State default)
20 kHz	at least 0.4 ms
8 kHz	at least 0.5 ms
1 kHz	at least 4 ms

Device Properties: Expansion

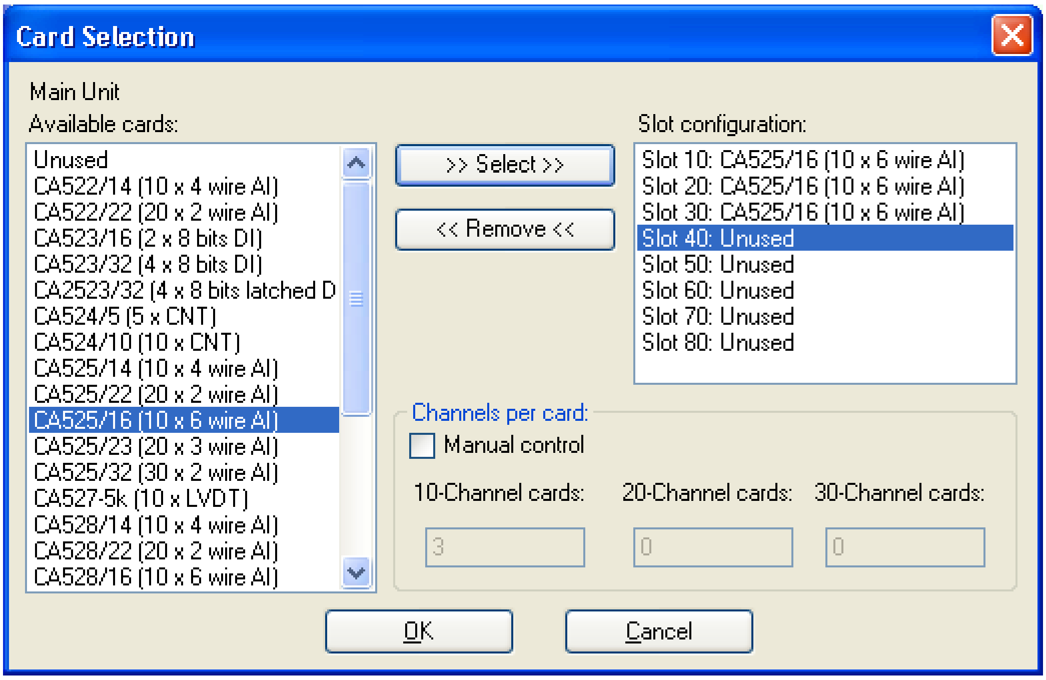


In this tab, you can configure the input/output cards of the main device and all expansion units. On the left side is a list on all available units. To configure a new expansion box, click on the first unused one in the list and check the 'Active' box to activate it. A default start address for the unit will be filled in, but you can change it to your liking, as long as the address range does not overlap with one of the other units.

The right part of this dialog shows the contents of the slots for the currently selected unit. If you look at the actual device or expansion box from the rear, you should match this list to the contents of the device or expansion box, with the topmost item in the list corresponding to the leftmost card in the device. On expansion boxes, the leftmost slots containing the CA-512 and CA-532/542 cards should be ignored. Slot 00 is the slot to the right of the CA-532/542 card. Use the 'Modify' button to open the Card Selection Dialog, which allows you to add or remove input/output cards.

Note: **After configuring the contents of an expansion box or the main unit of an Autolog 20xx (for example a 2005) you must adjust the settings of the CA-512 card contained therein to match those shown on the top right of this dialog. On the CA-512 card you can find 4 hex-coded rotary switches: 'SA' and 'SB' are used for the start adress, 'SC' and 'SD' for the number of channels per slot. Refer to your CA-512 hardware manual for information on how to adjust the rotary settings on the card.**

Card Selection Dialog



The right part of this dialog shows the contents of the slots for the currently selected unit. If you look at the actual device or expansion box from the rear, you should match this list to the contents of the device or expansion box, with the topmost item in the list corresponding to the leftmost card in the device. On expansion boxes, the leftmost slots containing the CA-512 and CA-532/542 cards should be ignored. Slot 00 is the slot to the right of the CA-532/542 card.

To place new cards in the slots, first select the type of card from the list on the left side. Then select one or more slots from the list on the right and press the 'Select' button to place the card.

To remove cards from the slots, select the cards in the list on the right and press the 'Remove' button.

When placing cards with different numbers of channels in the device, you should place cards with a low numbers of channels in the first slots and cards with higher numbers of channels in higher slot numbers. That means: put 20 channel cards to the right of 10 channel cards and 30 channel cards to the right of 20 channel cards.

The software will automatically determine how many slots have 10 channel cards, 20 channel cards and 30 channel cards, which you can see on the bottom right of the dialog. These values determine the settings of the CA-512 card (in the Autolog 2100 and Unilog base devices these settings are controlled by software). You can manually control these values, but you should only take manual control if you have a very specific reason for it.

The following cards are available for configuration in Autolog or Unilog devices:

CA522/14	Card for 10 analog inputs using up to 4 wires per input, specifically designed for ThermoCouple measurements. Usually the first 9 channels are connected to thermocouples (channels 0...8) and the 10^{th} channel is used for CJC with a Pt-100 (channel 9).
CA522/22	Card for 20 analog inputs using 2 wires per input, specifically designed for ThermoCouple measurements. Usually 18 channels are connected to thermocouples (channels 0...8 and 10...18), the 10^{th} channel is used for CJC with a Pt-100 (channel 9) and the 20^{th}

	channel is reserved (internally also used by the Pt-100).
CA523/16	Card for 16 digital inputs, grouped in 2 channels containing 8 bits each. The input channels will measure values between 0 and 255.
CA523/32	Card for 32 digital inputs, grouped in 4 channels containing 8 bits each. The input channels will measure values between 0 and 255.
CA2523/32	Card for 32 latched digital inputs, grouped in 4 channels containing 8 bits each. The input channels will measure values between 0 and 255.
CA524/5	Card for 5 16-bit counters. These counters can be set to frequency/interval or totalyzer using jumpers on the card.
CA524/10	Card for 10 16-bit counters. These counters can be set to frequency/interval or totalyzer using jumpers on the card.
CA525/14	Card for 10 analog relay inputs using up to 4 wires per input. Suitable for DC, ThermoCouple, Pt-100 and 4-wire Quarter Bridge strain measurements.
CA525/22	Card for 20 analog relay inputs using up to 2 wires per input. Suitable for DC and ThermoCouple measurements.
CA525/16	Card for 10 analog relay inputs using up to 6 wires per input. Suitable for DC, ThermoCouple, Pt-100 and strain measurements.
CA525/23	Card for 20 analog relay inputs using up to 3 wires per input. Suitable for DC, ThermoCouple and 3-wire Pt-100 measurements.
CA525/32	Card for 30 analog relay inputs using up to 2 wires per input. Suitable for DC and ThermoCouple measurements.
CA527-5k	Card for 10 analog LVDT inputs using a 5 kHz carrier frequency.
CA528/14	Card for 10 analog solid state inputs using up to 4 wires per input. Suitable for DC, ThermoCouple, Pt-100 and 4-wire Quarter Bridge constant current strain measurements.
CA528/22	Card for 20 analog solid state inputs using up to 2 wires per input. Suitable for DC and ThermoCouple measurements.
CA528/16	Card for 10 analog relay inputs using up to 6 wires per input. Suitable for DC, ThermoCouple, Pt-100 and constant current strain measurements.
CA528/23	Card for 20 analog solid state inputs using up to 3 wires per input. Suitable for DC, ThermoCouple and 3-wire Pt-100 measurements.
CA528/32	Card for 30 analog solid state inputs using up to 2 wires per input. Suitable for DC and ThermoCouple measurements.
CA574	Card for 32 digital outputs, grouped in 4 channels containing 8 bits each.
CA575/10	Card for 10 digital relay outputs, 1 output per channel.
CA575/20	Card for 20 digital relay outputs, 1 output per channel.

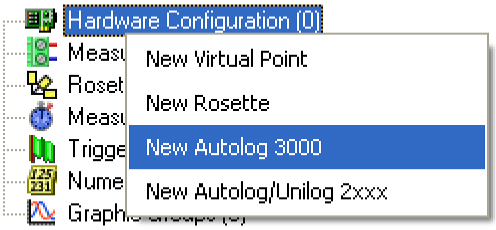
CA576	Card for 8 8-bit analog outputs. The output channels can be set to values between 0 and 255. Values from 0 to 127 correspond to 0 to +10 Volts, 128 to 255 correspond to -10 to 0 Volts.
CA577	Card for 2 14-bit analog outputs. The output channels can be set to values between 0 and 16383. The corresponding output voltage range can be set using jumpers on the card.
CA591	Card for miscellaneous purposes. Although the program allows the placement of this card in the device, it will not be able to control it in any way. No channels can be assigned to this card.
CA592	AC to DC converter card. Although the program allows the placement of this card in the device, it will not be able to control it in any way. No channels can be assigned to this card.
CA593	Internal resistance and isolation measurement card. This card is only allowed in the first available slot in the main unit of a device. Although the program allows the placement of this card in the device, it will not be able to control it in any way. No channels can be assigned to this card.

Autolog 3000

This chapter describes software functions specific to Autolog 3000.

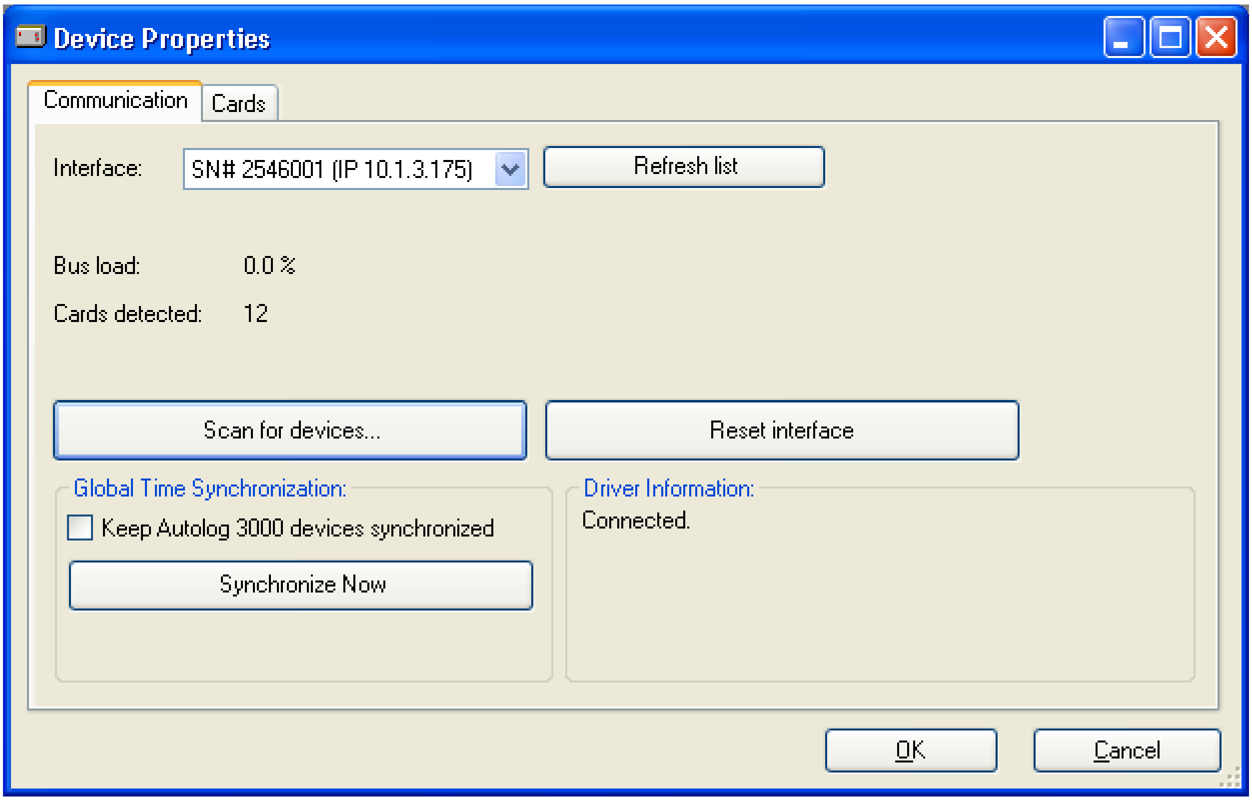
Adding an Autolog 3000

To add a new Autolog 3000 to your configuration, right-click the 'Hardware Configuration' item in the overview tree and select '**New Autolog 3000**' from the menu.



The device properties dialog appears, where you can configure the device and retrieve its configuration.

Device Properties: Communication



When adding a new Autolog 3000 device to the configuration, the first thing to do is to select the communication **interface** from the dropdown list. Devices connected through the CAN bus will use the CAN convertor interface, devices connected through an USB or ethernet interface are shown as separate entries in this list.

If a connected USB device does not show in the list, check that it is properly connected and that it shows as an additional COM-port in the Windows Device Manager. Press th**e 'Refresh list'** button to update the list of available interfaces.

If a connected ethernet device does not show in the list, be aware that it can take a few minutes for the Autolog 3000 to start and connect to the ethernet. If the device was switched on only recently, please wait a bit and press the **'Refresh list'** to see if it gets detected. Also note that the automatic detection of the Autolog 3000 depends on the device being in the same subnet as the PC (preferably connected through a private, local ethernet network) and must be able to respond to an UDP broadcast at port 5049 (i.e. this port must not be blocked by a firewall and the subnet mask must be configured correctly).

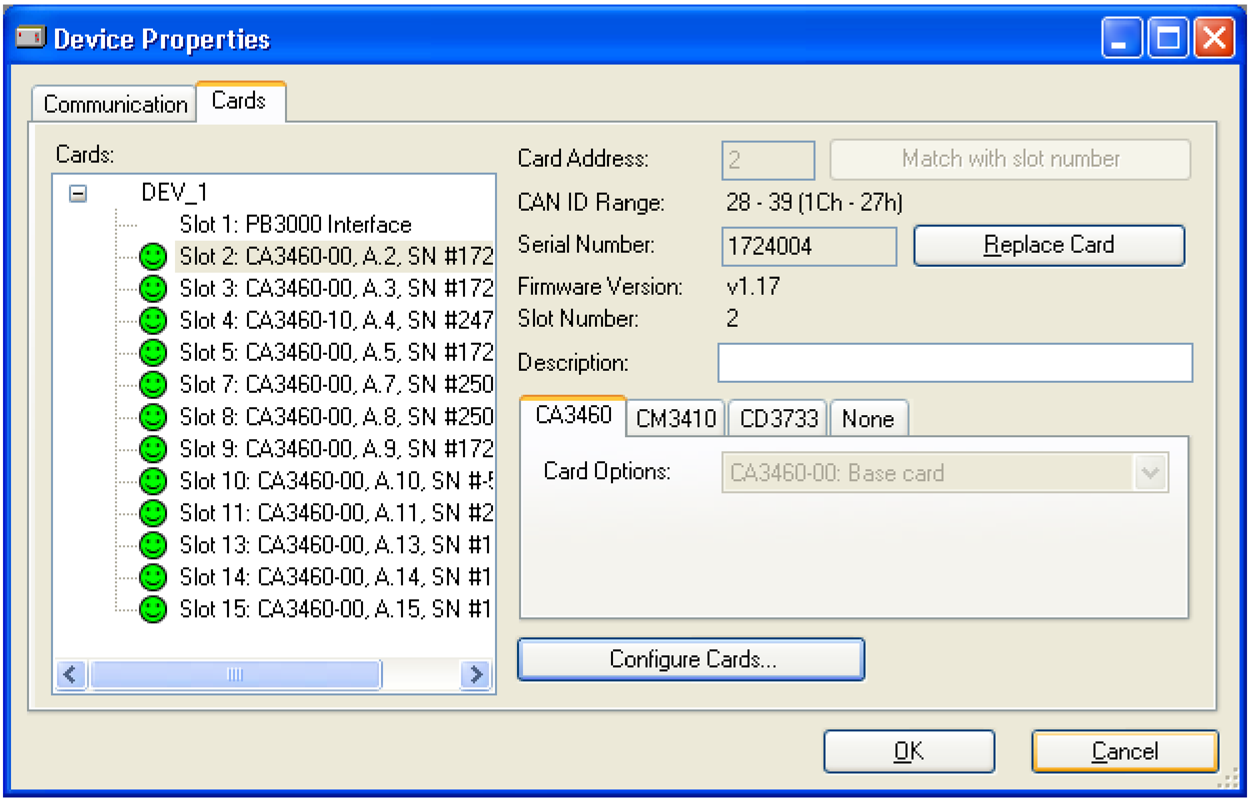
After selecting the interface, click the **'scan for devices'** to retrieve information about the device(s) connected to the interface. If successful, the **'cards detected'** line should show the number of measurement cards available in the device(s).

If a communication problem occurs, the **'reset interface'** button will close and re-open the communication with the device, which may help to re-establish communication.

The **'global time synchronization'** section is used to synchronize the time stamps for measurement data from multiple devices. Without synchronization, the PC has no way to accurately compare the time stamps of measurement values received from different communication interfaces, so measurements performed at the same time may gets time labels several hundred milliseconds apart. To ensure measurements from multiple Autolog 3000 devices a synchronous, you need to use a synchronization cable to allow the Autolog 3000 devices to time their measurements on a single reference clock. If, and only if, this synchronization cable is properly connected, should you use the **'Keep Autolog 3000 devices synchronized'** check box to indicate to the software that the time stamps from these devices are comparable and should get the same clock time assigned.
The **'Synchronize now'** button will send a synchronization command to all Autolog 3000 device, causing a reset of the internal counters. The connected synchronization cable will then ensure that the counters will increase at the same rate, using the same reference clock.

In short, to ensure measurement data from multiple Autolog 3000 devices is processed in a time synchronous manner, you must have both a synchronization cable to connect the devices and check the 'Keep Autolog 3000 devices synchronized' for the software to process the data correctly.

Device Properties: Cards



This part of the Autolog 3000 device properties dialog shows the list of detected devices and cards. The tree view on the left shows the devices and the cards, the items on the right show information on the currently selected item.

For each CM3410 multiplexer card, you should set the multiplexer configuration in this dialog. First select the card in the tree on the left, then use the **'Group mux'** settings on the right to configure the use of each of the three available groups of input channels. Refer to the Autolog 3000 manual to find which settings (number of wires) are appropriate for the various sensor types that can be connected to the CM3410 multiplexer card.

Replacing a card
To replace a card in the Autolog 3000 system, follow one of the procedures below, depending on the type of interface used to communicate with the PC.

CAN interface
When a CAN interface is used, all cards are assigned a unique CAN address. When you replace a card, the new card will get a new unique CAN address and needs to be specifically configured to act as a replacement for the old card. Follow these steps to make the replacement:

1. Close Autosoft 3000 and make a backup of the configuration file.
2. Note the serial number of the replacement card, you will need it later.
3. Switch off the device, and replace the card.
4. Switch the device back on and load the configuration. Now go to the device properties (see window above), and select the card that you replaced. It should show a red smiley to indicate that the new card does not match with the current configuration.
5. Manually type in the serial number of the replacement card, then press the 'Replace Card' button. The software should now transfer the settings of the old card to the new one.

USB or Ethernet interface
For these interfaces, cards are identified by the slot number they are placed in. This makes replacing a

card easier than using the CAN interface. Follow these steps to make the replacement:

1. Close Autosoft 3000 and make a backup of the configuration file.
2. Switch off the device, and replace the card.
3. Switch the device back on and load the configuration. Now go to the device properties (see window above), and select the card that you replaced. It should show a red smiley to indicate that the new card does not match with the current configuration.

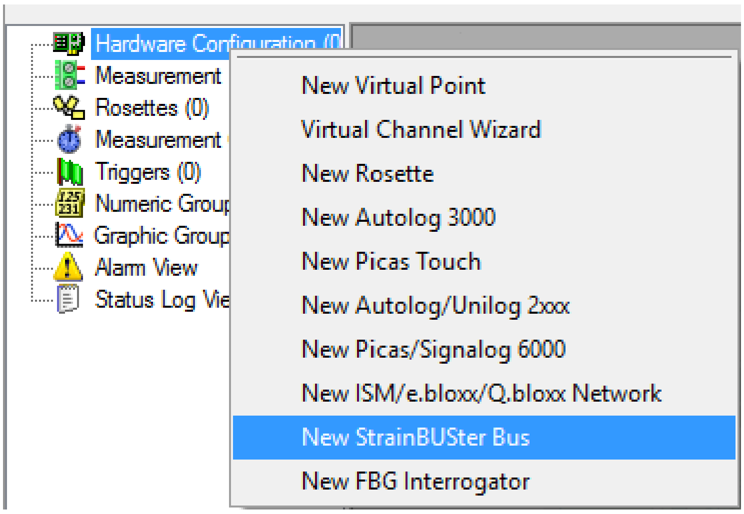
Press the 'Replace Card' button. The software should the new card in the slot and transfer the settings of the old card to the new one.

StrainBUSter

This chapter describes software functions specific to StrainBUSter modules.

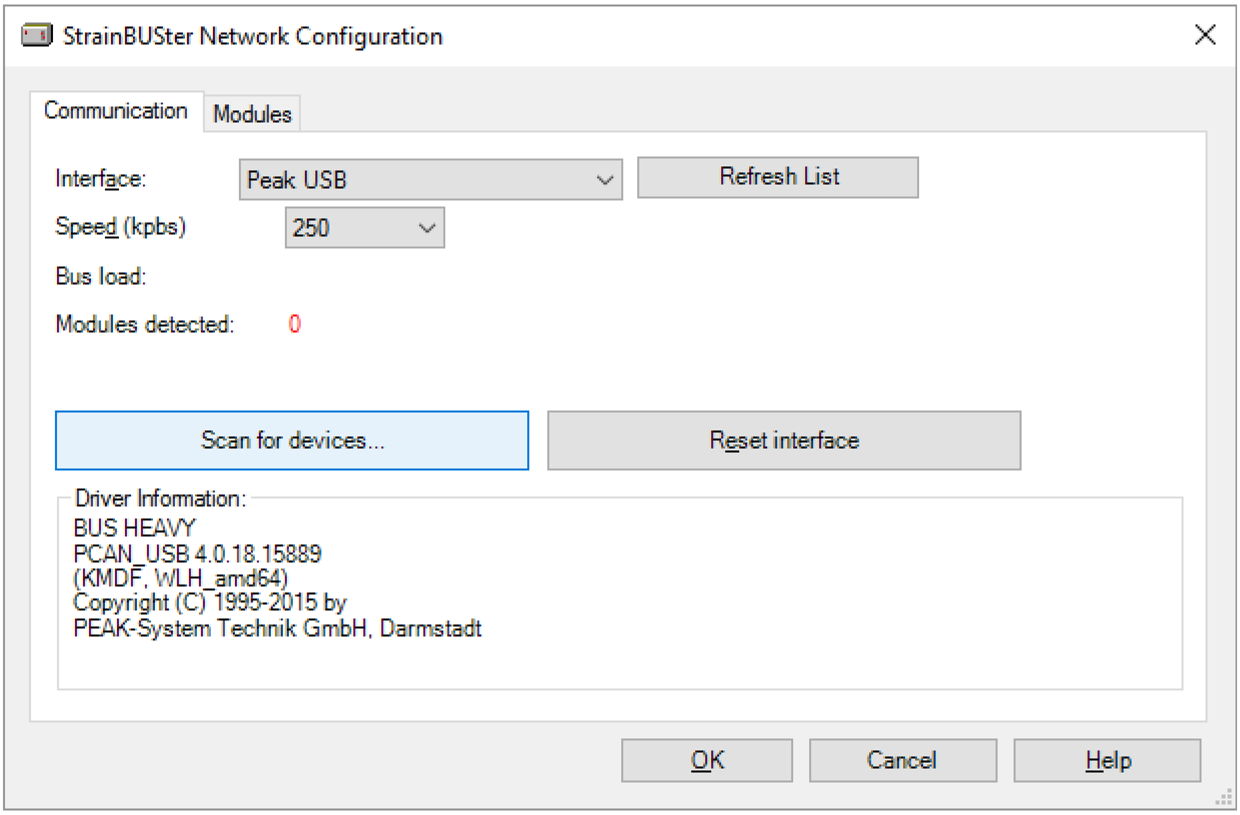
Adding a StrainBUSter bus

To add a new StrainBUSter bus to your configuration, right-click the 'Hardware Configuration' item in the overview tree and select '**New StrainBUSter Bus'** from the menu.



The StainBUSter Network Configuration dialog appears, where you can configure the device and retrieve its configuration.

StrainBUSter: Communication

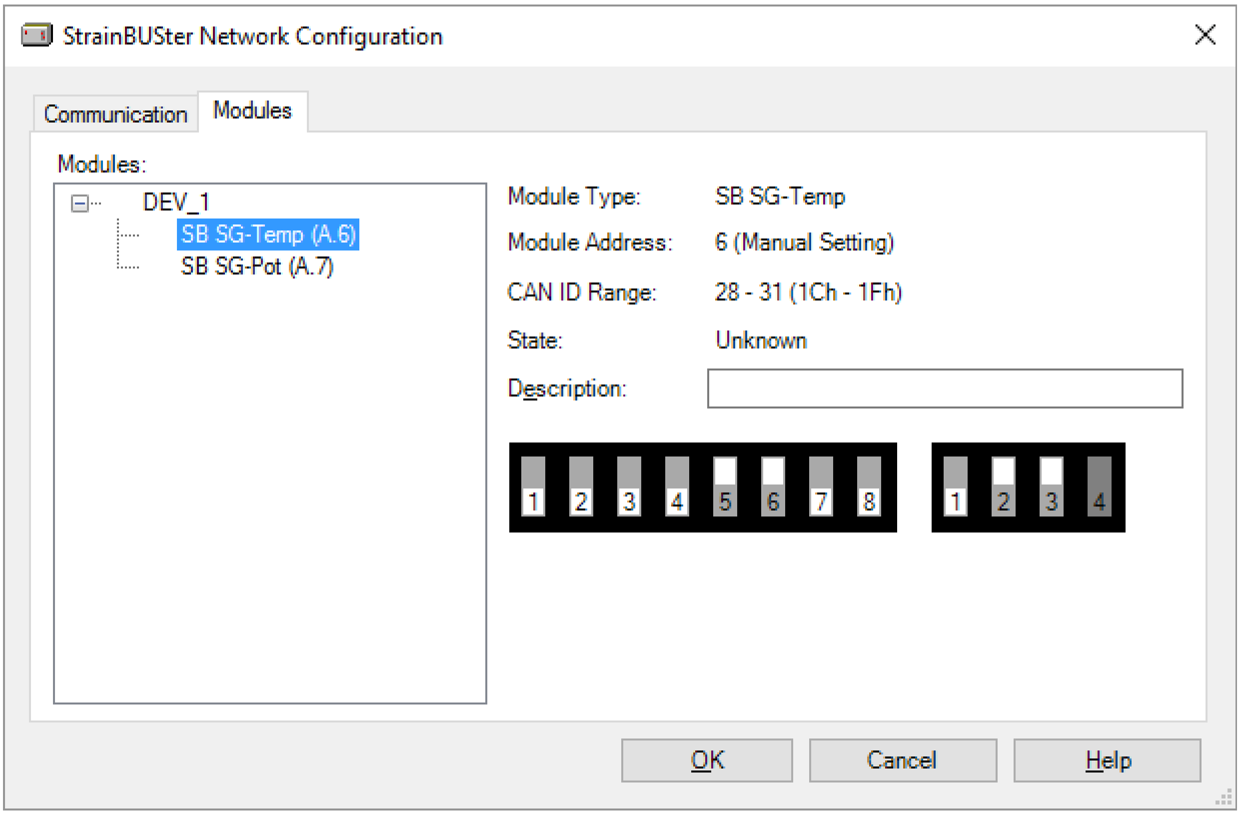


When adding a new StrainBUSter network to the configuration, the first thing to do is to select the communication **interface** from the dropdown list. Devices connected through the CAN bus will use the CAN convertor interface (Peak USB).

After selecting the interface, click the **'scan for devices'** button to retrieve information about the device(s) connected to the interface. If successful, the **'modules detected'** line should show the number of StrainBUSters detected on the CAN bus.

If a communication problem occurs, the **'reset interface'** button will close and re-open the communication with the device, which may help to re-establish communication.

StrainBUSter: Modules



The tree structure on the left shows the detected modules, on the right side of the dialog details about the selected module are shown.

Module Type: Shows which type of StrainBUSter is connected.

Module Address: Shows the address at which the module is set, through the DIP switches. The picture shows the DIP switch settings corresponding to the module address, and selected CAN bus speed.

CAN ID Range: Shows which CAN addresses the module uses.

State: Shows the current communication state.

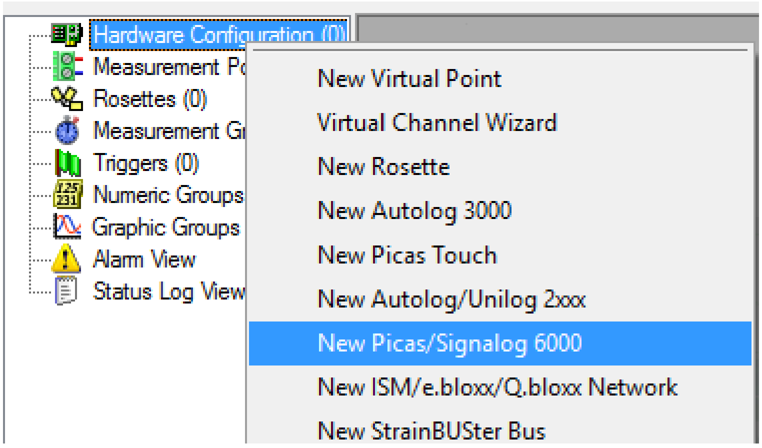
Description: A freely configurable description of the module.

PICAS/Signalog 6000

This chapter describes software functions specific to PICAS and Signalog 6000 devices.

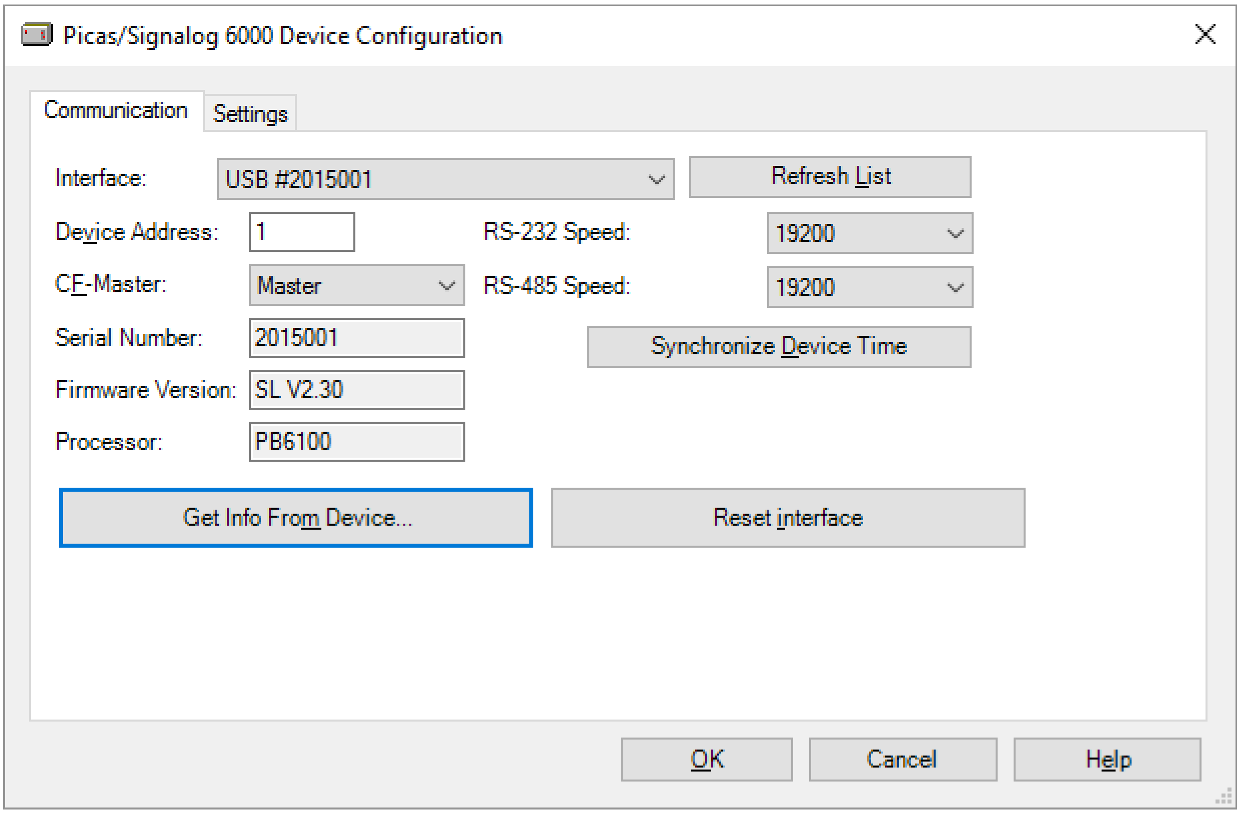
Adding a Picas/Signalog 6000 device

To add a new PICAS or Signalog 6000 device to your configuration, right-click the 'Hardware Configuration' item in the overview tree and select **'New Picas/Signalog 6000'** from the menu.



The Picas/Signalog 6000 Device Configuration dialog appears, where you can configure the device and retrieve its configuration.

Picas: Communication



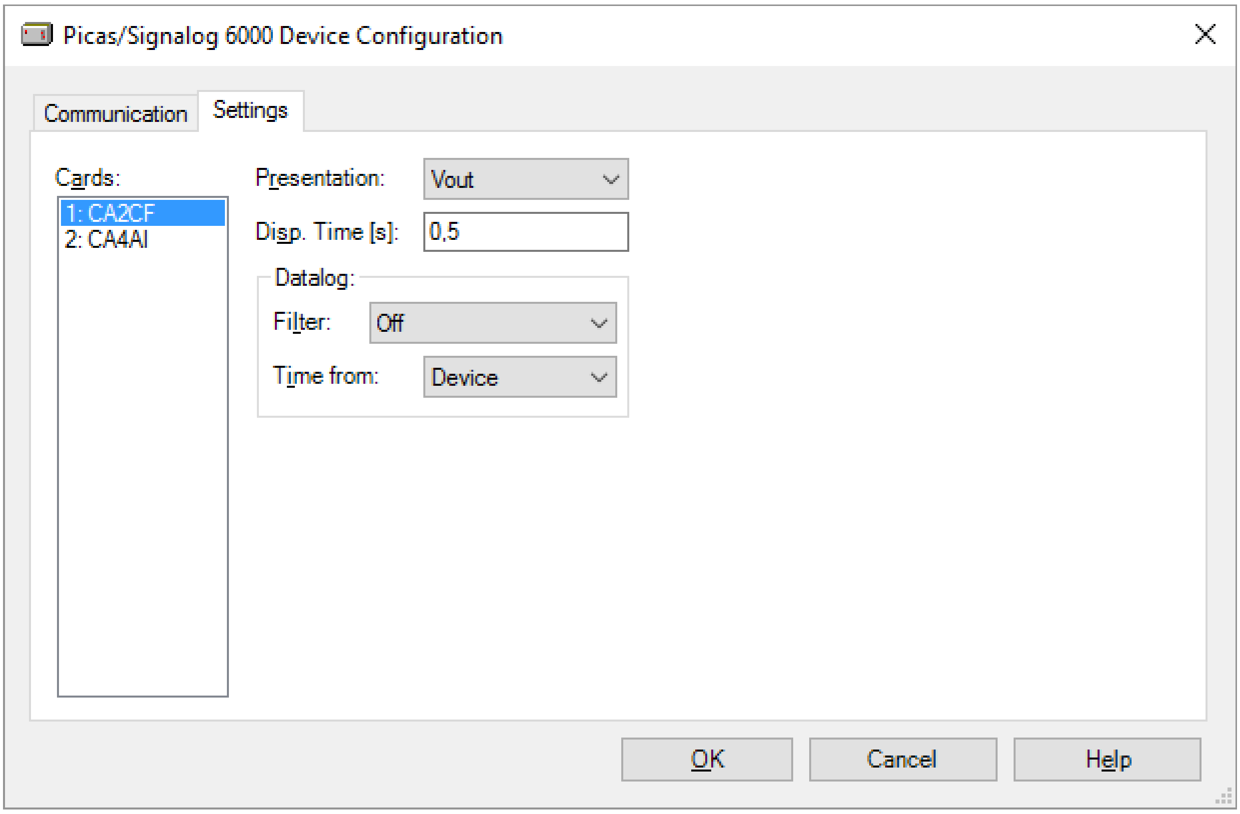
When adding a new PICAS or Signalog 6000 device to the configuration, the first thing to do is to select the communication **interface** from the dropdown list. Devices are connected through USB or serial ports. Should a USB device not be listed, check that is connected properly and switched on, and use the button "**Refresh List**" to try again.

After selecting the interface, click the **'Get Info From Device...'** button to retrieve information about the device connected to the interface. If successful, the fields below the Interface-line should be filled with relevant information like serial number and firmware version.

Use the button "**Synchronize Device Time**" to synchronize the internal clock of the PICAS/Signalog 6000 with the PC clock. This is needed to receive accurate time stamps from the device, and makes sure that measurement data is stored correctly.

If a communication problem occurs, the **'Reset Interface'** button will close and re-open the communication with the device, which may help to re-establish communication.

Picas: Settings



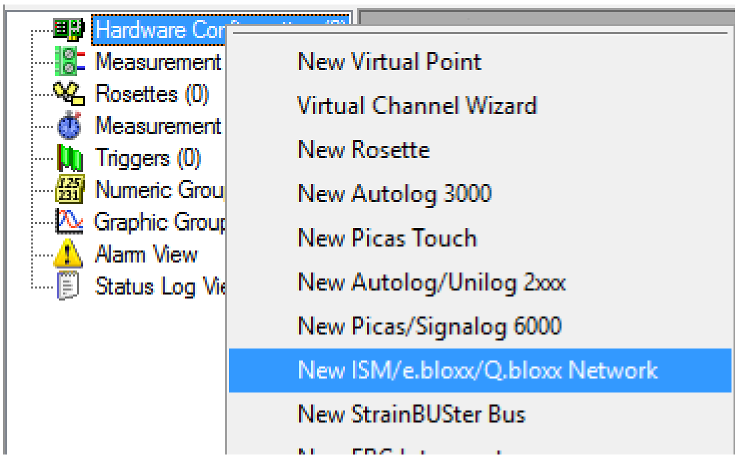
This part of the dialog shows information about device settings. On the left is a list of detected cards, next to that some internal settings that are mostly irrelevant for online use with Autosoft 3000. These settings **do** have impact on local presentation and logging of values on the device.

ISM/e.bloxx/Q.bloxx Network

This chapter describes software functions specific to ISM/e.bloxx/Q.bloxx modules.

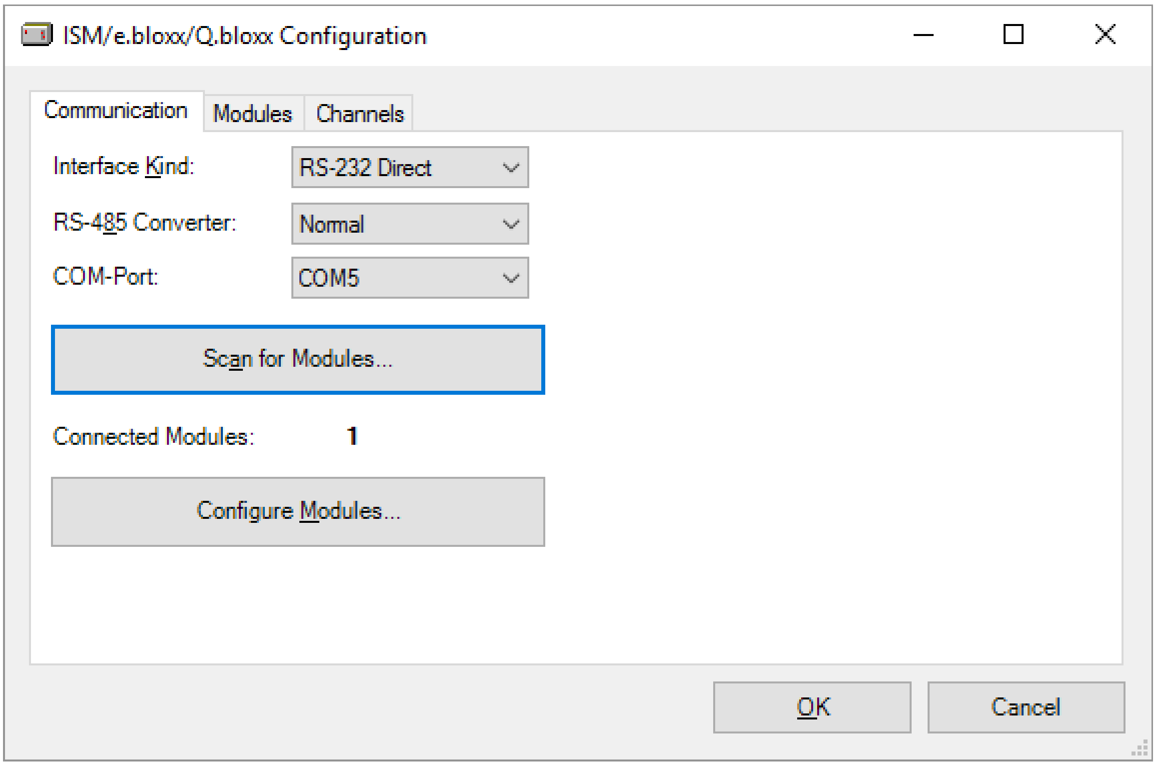
Adding an ISM/e.bloxx/Q.bloxx Network

To add a new ISM/e.bloxx/Q.bloxx network to your configuration, right-click the 'Hardware Configuration' item in the overview tree and select **'New ISM/e.bloxx/Q.bloxx Network'** from the menu.



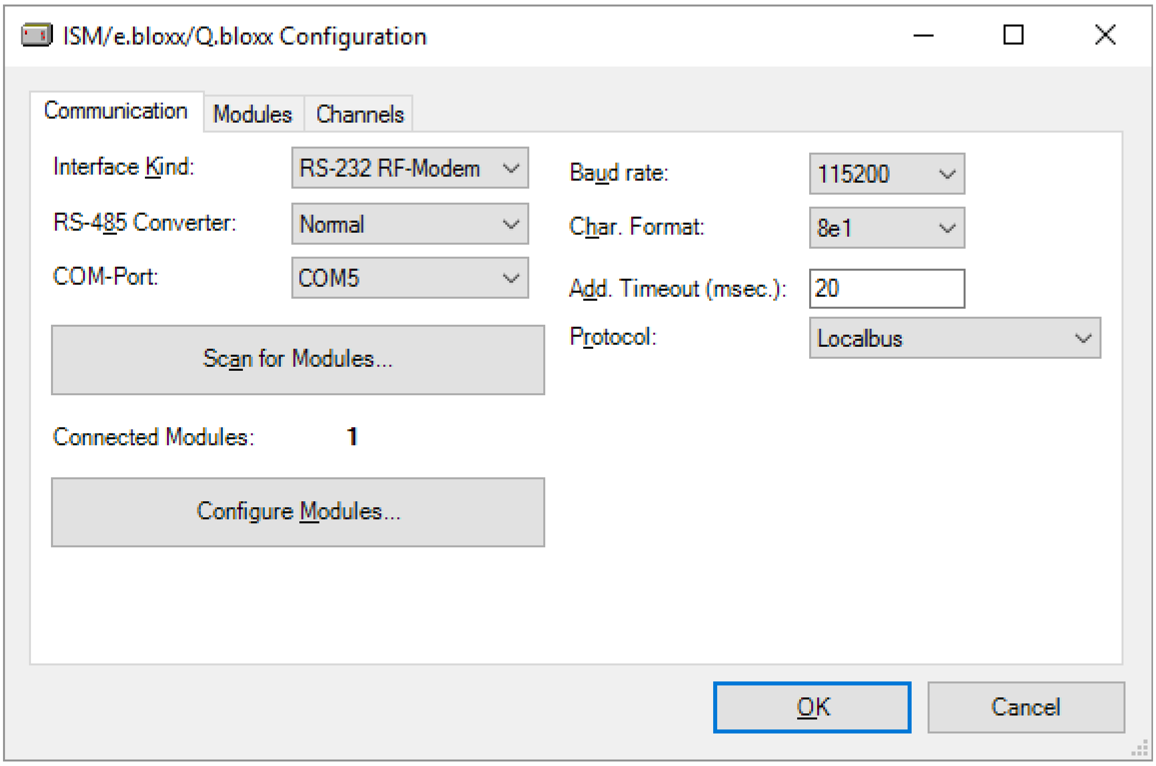
The ISM/e.bloxx/Q.bloxx Configuration dialog appears, where you can configure the modules and retrieve their configuration.

ISM/e.bloxx/Q.bloxx: Communication



When adding a new ISM/e.bloxx/Q.bloxx network to the configuration, the first thing to do is to select the **Interface Kind** from the dropdown list. In most cases this will be "RS-232 Direct" using a "normal" RS-485 Converter. Select the COM-port to which the converter is registered in Windows.

When using a non-standard converter, or communication can not be established using the settings mentioned above, it may be necessary to use the "**RS-232 RF-Modem**" Interface Kind. In that case, the additional settings shown on the right of the dialog must be set manually to match the settings of the modules. The picture below shows the default settings.

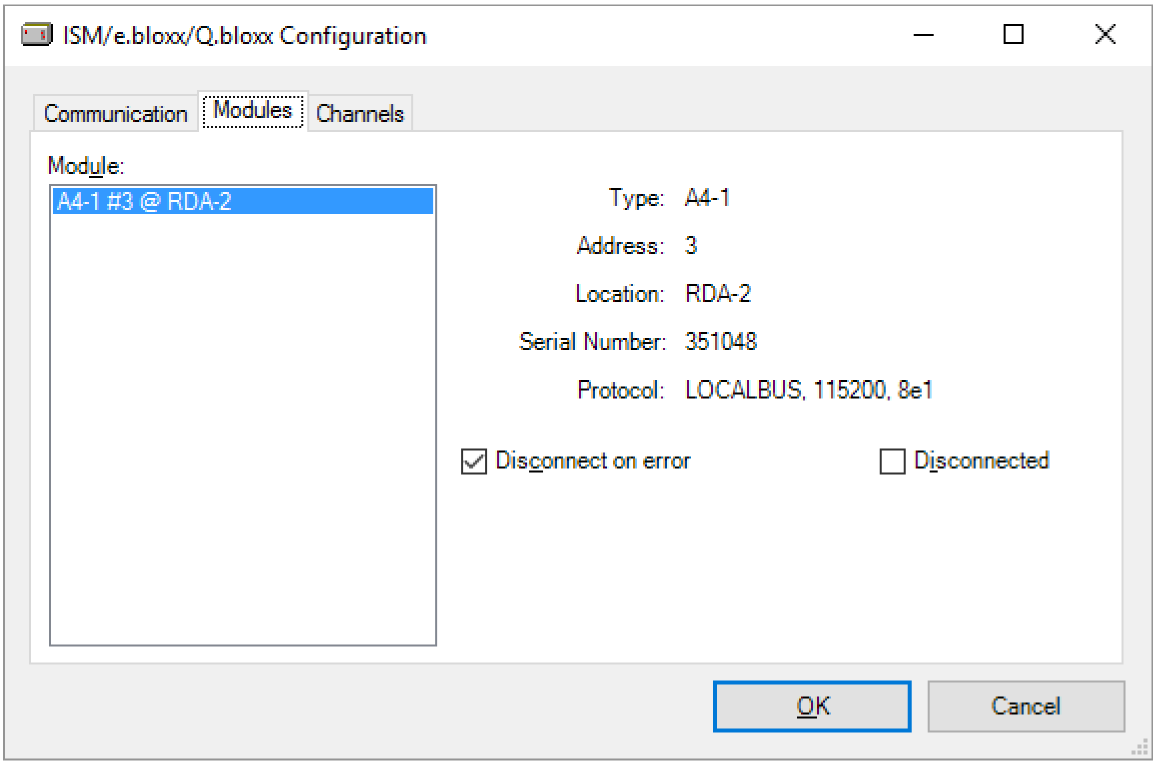


After selecting the interface, click the **'Scan for Modules...'** button to retrieve information about the module(s) connected to the interface. If successful, the **'Connected Modules'** line should show the number of ISM/e.bloxx/Q.bloxx modules detected on the RS-485 network.

All available modules will be detected, but the software will not be able to retrieve measurement data from e.bloxx modules if their communication speed is set higher than 115200.
For maximum communication efficiency, make sure all connected modules use the same communication speed (no higher than 115200) and protocol!

To configure the modules, click on the **"Configure Modules..."** button. This will start the external ICP100 software, which must be installed separately.

ISM/e.bloxx/Q.bloxx: Modules



The tree structure on the left shows the detected modules, on the right side of the dialog details about the selected module are shown.

Type: Shows which type of ISM/e.bloxx/Q.bloxx module is connected.

Address: Shows the slave address at which the module is set. All modules on the bus must have a unique slave address to avoid communication errors.

Location: A freely configurable text, set in the module.

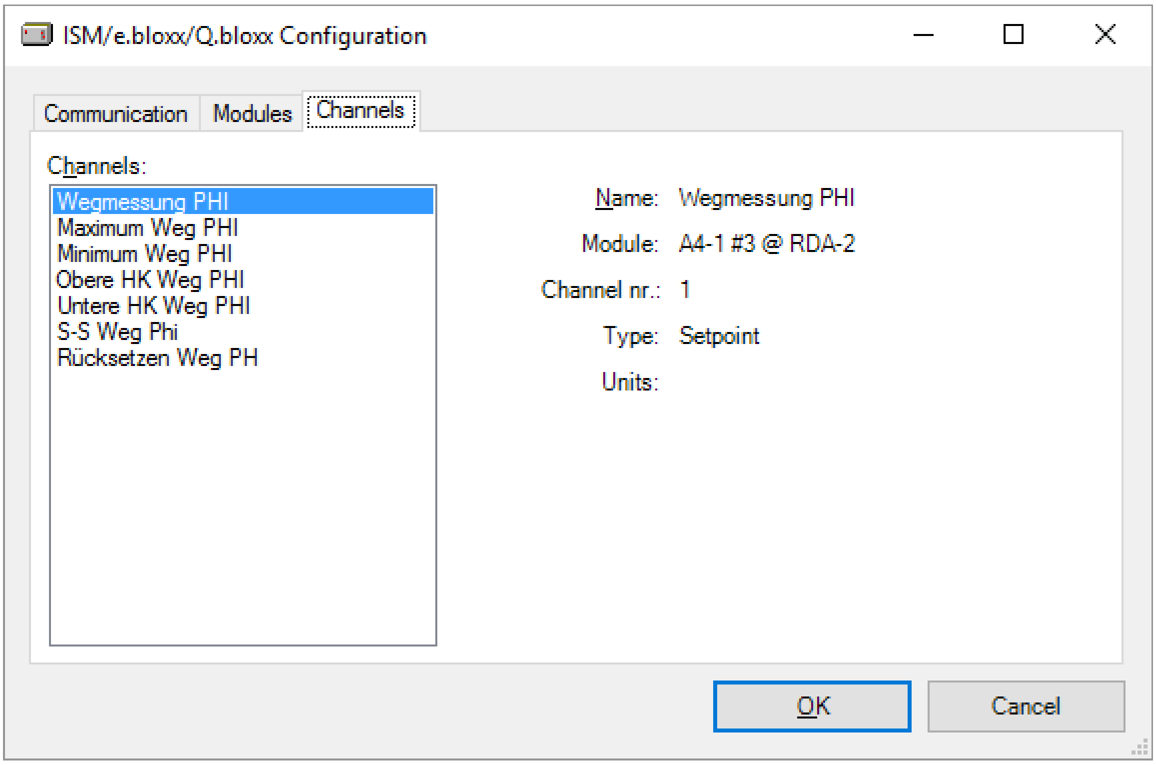
Serial Number: The serial number of the module.

Protocol: The communication protocol used by the module.

Disconnect on error: Leave this box checked (default state) unless you have a specific reason not to. If this box is checked and a communication error occurs, the software will disable this module after two retries. This ensures that the communication with other modules will not be slowed down by timeouts.

Disconnected: A module will be set to disconnected if it does not respond to queries and the setting "Disconnect on error" is active. You can manually uncheck this box to restore the communication, after the cause of the problem is determined.

ISM/e.bloxx/Q.bloxx: Channels



The tree structure on the left shows which channels are available in all of the detected modules, on the right side of the dialog details about the selected module are shown.

Name: Name of the channel

Module: Module to which the channel belongs

Channel nr.: Unique channel number within the module

Type: Channel type, e.g. Analog Input or Setpoint.

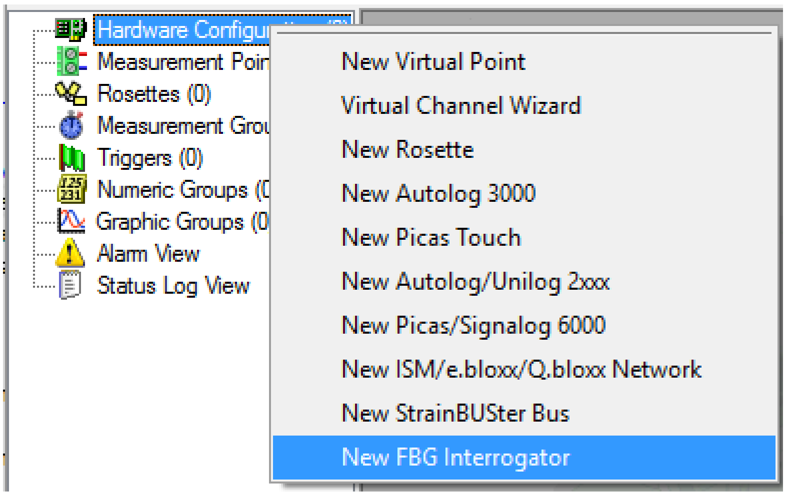
Units: Engineering units for the channel.

FBG Interrogator

This chapter describes software functions specific to FBG Interrogators.

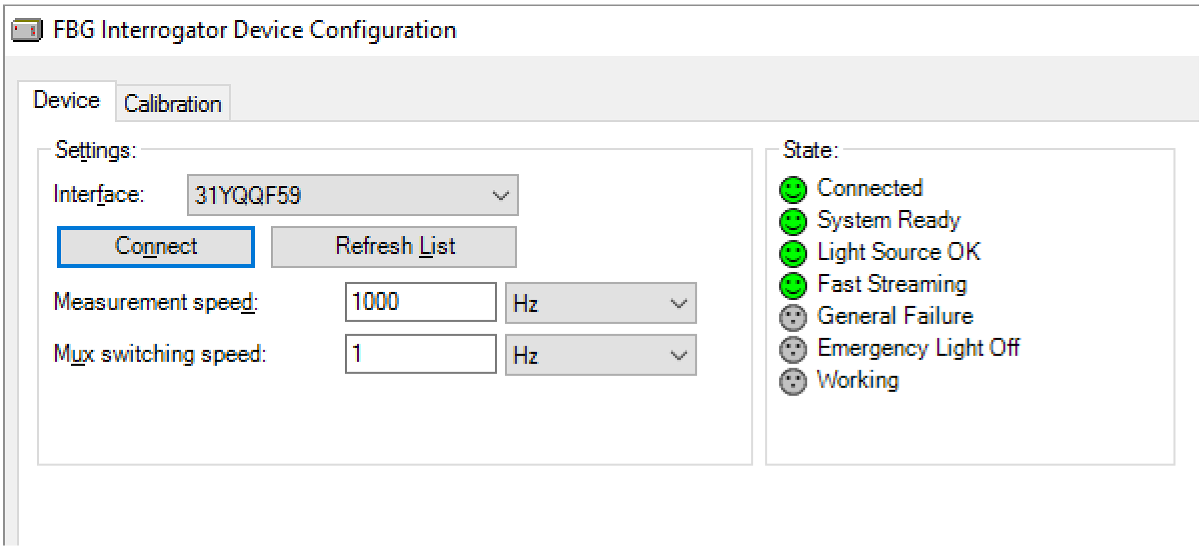
Adding an FBG Interrogator

To add a new FBG Interrogator to your configuration, right-click the 'Hardware Configuration' item in the overview tree and select '**New FBG Interrogator'** from the menu.



The FBG Interrogator Device Configuration dialog appears, where you can configure and calibrate the interrogator.

FBG Interrogator: Device



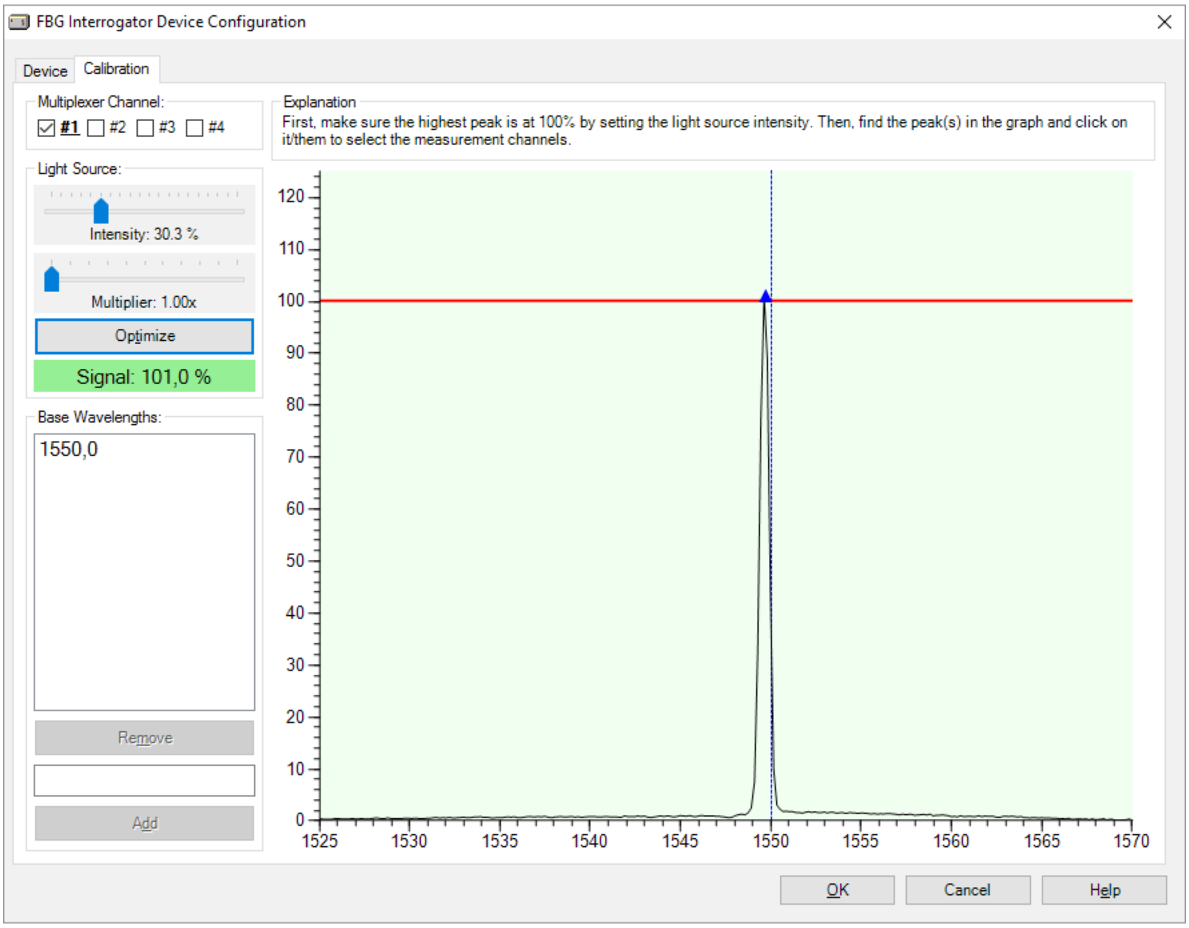
When adding a new FBG Interrogator to the configuration, the first thing to do is to select the communication **Interface** from the drop-down list. Devices are connected through USB and listed by their internal serial number, similar to the one shown in the picture above. Should a USB device not be listed, check that is connected properly and switched on, and use the button "**Refresh List**" to try again.

After selecting the interface, click the **'Connect'** button to establish connection. If successful, the first 4 smileys on the right will turn green. This shows the devices is connected, the light source is working, and measurement data is received.

Use the "**Measurement speed**" to determine the speed at which the device analyzes the light spectrum. The maximum allowable speed is 15 kHz, the default selection is 1 kHz. This setting also defines the upper limit for the measurement speed for channels from this device. If channels are set to measure at a lower rate than this base measurement speed, the measurement value will be averaged over multiple scans to increase stability.

Use the "**Mux switching speed**" setting to determine at which rate the device should switch between multiplexed inputs. This setting is only used if channels from more than one multiplexer input are actually measured. Note that switching between inputs is a physical process that takes some time (20 – 50 msec.), so the switching speed should not be set too high.

FBG Interrogator: Calibration



After connecting an FBG Interrogator, its light source must be calibrated for optimal signal strength. By default, the light source intensity is set to 20%. This is too low in most situations; you should aim for a signal strength (of the strongest signal/peak in the spectrum) of around 100%. Steps to follow:

1. Move the "Intensity" slider to get the signal to around 100%.

2. Should the signal still be too weak when setting the Intensity to maximum, use the Multiplier to further increase the signal. Otherwise, always leave the Multiplier at 1.00x.
   Note: The mulitplier factor actually increases the integration time of the A/D conversion. When the signal strength goes over 100%, there is a risk that the signal will distort, leading to inaccurate measurements.

3. Use the "Optimize" button to automatically optimize the signal strength as a last step.

After calibrating the light source, you should manually identify at which positions in the spectrum the peaks/channels are visible. Click on them to add the channels to the list of base wave lengths. Each channel in the list is shown by a vertical blue dotted line, and its corresponding measurement value by a blue triangle. Make sure the number of channels matches the number of peaks, even when you are not interested in every signal.
Click and drag an existing blue dotted line to move it to a new position, or remove it from the spectrum altogether. Alternatively, you can also use the list of "Base Wavelengths" and corresponding "Add"/"Remove" buttons on the left.

The "Multiplexer Channel" setting at the top left is used to switch between multiplexer inputs, in case you need more than one. The bold/underlined number shows the current selection, all checked channels are actually used in the software as valid multiplexer inputs.

Note: When using multiple mux inputs, always optimize the signal strength based on the single strongest signal over all multiplexer inputs.

Formulas

This appendix lists the formulas used in Autosoft 3000.

Formulas for Pt-100

For Pt-100 channels, the following formulas are used to convert the input signal in Ohms to the output in degrees Celsius.
Input range (Ohms): 18 ... 391
Output range (°C): -200 ... 850:

18 <= Ohms < 100 (-200 °C to 0 °C)	Celsius = -241.801785984373 + Ohms * (2.20988141824334 + Ohms * (2.95384267291581e-3 + Ohms * (-1.06576196498251e-5 + Ohms * 1.93461347561497e-8)))
100 <= Ohms <= 391 (0 °C to 850 °C)	Celsius = -247.662176333752 + Ohms * (2.46319897832119 + Ohms * (-1.52891333213348e-3 + Ohms * (3.40062062617462e-5 + Ohms * (-2.80314519254078e-7 + Ohms * (1.45588665350652e-9 + Ohms * (-4.74313834752117e-12 + Ohms * (9.47314186591109e-15 +Ohms * (-1.05994823878819e-17 + Ohms * 5.09461509269188e-21))))))))

Formulas for Thermocouples

For Thermocouple channels, the following formulas are used to convert the input signal in mV to the output in degrees Celsius.

ITS90 Type B: Platinum-30% Rhodium versus Platinum-6% Rhodium
Input range (mV): 0.291 ... 13.820
Output range (°C): 250 ... 1820

0.291 <= mV < 2.431 (-250 °C to 700 °C)	Celsius = 9.8423321e+1 + mV * (6.9971500e+2 + mV * (-8.4765304e+2 + mV * (1.0052644e+3 + mV * (-8.3345952e+2 + mV * (4.5508542e+2 + mV * (-1.5523037e+2 + mV * (2.9886750e+1 + mV * (-2.4742860))))))))
2.431 <= mV <= 13.820 (700 °C to 1820 °C)	Celsius = 213.15071 + mV * (2.8510504e+2 + mV * (-5.2742887e+1 + mV * (9.9160804 + mV * (-1.2965303 + mV * (1.1195870e-1 + mV * (-6.0625199e-3 + mV * (1.8661696e-4 + mV * (-2.4878585e-6))))))))

CJC correction added to input value for Type B (0 °C ... 630 °C):

Correction = CJCCelsius * (-2.4650818346e-4 + CJCCelsius * (5.9040421171e-6 + CJCCelsius * (-1.3257931636e-9 + CJCCelsius * (1.5668291901e-12 + CJCCelsius * (-1.6944529240e-15 + CJCCelsius * (6.2990347094e-19))))))

ITS90 Type E: Nickel-Chromium versus Copper-Nickel
Input range (mV): -8.825 ... 76.373
Output range (°C): -200 ... 1000

-8.825 <= mV < 0 (-200	Celsius = mV * (1.6977288e+1 + mV * (-4.3514970e-1 + mV *

°C to 0 °C)	(-1.5859697e-1 + mV * (-9.2502871e-2 + mV * (-2.6084314e-2 + mV * (-4.1360199e-3 + mV * (-3.4034030e-4 + mV * (-1.1564890e-5))))))))
0 <= mV <= 76.373 (0 °C to 1000 °C)	Celsius = mV * (1.7057035e+1 + mV * (-2.3301759e-1 + mV * (6.5435585e-3 + mV * (-7.3562749e-5 + mV * (-1.7896001e-6 + mV * (8.4036165e-8 + mV * (-1.3735879e-9 + mV * (1.0629823e-11 + mV * (-3.2447087e-14)))))))))

CJC correction added to input value for Type E (0 °C ... 1000 °C):

Correction = CJCCelsius * (5.8665508710e-2 + CJCCelsius * (4.5032275582e-5 + CJCCelsius * (2.8908407212e-8 + CJCCelsius * (-3.3056896652e-10 + CJCCelsius * (6.5024403270e-13 + CJCCelsius * (-1.9197495504e-16 + CJCCelsius * (-1.2536600497e-18 + CJCCelsius * (2.1489217569e-21 + CJCCelsius * (-1.4388041782e-24 + CJCCelsius * 3.5960899481e-28)))))))))

ITS90 Type J: Iron versus Copper-Nickel
Input range (mV): -8.095 ... 69.553
Output range (°C): -210 ... 1200

-8.095 <= mV < 0 (-210 °C to 0 °C)	Celsius = mV * (19.528268 + mV * (-1.2286185 + mV * (-1.0752178 + mV * (-5.9086933e-1 + mV * (-1.7256713e-1 + mV * (-2.8131513e-2 + mV * (-2.3963370e-3 + mV * -8.3823321e-5)))))))
0 <= mV <= 42.919 (0 °C to 760 °C)	Celsius = mV * (19.78425 + mV * (-2.001204e-1 + mV * (1.036969e-2 + mV * (-2.549687e-4 + mV * (3.585153e-6 + mV * (-5.344285e-8 + mV * 5.09989e-10))))))
42.919 <= mV <= 69.553 (760 °C to 1200 °C)	Celsius = -3.113581e3 + mV * (3.00543684e+2 + mV * (-9.94773230 + mV * (1.7027663e-1 + mV * (-1.43033468e-3 + mV *4.73886084e-6))))

CJC correction added to input value for Type J (0 °C ... 70 °C):

Correction = -3.42189283286958e-5 + CJCCelsius * (50.3917670948465e-3 + CJCCelsius * (2.97699827978468e-5 + CJCCelsius * -6.92590934405273e-8))

ITS90 Type K: Nickel-Chromium versus Nickel-Aluminum
Input range (mV): -5.891 ... 54.886
Output range (°C): -200 ... 1372

-5.891 <= mV < 0 (-200 °C to 0 °C)	Celsius = mV * (2.5173462e+1 + mV * (-1.1662878 + mV * (-1.0833638 + mV * (-8.9773540e-1 + mV * (-3.7342377e-1 + mV * (-8.6632643e-2 + mV * (-1.0450598e-2 + mV * -5.1920577e-4)))))))
0 <= mV <= 20.644 (0 °C to 500 °C)	Celsius = mV * (2.508355e+1 + mV * (7.860106e-2 + mV * (-2.503131e-1 + mV * (8.315270e-2 + mV * (-1.228034e-2 + mV * (9.804036e-4 + mV * (-4.413030e-5 + mV * (1.057734e-6 + mV * -1.052755e-8))))))))
20.644 <= mV <= 54.886 (500 °C to 1372 °C)	Celsius = -1.318058e+2 + mV * (4.830222e+1 + mV * (-1.646031 + mV * (5.464731e-2 + mV * (-9.650715e-4 + mV * 8.802193e-6 + mV * -3.110810e-8)))))

CJC correction added to input value for Type K (0 °C ... 70 °C):

Correction = -6.11053935922907e-6 + CJCCelsius * (39.4533818469946e-3 + CJCCelsius * (2.40423569466323e-5 + CJCCelsius * (-6.51590136125687e-8 + CJCCelsius * -2.47299023308676e-10)))

ITS90 Type N: Nickel-Chromium-Silicon versus Nickel-Silicon-Magnesium
Input range (mV): -3.990 ... 47.513
Output range (°C): -200 ... 1300

-3.990 <= mV < 0 (-200 °C to 0 °C)	Celsius = mV * (3.8436847e+1 + mV * (1.1010485 + mV * (5.2229312 + mV * (7.2060525 + mV * (5.8488586 + mV * (2.7754916 + mV * (7.7075166e-1 + mV * (1.1582665e-1 + mV * (7.3138868e-3)))))))))
0 <= mV <= 20.613 (0 °C to 600 °C)	Celsius = mV * (3.86896e+1 + mV * (-1.08267 + mV * (4.70205e-2 + mV * (-2.12169e-6 + mV * (-1.17272e-4 + mV * (5.39280e-6 + mV * (-7.98156e-8)))))))
20.613 <= mV <= 47.513 (600 °C to 1300 °C)	Celsius = 1.972485e+1 + mV * (3.300943e+1 + mV * (-3.915159e-1 + mV * (9.855391e-3 + mV * (-1.274371e-4 + (mV * 7.767022e-7)))))

CJC correction added to input value for Type N (0 °C ... 1300 °C):

Correction = CJCCelsius * (2.592939460e-2 + CJCCelsius * (1.5710141880e-5 + CJCCelsius * (4.3825627237e-8 + CJCCelsius * (-2.5261169794e-10 + CJCCelsius * (6.4311819339e-13 + CJCCelsius * (-1.0063471519e-15 + CJCCelsius * (9.9745338992e-19 + CJCCelsius * (-6.0863245607e-22 + CJCCelsius * (2.0849229339e-25 + CJCCelsius * -3.0682196151e-29)))))))))

ITS90 Type R: Platinum-13% Rhodium versus Platinum
Input range (mV): -0.226 ... 21.103
Output range (°C): -50 ... 1768.1

-0.226 <= mV < 1.923 (-50 °C to 250 °C)	Celsius = mV * (1.8891380e+2 + mV * (-9.3835290e+1 + mV * (1.3068619e+2 + mV * (-2.2703580e+2 + mV * (3.5145659e+2 + mV * (-3.8953900e+2 + mV * 2.8239471e+2 + mV * (-1.2607281e+2 + mV * (3.1353611e+1 + mV * (-3.3187769)))))))))
1.923 <= mV <= 11.361 (250 °C to 1064 °C)	Celsius = 1.334584505e+1 + mV * (1.472644573e+2 + mV * (-1.844024844e+1 + mV * (4.031129726 + mV * (-6.249428360e-1 + mV * (6.468412046e-2 + mV * (-4.458750426e-3 + mV * (1.994710149e-4 + mV * (-5.313401790e-6 + mV * (6.481976217e-8)))))))))
11.361 <= mV <= 19.739 (1064 °C to 1664.5 °C)	Celsius = -8.199599416e+1 + mV * (1.553962042e+2 + mV * (-8.342197663 + mV * (4.279433549e-1 + mV * (-1.191577910e-2 + mV * (1.492290091e-4)))))
19.739 <= mV <= 21.103 (1664.5 °C to 1768.1 °C)	Celsius = 3.406177836e+4 + mV * (-7.023729171e+3 + mV * (5.582903813e+2 + mV * (-1.952394635e+1 + mV * (2.560740231e-1))))

CJC correction added to input value for Type R (-50 °C ... 1064 °C):

Correction = CJCCelsius * (5.28961729765e-3 + CJCCelsius * (1.39166589782e-5 + CJCCelsius * (-2.38855693017e-8 + CJCCelsius * (3.5691601063e-11 + CJCCelsius * (-4.62347666298e-14 +

CJCCelsius *(5.00777441034e-17 + CJCCelsius * (-3.73105886191e-20 + CJCCelsius *(1.57716482367e-23 + CJCCelsius * (-2.81038625251e-27)))))))))

ITS90 Type S: Platinum-10% Rhodium versus Platinum
Input range (mV): -0.235 ... 18.693
Output range (°C): -50 ... 1768.1

-0.235 <= mV < 1.874 (-50 °C to 250 °C)	Celsius = mV * (1.84949460e+2 + mV * (-8.00504062e+1 + mV * (1.02237430e+2 + mV * (-1.52248592e+2 + mV * (1.88821343e+2 + mV * (-1.59085941e+2 + mV * (8.23027880e+1 + mV * (-2.34181944e+1 + mV * 2.79786260))))))))
1.874 <= mV <= 11.950 (250 °C to 1200 °C)	Celsius = 1.291507177e+1 + mV * (1.466298863e+2 + mV * (-1.534713402e+1 + mV * (3.145945973 + mV * (-4.163257839e-1 + mV * (3.187963771e-2 + mV * (-1.291637500e-3 + mV * (2.183475087e-5 + mV * (-1.447379511e-7 + mV * 8.211272125e-9))))))))
11.950 <= mV <= 17.536 (1200 °C to 1664.5 °C)	Celsius = -8.087801117e+1 + mV * (1.621573104e+2 + mV * (-8.536869453 + mV * (4.719686976e-1 + mV * (-1.441693666e-2 + mV * 2.081618890e-4))))
17.536 <= mV <= 18.693 (1664.5 °C to 1768.1 °C)	Celsius = 5.333875126e+4 + mV * (-1.235892298e+4 + mV * (1.092657613e+3 + mV * (-4.265693686e+1 + mV * 6.247205420e-1)))

CJC correction added to input value for Type S (0 °C ... 70 °C):

Correction = -8.61838794792875e-6 + CJCCelsius * (5.40579212101081e-3 + CJCCelsius * (1.24166233625116e-5 + CJCCelsius * -1.9163457510135e-8))

ITS90 Type T: Copper versus Copper-Nickel
Input range (mV): -5.603 ... 20.872
Output range (°C): -200 ... 400

-5.603 <= mV < 0 (-200 °C to 0 °C)	Celsius = mV * (2.5949192e+1 + mV *(-2.1316967e-1 + mV * (7.9018692e-1 + mV * (4.2527777e-1 + mV * (1.3304473e-1 + mV * (2.0241446e-2 + mV * 1.2668171e-3))))))
0 <= mV <= 20.872 (0 °C to 400 °C)	Celsius = mV * (2.592800e+1 + mV * (-7.602961e-1 + mV * (4.637791e-2 + mV * (-2.165394e-3 + mV * (6.048144e-5 + mV * -7.293422e-7)))))

CJC correction added to input value for Type T (0 °C ... 70 °C):

Correction = 2.18031481725975e-6 + CJCCelsius * (38.746564651674e-3 + CJCCelsius * (3.35256077037807e-5 + CJCCelsius * (1.9229303001792e-7 + CJCCelsius * (-1.79865206510148e-9 + CJCCelsius * 5.70614795196761e-12))))

Formulas for Strain Gauges

For Carrier Frequency straingauge measurements as used in Autolog/Unilog 2xxx, the non linearity of the Wheatstone Bridge makes a correction necessary when measuring 1/4 bridges. The correction formula used in Autosoft NT is:

CorrectedValue = 1e6 · (1 - sqrt(1 - 2 · MeasuredValue / 1e6))

To obtain the 'Stress' value in N/mm² from the 'Strain' value in µm/m, the following formula is used:

Stress = E-modulus · Strain / 1e6

Formulas for 0°/90° Rosette

Variables used:

EPSa' = strain input at 0°
EPSb' = strain input at 90°
V0 = Poisson (ref.)
Vs = Poisson (spec.)
Emod = E-modulus
Kt = Transverse sensitivity

(Note: Kt is specified in % in the rosette configuration dialog, the value of Kt used in the formulas below is therefore 1/100 of this value. For example: Kt specified in dialog = 2, then Kt used in formulas = 0.02)

Calculation of strain after correction for transverse sensitivity:

$EPSa = ((1 - V0 \cdot Kt) / (1 - Kt^2)) \cdot (EPSa' - Kt \cdot EPSb')$
$EPSb = ((1 - V0 \cdot Kt) / (1 - Kt^2)) \cdot (EPSb' - Kt \cdot EPSa')$

EPSa is called 'Strain A' or 'EPS_A' in the software.
EPSb is called 'Strain B' or 'EPS_B' in the software.

Calculation of stress:

$SIGx = (Emod / (1 - Vs^2)) \cdot (EPSa + Vs \cdot EPSb)$
$SIGy = (Emod / (1 - Vs^2)) \cdot (EPSb + Vs \cdot EPSa)$

SIGx is called 'Stress 0°' or 'SIG_X' in the software.
SIGy is called 'Stress 90°' or 'SIG_Y' in the software.

Formulas for 0°/45°/90° Rosettes

Variables used:

EPSa' = strain input at 0°
EPSb' = strain input at 45°
EPSc' = strain input at 90°
V0 = Poisson (ref.)
Vs = Poisson (spec.)
Emod = E-modulus
Kt = Transverse sensitivity

(Note: Kt is specified in % in the rosette configuration dialog, the value of Kt used in the formulas below is therefore 1/100 of this value. For example: Kt specified in dialog = 2, then Kt used in formulas = 0.02)

Calculation of strain after correction for transverse sensitivity:

$EPSa = ((1 - V0 \cdot Kt) / (1 - Kt^2)) \cdot (EPSa' - Kt \cdot EPSc')$
$EPSb = ((1 - V0 \cdot Kt) / (1 - Kt^2)) \cdot (EPSb' - Kt \cdot (EPSa' + EPSc' - EPSb'))$
$EPSc = ((1 - V0 \cdot Kt) / (1 - Kt^2)) \cdot (EPSc' - Kt \cdot EPSa')$

EPSa is called 'Strain A' or 'EPS_A' in the software.
EPSb is called 'Strain B' or 'EPS_B' in the software.

EPSc is called 'Strain C' or 'EPS_C' in the software.

Calculation of the principal strains:

$$EPSmaj = (EPSa + EPSc) / 2 + \mathrm{sqrt}(((EPSa - EPSb)^2 + (EPSc - EPSb)^2) / 2)$$
$$EPSmin = (EPSa + EPSc) / 2 - \mathrm{sqrt}(((EPSa - EPSb)^2 + (EPSc - EPSb)^2) / 2)$$

EPSmaj is called 'Major Strain' or 'EPS_MAJ' in the software.
EPSmin is called 'Minor Strain' or 'EPS_MIN' in the software.

Calculation of angle:

This is the angle between the major strain (or stress) and the first strain filament (0°), expressed as a number between -90 and +90 degrees.

$$ALPHA = \arctan((2 \cdot EPSb - EPSa - EPSc) / (EPSa - EPSc)) / 2$$

ALPHA is called 'Angle' or 'ALPHA' in the software.

Calculation of the principal stresses:

$$SIGmaj = (Emod / (1 - Vs^2)) \cdot (EPSmaj + Vs \cdot EPSmin)$$
$$SIGmin = (Emod / (1 - Vs^2)) \cdot (EPSmin + Vs \cdot EPSmaj)$$

SIGmaj is called 'Major Stress' or 'SIG_MAJ' in the software.
SIGmin is called 'Minor Stress' or 'SIG_MIN' in the software.

Calculation of the stress:

These calculations determine the stress in the direction of the first strain filament (0°) and in the direction perpendicular to the first strain filament (90°), as well as the shear stress.

$$SIGx = ((SIGmaj + SIGmin) + (SIGmaj - SIGmin) \cdot \cos(2 \cdot ALPHA)) / 2$$
$$SIGy = ((SIGmaj + SIGmin) - (SIGmaj - SIGmin) \cdot \cos(2 \cdot ALPHA)) / 2$$
$$TAUxy = (SIGmaj - SIGmin) \cdot \sin(2 \cdot ALPHA) / 2$$

SIGx is called 'Stress 0°' or 'SIG_X' in the software.
SIGy is called 'Stress 0°' or 'SIG_Y' in the software.
TAUxy is called 'Shear Stress' or 'TAU_XY' in the software.

Calculation of the equivalent stress:

The equivalent stress for the principal stresses (von Mises criterion):

$$SIGe = \mathrm{sqrt}(SIGmaj^2 + SIGmin^2 - SIGmaj \cdot SIGmin)$$

Formulas for 0°/60°/120° Rosettes

Variables used:

EPSa' = strain input at 0°
EPSb' = strain input at 60°
EPSc' = strain input at 120°
V0 = Poisson (ref.)
Vs = Poisson (spec.)
Emod = E-modulus
Kt = Transverse sensitivity

(Note: Kt is specified in % in the rosette configuration dialog, the value of Kt used in the formulas below is therefore 1/100 of this value. For example: Kt specified in dialog = 2, then Kt used in formulas = 0.02)

Calculation of strain after correction for transverse sensitivity:

$$EPSa = ((1 - V0 \cdot Kt) / (1 - Kt^2)) \cdot ((1 + Kt / 3) \cdot EPSa' - (2 / 3) \cdot Kt \cdot (EPSb' + EPSc'))$$
$$EPSb = ((1 - V0 \cdot Kt) / (1 - Kt^2)) \cdot ((1 + Kt / 3) \cdot EPSb' - (2 / 3) \cdot Kt \cdot (EPSa' + EPSc'))$$
$$EPSc = ((1 - V0 \cdot Kt) / (1 - Kt^2)) \; ((1 + Kt / 3) \cdot EPSc' - (2 / 3) \cdot Kt \cdot (EPSa' + EPSb'))$$

EPSa is called 'Strain A' or 'EPS_A' in the software.
EPSb is called 'Strain B' or 'EPS_B' in the software.
EPSc is called 'Strain C' or 'EPS_C' in the software.

Calculation of the principal strains:

$$EPSmaj = (EPSa + EPSb + EPSc) / 3 + sqrt(((2\, EPSa - EPSb - EPSc)^2 + 3 \cdot (EPSb - EPSc)^2) / 9)$$
$$EPSmin = (EPSa + EPSb + EPSc) / 3 - sqrt(((2\, EPSa - EPSb - EPSc)^2 + 3 \cdot (EPSb - EPSc)^2) / 9)$$

EPSmaj is called 'Major Strain' or 'EPS_MAJ' in the software.
EPSmin is called 'Minor Strain' or 'EPS_MIN' in the software.

Calculation of angle:

This is the angle between the major strain (or stress) and the first strain filament (0°), expressed as a number between -90 and +90 degrees.

$$ALPHA = arctan((sqrt(3) \cdot (EPSb - EPSc)) / (2\, EPSa - EPSb - EPSc)) / 2$$

ALPHA is called 'Angle' or 'ALPHA' in the software.

Calculation of the principal stresses:

$$SIGmaj = (Emod / (1 - Vs^2)) \cdot (EPSmaj + Vs \cdot EPSmin)$$
$$SIGmin = (Emod / (1 - Vs^2)) \cdot (EPSmin + Vs \cdot EPSmaj)$$

SIGmaj is called 'Major Stress' or 'SIG_MAJ' in the software.
SIGmin is called 'Minor Stress' or 'SIG_MIN' in the software.

Calculation of the stress:

These calculations determine the stress in the direction of the first strain filament (0°) and in the direction perpendicular to the first strain filament (90°), as well as the shear stress.

$$SIGx = ((SIGmaj + SIGmin) + (SIGmaj - SIGmin) \cdot cos(2 \cdot ALPHA)) / 2$$
$$SIGy = ((SIGmaj + SIGmin) - (SIGmaj - SIGmin) \cdot cos(2 \cdot ALPHA)) / 2$$
$$TAUxy = (SIGmaj - SIGmin) \cdot sin(2 \cdot ALPHA) / 2$$

SIGx is called 'Stress 0°' or 'SIG_X' in the software.
SIGy is called 'Stress 0°' or 'SIG_Y' in the software.
TAUxy is called 'Shear Stress' or 'TAU_XY' in the software.

Calculation of the equivalent stress:

The equivalent stress for the principal stresses (von Mises criterion):

$$SIGe = sqrt(SIGmaj^2 + SIGmin^2 - SIGmaj \cdot SIGmin)$$