

Signasoft 6000 User Manual

Version number	2.2.1
Release date	23-11-2009
Author	ACN

Copyright notice:

The SignaSoft 6000 software package as distributed by Peekel Instruments is property of:

PEEKEL INSTRUMENTS B.V.
Industrieweg 161
NL 3044 AS Rotterdam
The Netherlands

which company possesses all rights to distribute and supply SignaSoft 6000.

It is not allowed to copy, distribute or sell the programs or any material originating from it such as listings, disks, diskettes or any other medium to copy the program or part thereof to any party, company, organisation, institution or person without written permission of the proprietor.

Any violation of this shall be brought to lawsuit and prosecution. In accepting and receiving this document the recipient automatically agrees to act conform the requirements stated above.



PEEKEL INSTRUMENTS B.V.
INDUSTRIEWEG 161
3044 AS ROTTERDAM
TEL: (010)-415 27 22
FAX: (010)-437 68 26
EMAIL: sales@peekel.nl

PEEKEL INSTRUMENTS GMBH
BERGMANNSTRASSE 43
44809 BOCHUM
TEL: 0234/904 1603
FAX: 0234/904 1605
EMAIL: Peekel@t-online.de

1. Introduction	4
2. Configuration Selection Dialog	4
3. File Menu Commands	5
3.1 New Command	5
3.2 Open Command	6
3.3 Close Command	6
3.4 Save Command	6
3.5 Save As Command	6
3.6 Export As Text Command	6
3.7 Language Command	7
3.8 Recent File Command	7
3.9 Exit Command	7
4. Edit Menu Commands	7
4.1 Cut Command	7
4.2 Copy Command	7
4.3 Paste Command	8
5. Insert Menu Commands	8
5.1 Add Picas/Signalog Device	8
5.1.1 General Settings	9
5.1.2 Analog Outputs	10
5.1.3 CA2CF General Settings	10
5.1.4 CA2CF Sensor Settings	11
5.1.5 CA2CF Strain Settings	11
5.1.6 CA2CF Range Settings	12
5.1.7 CA2CF Balance Settings	13
5.1.8 CA2CF Trip Settings	13
5.1.9 CA4AI General Settings	14
5.1.10 CA4AI Sensor Settings	15
5.1.11 CA4AI Tare Settings	15
5.1.12 CA4AI Trip Settings	16
5.2 Add StrainBUSTER Network	17
5.2.1 Network Settings	17
5.2.2 CAN Cable Length	18
5.2.3 Channel Settings	18
5.2.4 Scaling	20
5.2.5 Lead Wire Resistance	21
5.2.6 Trip Settings	21
5.3 Add Autolog 3000 Device	23
5.3.1 Network Configuration	23
5.3.2 CAN Bus Cable Length	24
5.3.3 Cards Configuration	24
5.3.4 Replacing a Card	26
5.3.5 Channels Configuration	26
5.3.6 Channels: Bridge Sensor	27
5.3.7 Channels: Thermocouple Sensor	28
5.3.8 Channels: Measurement	29
5.3.9 Channels: Balance/Tare	30
5.3.10 Channels: Scaling	31
5.3.11 Channels: Shunt	32
5.3.12 Trips Configuration	33
5.4 Add ISM/e.bloxx Network	35
5.4.1 Communication Settings	35
5.4.2 Module Settings	35
5.4.3 Channel Settings	36
5.4.4 Trips Settings	37
5.5 Add e.gate/e.xact	38
5.5.1 Communication Settings	39
5.5.2 Channel Settings	40
5.5.3 Trips Settings	41

5.6 Add Virtual Channel.....	42
5.6.1 Channel Settings Property Page.....	42
5.6.2 Formula Property Page.....	44
5.6.3 Trip Levels Property Page.....	44
5.7 Add Rosette Channel.....	45
5.8 New Graphics Group.....	46
5.8.1 Graphic Button Bar.....	48
5.8.2 Graphic Settings.....	49
5.8.3 Graphic Axes.....	49
5.8.4 Graphic Line Style.....	51
5.8.5 Graphic Colors.....	52
5.8.6 Graphic Font.....	52
5.9 New Numeric Group.....	53
5.9.1 Numeric Settings.....	54
5.9.2 Numeric Presentation.....	54
5.9.3 Numeric Font.....	55
5.9.4 Channel Selection.....	56
5.9.5 Print Function.....	57
6. Measurement Menu Commands.....	57
6.1 Perform Loadstep.....	57
6.2 Logging Group Configuration.....	58
6.2.1 Channel Selection.....	59
6.2.2 Rule Configuration.....	60
6.2.3 Periodic Rule.....	61
6.2.4 Trip-based Rule.....	61
6.2.5 Trip Selection.....	62
6.3 Measurement Export Configuration.....	63
6.4 Clean Archives.....	65
6.4.1 Channel Selection.....	65
7. Device Commands.....	66
7.1 Store Measurement Data.....	67
8. Channel Menu Commands.....	67
9. Graphic Menu Commands.....	67
10. Numeric Menu Commands.....	68
11. Trips Menu Commands.....	68
12. View Menu Commands.....	68
12.1 Device Overview Window.....	69
12.2 Channel Overview Window.....	69
12.3 Bargraph View Window.....	70
12.4 Settings View Window.....	71
12.5 Trips View Window.....	71
12.5.1 Channel Selection.....	72
12.5.2 Print Function.....	72
12.5.3 Trips Settings.....	73
12.5.4 Execute Application.....	73
12.5.5 Trip Font.....	74
12.6 Event View Window.....	75
13. Window Menu Commands.....	76
13.1 Cascade Command.....	76
13.2 Tile Command.....	77
13.3 Window Arrange Icons Command.....	77
13.4 1, 2, ... Command.....	77
14. Help Menu Commands.....	77

1. Introduction

SignaSoft 6000 is a software package designed to configure and control Signalog data acquisition systems and StrainBUSters produced by Peekel Instruments B.V. To run SignaSoft 6000, you need at least Windows 2000 (Service Pack 4). Recommended: Windows XP (Service Pack 2 or higher), Windows Vista or Windows 7.

When the program starts, the Configuration Selection dialog will show. This dialog will allow you to create a new configuration, or load a recent or any other existing configuration.

To measure channels using SignaSoft 6000, first configure one or more measurement devices. After that, create channels to measure from the Device Overview.

After your devices and channels are configured, you can create numerical groups, allowing you to show measurement data on screen in numerical form. Online graphics are supported using graphical groups, which can show up to 16 channels in a single graphical display. A trips group can be used to configure trips and generate alarm messages.

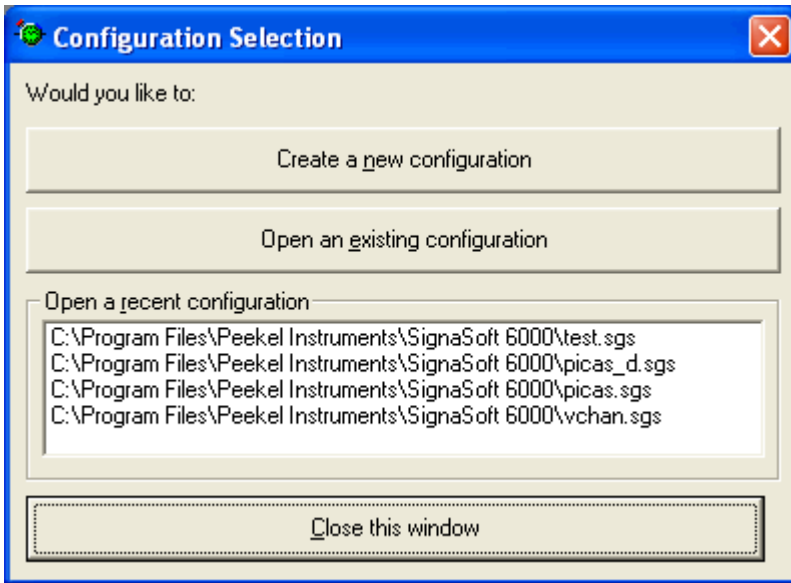
From the channel overview, you can specify a logging interval for each channel. Using these intervals, SignaSoft 6000 can log measurement data to a file.

Throughout the use of the program you can press the F1 key to obtain context sensitive online help.

Commands

- File menu
- Edit menu
- Insert menu
- Measurement menu
- Device menu
- Channel menu
- Graphics menu
- Numerics menu
- Trips menu
- View menu
- Window menu
- Help Menu Commands

2. Configuration Selection Dialog



To create a new configuration, press the 'Create a new configuration' button. You will then be asked to save the configuration. After the configuration is saved, an 'archive' directory for storing measurement data will automatically be created. If, for example, you store your configuration under 'C:\data\config.sgs', the archive directory will be named 'C:\data\config_Archive'.

When you press the 'Open an existing configuration' button, you can browse for the configuration to load.

The 'Open a recent configuration' list shows the four most recent configurations you used. You can double-click on a configuration to load it.

If you close this window without loading a configuration, you can create or load a configuration using the File menu.

3. File Menu Commands

The File menu offers the following commands:


New	Creates a new configuration.
Open	Opens an existing configuration.
Close	Closes an opened configuration.
Save	Saves an opened configuration using the same file name.
Save As	Saves an opened configuration to a specified file name.
Export as Text	Stores the configuration as a Tab-separated ASCII file.
Print	Prints the active view.
Language	Selects the active language for SignaSoft 6000.
Exit	Exits SignaSoft 6000.

3.1 New Command

Use this command to create a new configuration in SignaSoft 6000. You will be asked to save the configuration. After the configuration is saved, an 'archive' directory for storing measurement data will automatically be created. If, for example, you store your configuration under 'C:\data\config.sgs', the archive directory will be named 'C:\data\config_Archive'.

You can open an existing configuration with the Open command.

Shortcuts


Toolbar: 
Keys: CTRL+N

3.2 Open Command

Use this command to open an existing configuration.

You can create new configurations with the New command.

Shortcuts

Toolbar: 
Keys: CTRL+O

3.3 Close Command

Use this command to close all windows containing the active configuration. Signasoft 6000 suggests that you save changes to your configuration before you close it. If you close a configuration without saving, you lose all changes made since the last time you saved it. Before closing an untitled configuration, Signasoft 6000 displays the *Save As dialog box* and suggests that you name and save the configuration.


You can also close a configuration by using the Close icon on all of the configuration's windows, as shown below:



3.4 Save Command

Use this command to save the active configuration to its current name and directory. When you save a configuration for the first time, Signasoft 6000 displays the *Save As dialog box* so you can name your configuration. If you want to change the name and directory of an existing configuration before you save it, choose the *Save As* command.

Shortcuts

Toolbar: 
Keys: CTRL+S

3.5 Save As Command

Use this command to save and name the active configuration. Signasoft 6000 displays the *Save As dialog box* so you can name your configuration.

To save a configuration with its existing name and directory, use the *Save* command.

3.6 Export As Text Command

Use this command to store the configuration in a tab-separated ASCII file. You can use this file as a reference containing all device and channel parameters for a specific configuration.

3.7 Language Command

Use this command to select one of the three available languages for SignaSoft 6000. After selecting a new language, you should close SignaSoft 6000 and restart it.

3.8 Recent File Command

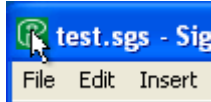
Use the filenames listed at the bottom of the File menu to open the last four configurations you closed. Choose the number that corresponds with the configuration you want to open.

3.9 Exit Command

Use this command to end your SignaSoft 6000 session. You can also use the Close command on the application Control menu. SignaSoft 6000 prompts you to save configurations with unsaved changes.

Shortcuts

Mouse: Double-click the application's Control menu button.



Keys: ALT+F4

4. Edit Menu Commands

The Edit menu offers the following commands:


Cut	Deletes data from the configuration and moves it to the clipboard.
Copy	Copies data from the configuration to the clipboard.
Paste	Pastes data from the clipboard into the configuration.

4.1 Cut Command

Use this command to remove the currently selected data from the configuration and put it on the clipboard. This command is unavailable if there is no data currently selected.

Cutting data to the clipboard replaces the contents previously stored there.

Shortcuts

Toolbar: 


Keys: CTRL+X

4.2 Copy Command

Use this command to copy selected data onto the clipboard. This command is unavailable if there is no data currently selected.

Copying data to the clipboard replaces the contents previously stored there.


Shortcuts

Toolbar: 
 Keys: CTRL+C

4.3 Paste Command

Use this command to insert a copy of the clipboard contents at the insertion point. This command is unavailable if the clipboard is empty.

Shortcuts

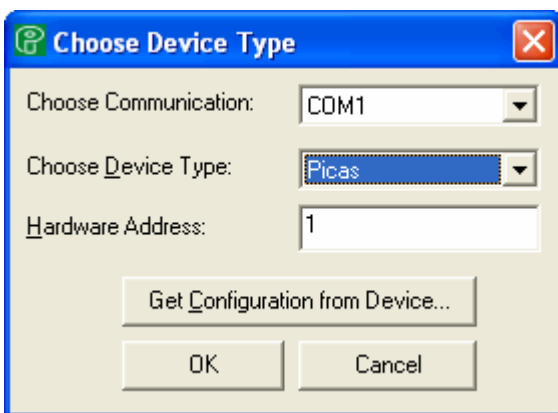
Toolbar: 
 Keys: CTRL+V

5. Insert Menu Commands

The Insert menu offers the following commands:

- | | |
|------------------------------|--|
| Add Picas/Signalog Device... | Creates a new Signalog or Picas device. |
| Add StrainBUSTer Network | Creates a new StrainBUSTer Network. |
| Add Autolog 3000 Device | Creates a new Autolog 3000 device. |
| Add ISM/e.bloxx Network | Creates a new ISM/e.bloxx Network. |
| Add e.gate | Creates a new connection to an e.gate. |
| Add Virtual Channel | Creates a new virtual channel and opens the configuration dialog. |
| Add Rosette Channel | Creates a new rosette channels and opens the configuration dialog. |
| New Graphics Group | Creates a new graphics group. |
| New Numerics Group | Creates a new numerics group. |

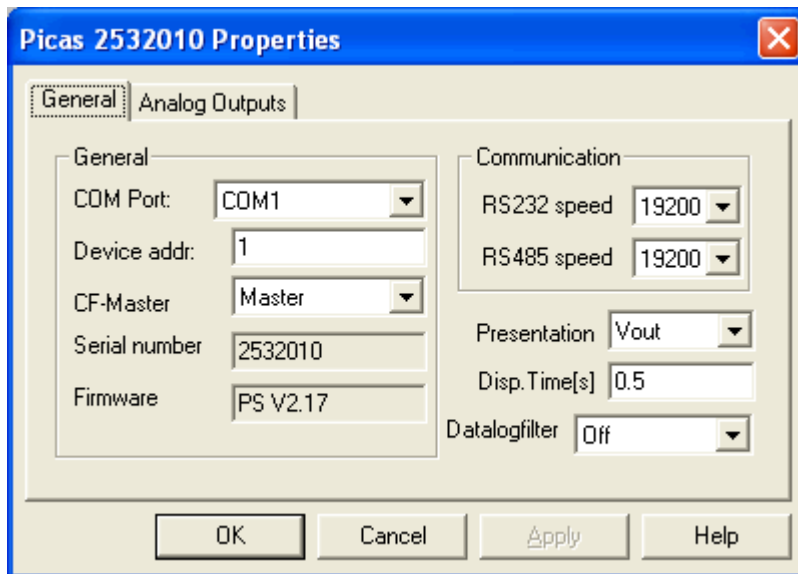
5.1 Add Picas/Signalog Device



Use this command to create a new device. You can select the type of device (Signalog or Picas) and specify its hardware address. Press the '**Get Configuration from Device...**' to retrieve the current configuration (cards and channels) from a connected Signalog device.

If a connected device contains unsaved measurement data in its internal buffer, a message box, which allows you to store this data, will appear at this point. If you choose to store the data, the data will be retrieved from the device and stored on the PC.

5.1.1 General Settings



In this dialog box the general settings for the PICAS/Signalog 6000 device can be made. The following settings can be adjusted here:

Device address: This is the address of the device on the RS485 link. The device, which is directly connected to the PC, must have number 1. The highest number is 253.

CF-Master: This setting determines where the 5KHz carrier for the supply of the inputs is coming from. If it is on Local, this signal is generated on board on the PB6000. If it is set at Master, it is also generated on board, but it will be present on the synchronization lines in the RS485 cable. Other devices can use this signal in the Slave mode.

Serial number: This is the serial number of the processor card.

Firmware: This is the Firmware version number of the PB6000/PB6100.

RS232 speed: The communication speed of the RS232 line can be selected here.

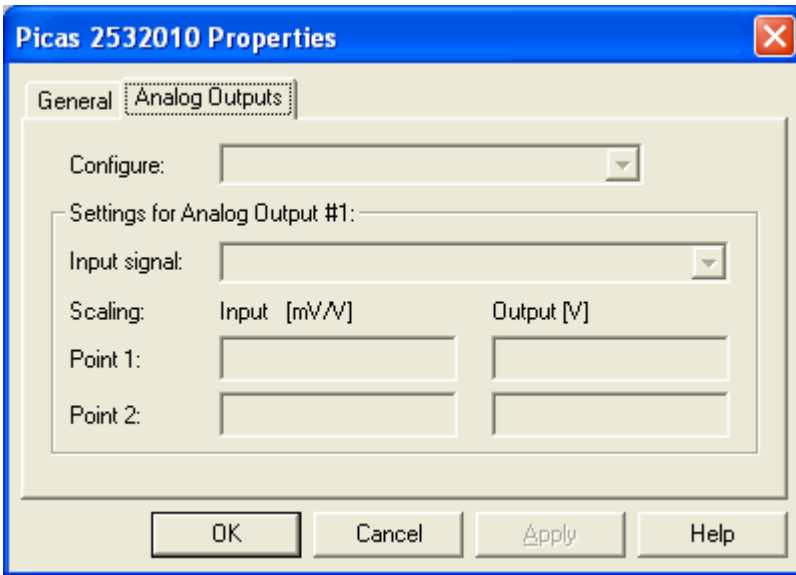
RS485 speed: The communication speed of the RS485 line can be selected here.

Presentation: This setting will influence the measured value, which the channel controls will deliver to the main application. When set to *Vout*, the output voltages will be delivered. Setting it to *Vin* will result in a measurement value of the input voltage. Choosing *Phys* the calculated physical value of the measurement will be delivered. This setting can be overruled by the channel control(s).

Display time: This field holds the interval time over which the average is determined from the measurements, before the value is passed to the control as actual measurement value.

Datalogfilter: This field determines how datalogging in the device should be filtered. The default value 'off' means all measurement values are stored during datalog operations. Otherwise, measurement values are only stored when the selected digital input or trip on a channel is active.

5.1.2 Analog Outputs



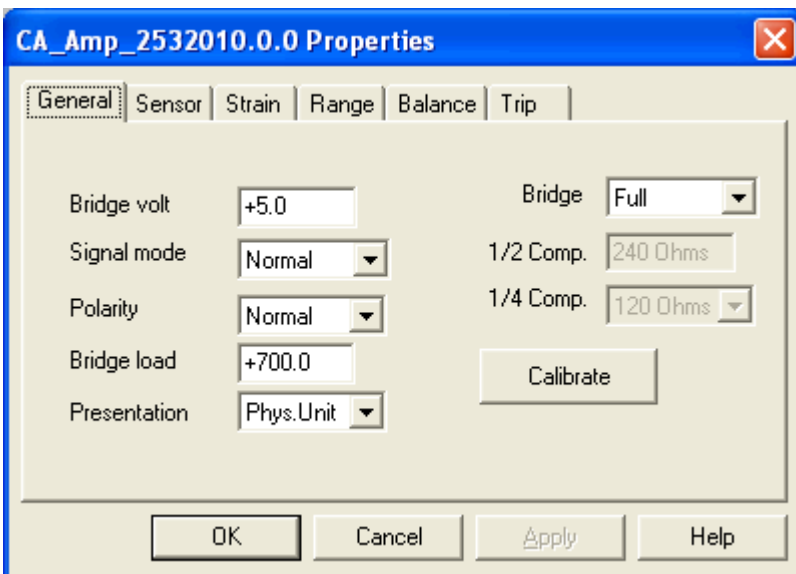
Some PICAS/Signalog 6000 devices have configurable analog outputs. This window configures these outputs. The individual settings explained:

Configure: Select the output to configure here, all settings below are applied to the selected output.

Input signal: Select one of the measured input signals to use as the source for the analog output.

Scaling Point 1/2: Here, you can set the scaling of the input value to the output value. For two different input values, specify the corresponding output voltage to create a linear scaling of the input signal.

5.1.3 CA2CF General Settings



In this dialog box the general settings for this channel can be made. The following settings can be adjusted here:

Bridge supply: This is the supply voltage for the bridge.

Signal mode: A selection can be made here for normal or capacitive measurement.

Polarity: The polarity of the input signal can be changed here.

Bridge load: This is the impedance of the sensor, seen from the input of the amplifier.

Presentation: This setting will influence the measured value which the control will deliver to the main application. When this value is set at *default* the setting of the processor card will be used. When set to *Vout*, the output voltages will be delivered. Setting it to *Vin* will result in a measurement value of the input voltage. Choosing *Phys* the calculated physical value of the measurement will be delivered.

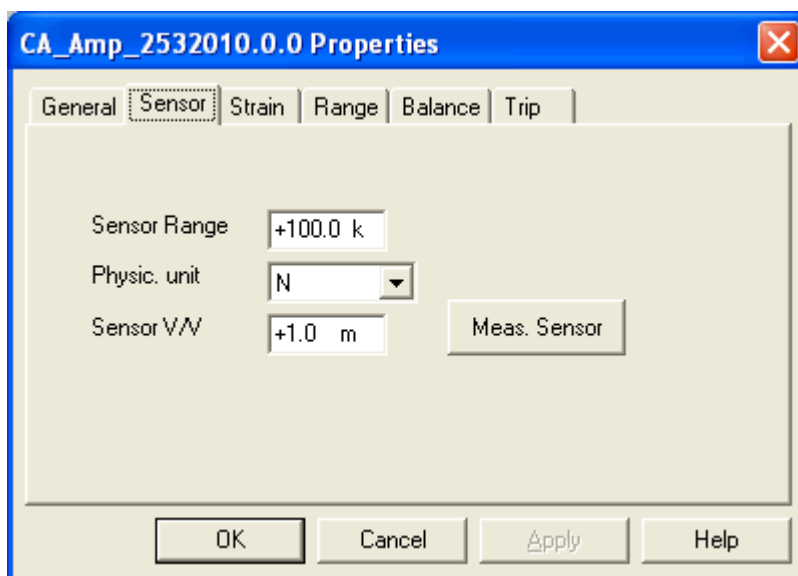
Bridge configuration: A selection can be made from *Full/Half or Quarter Bridge* complementation.

½ Comp.: The complementation resistor for the half bridge is fixed at 240 Ohms.

¼ Comp.: Select the quarter bridge complementation resistor value to use.

Calibrate: At this command, the amplifier will automatically perform a calibration. It measures the real bridge-voltage using the sense-lines and corrects any deviation

5.1.4 CA2CF Sensor Settings



The next settings can be adjusted here:

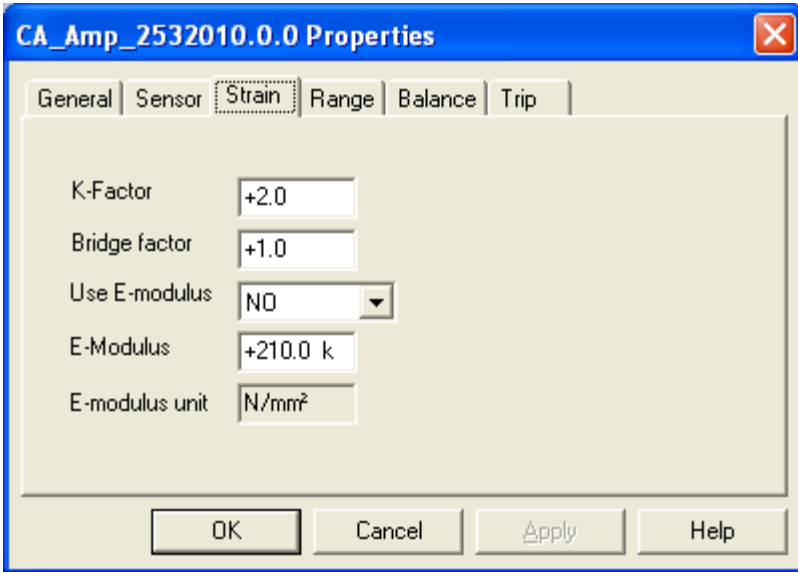
Sensor range: This field holds the maximum range of the sensor.

Unit of sensor: The physical unit of the sensor measurement.

Sensor V/V: The sensitivity of the sensor should be entered here.

Meas. Sensor: Pressing this button causes a measurement to be performed and the result (in V/V) to be placed at the Sensor V/V field. If you enter the value for the sensor range to match the physical condition under which the measurement was done, you have obtained calibration values for this sensor.

5.1.5 CA2CF Strain Settings



The following settings can be adjusted here:

K-Factor: The *k-factor* or *gage factor* can be read from the datasheet of the manufacturer of the strain gauge.

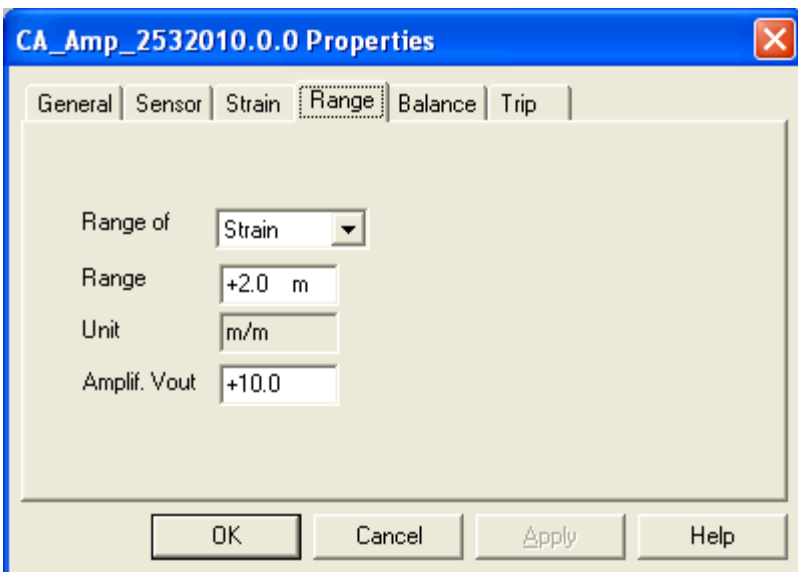
Bridge factor: The *bridge factor* is basically equal to the number of active strain gauges in the bridge. If applicable, it can also be used for the correction of Poisson effects in strain gauge configurations.

Use E modulus: The above parameters allow calculation of the signal into the strain unit *m/m*. If further calculations are to be done to obtain the stress in the material, the *e-modulus* parameters may be set. Set the *use e-modulus* line to yes and set the e-modulus-unit and -value as appropriate for the material to be tested.

E-modulus value: The value van de *E-modulus* or *Young's Modulus*. This value depends on the material on which the strain gauges are mounted.

E-modulus unit: The unit of the measurement is displayed here, when the E-modulus is used.

5.1.6 CA2CF Range Settings



In this dialog box the settings for the range of the amplifier can be made. The following settings can be adjusted here:

Range of: With this field the mode of operation of the amplifier can be selected. A selection can be made from:

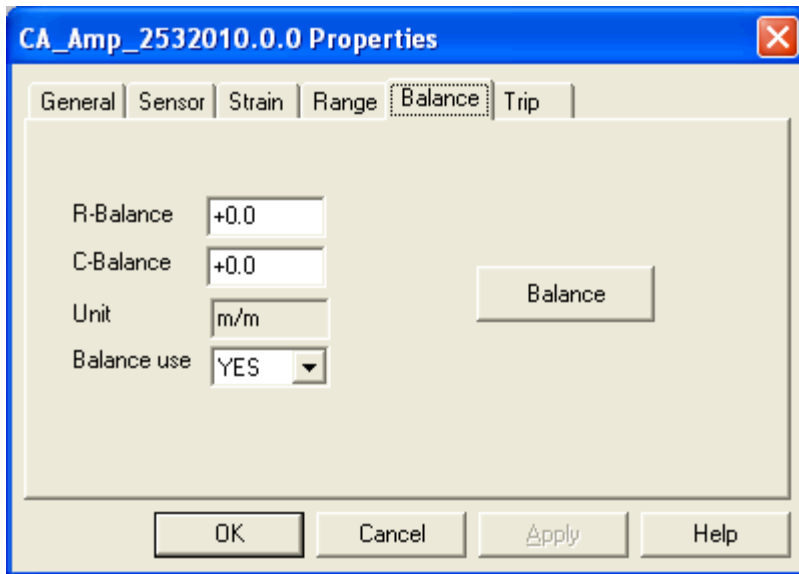
1. Amplifier
2. Sensor
3. Strain

Range: This field contains the range of the amplifier at the above mentioned operating mode.

Amplif. Vout: A maximum output voltage can be entered here.

Unit: This field shows the physical unit of measurement. It can not be adjusted here. This must be done in the sensor or strain menu.

5.1.7 CA2CF Balance Settings



The following settings can be adjusted here:

R-Balance value: This field contains the value of the R balance.

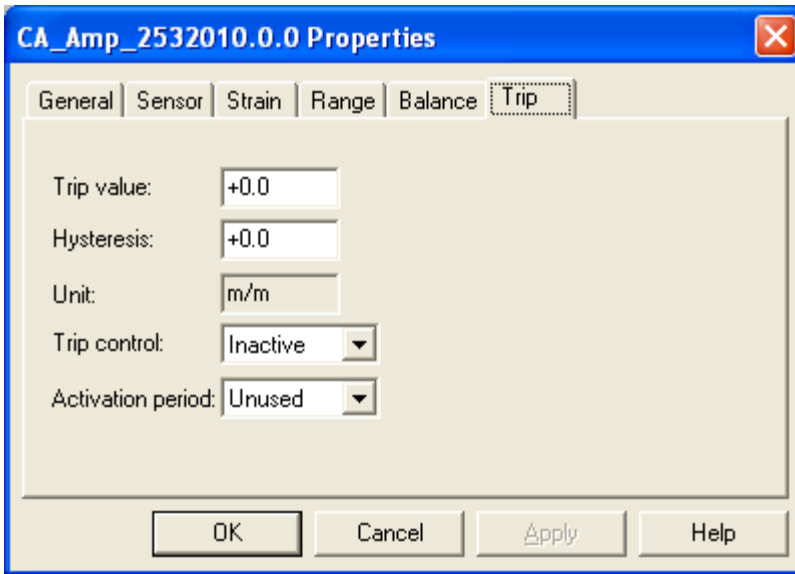
C-Balance value: This field contains the value of the C balance in V/V.

Unit: This field will hold the physical unit of measurement. It can not be adjusted here. This must be done in the sensor or strain menu.

Balance use: A selection can be made whether or not to use the balance option.

Balance command: When the "Balance" button is pressed, a measurement is done and the result is placed on the *R&C-balance-V/V* lines. After balancing, the output of the amplifier should be near 0 Volts.

5.1.8 CA2CF Trip Settings



The following settings can be adjusted here:

Trip value: The Trip value, entered here is the signal level on which the trip action will react.

Hysteresis: The Hysteresis, entered here is used to reset the trip action. When the measured signal is Hysteresis away from the trip value, the trip action is reset.

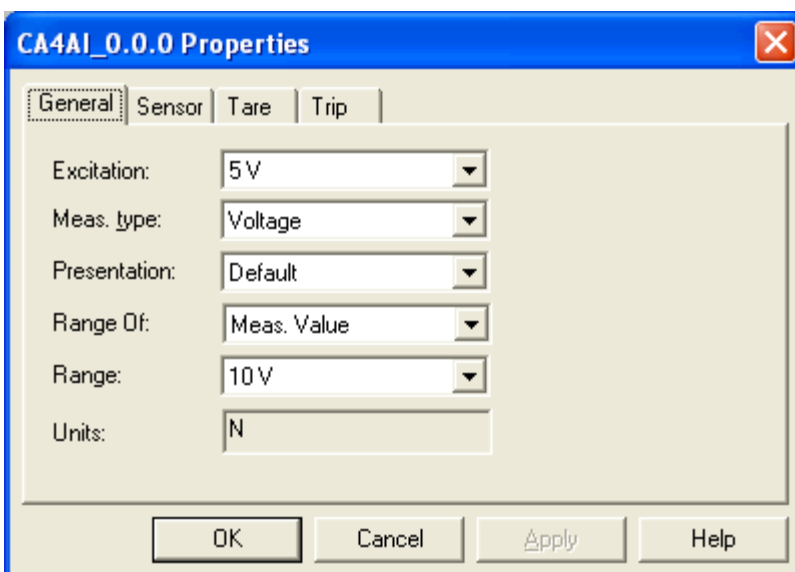
Unit: This field will hold the physical unit of measurement. It can not be adjusted here. This must be done in the sensor or strain menu.

Trip Control: Selectable trip action. A selection can be made from:

- 0 Inactive: no trip action
- 1 High signal: trip on value higher then Trip Value
- 2 Low signal: trip on value lower then Trip Value

Activation period: Determines the minimum amount of time a trip will remain active after the trip value was reached.

5.1.9 CA4AI General Settings



In this dialog box the general settings for this channel can be made.
The following settings can be adjusted here:

Excitation: Select between a 5 V or 1 mA excitation to be applied to the sensor.

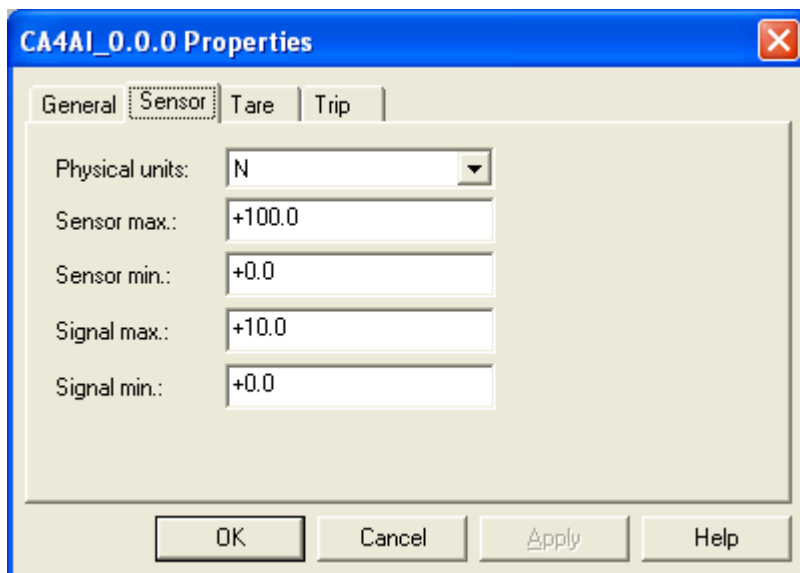
Meas. type: Select the type of sensor to be measured. Note: the first channel of the CA4AI card can not be used to measure thermocouple channels.

Presentation: This setting will influence the measured value which the control will deliver to the main application. When this value is set at *default* the setting of the processor card will be used. Setting it to *Meas. Value* will deliver the measured input value. Choose *Phys. Units* to show the calculated physical value of the measurement.

Range of: With this field the mode of operation of the amplifier can be selected.
A selection can be made from *Meas. Value* or *Sensor*.

Range: This field allows the selection of the range of the amplifier, depending on the selected measurement type.

5.1.10 CA4AI Sensor Settings



The following settings can be adjusted here:

Physical Units: The physical unit of the sensor measurement.

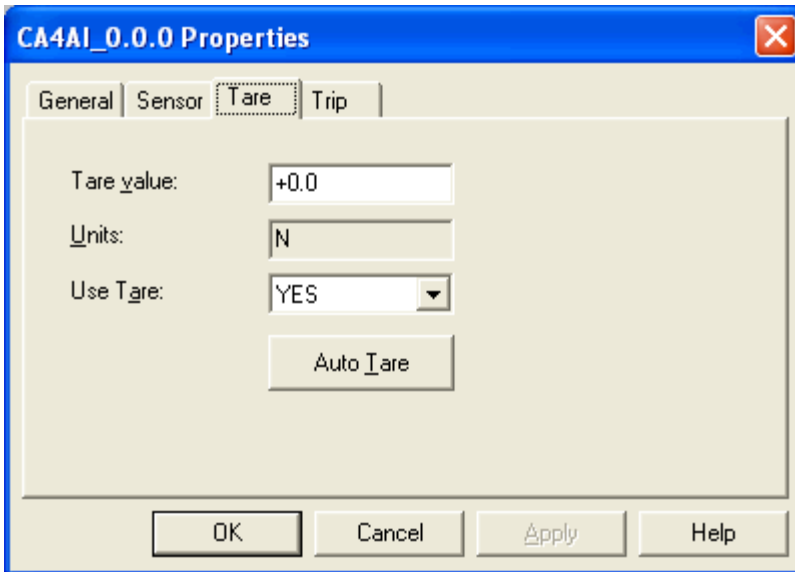
Sensor max.: The sensor value in physical units when the measured value equals signal max.

Sensor min.: The sensor value in physical units when the measured value equals signal min.

Signal max.: The measured value to be used as a reference for the specified sensor max. value.

Signal min.: The measured value to be used as a reference for the specified sensor min. value.

5.1.11 CA4AI Tare Settings



The following settings can be adjusted here:

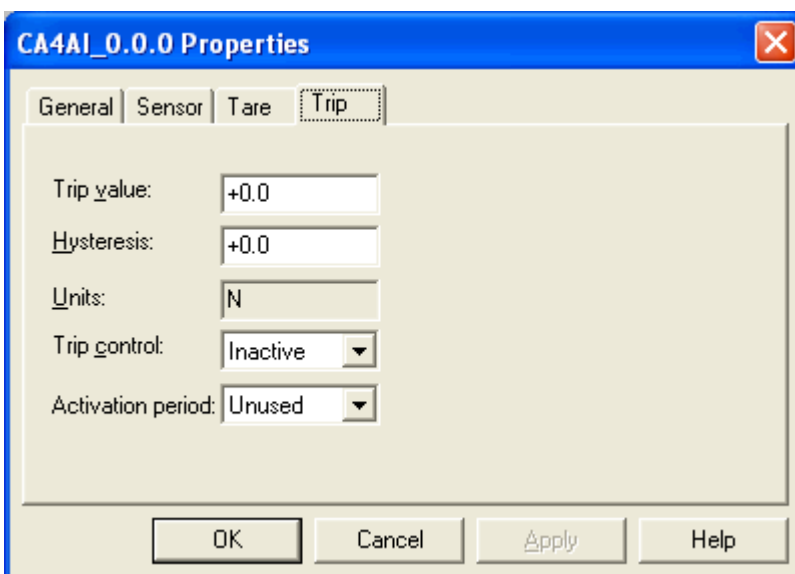
Tare value: This field contains the tare value.

Unit: This field will hold the physical unit of measurement. It can not be adjusted here. This must be done in the sensor menu.

Use tare: A selection can be made for the use of the tare option.

Auto Tare: When the 'Auto Tare' button is pressed, a measurement is done and the result is placed on the Tare value line. The output of the amplifier should now be near 0.

5.1.12 CA4AI Trip Settings



The following settings can be adjusted here:

Trip Value: The Trip value, entered here is the signal level on which the trip action will react.

Hysteresis: The Hysteresis, entered here is used to reset the trip action. When the measured signal is Hysteresis away from the trip value, the trip action is reset.

Unit: This field will hold the physical unit of measurement. It can not be adjusted here. This must be done in the sensor or strain menu.

Trip Control: Selectable trip action. A selection can be made from :

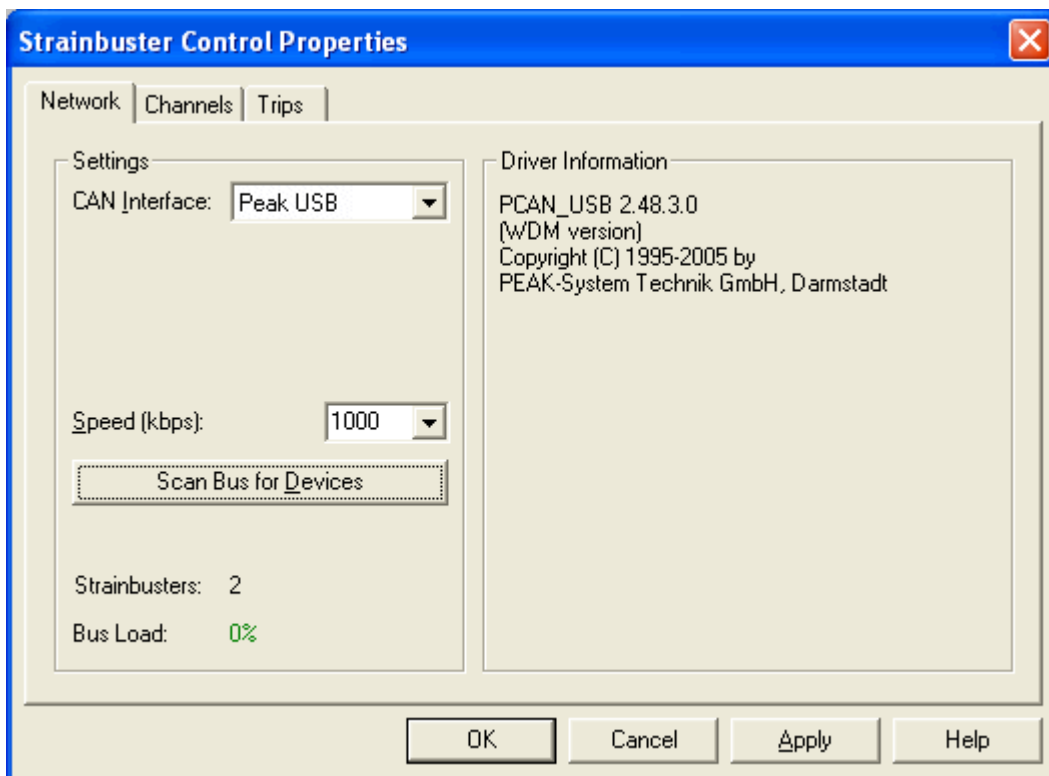
1. Inactive: no trip action
2. high signal: trip on value higher then Trip Value
3. low signal: trip on value lower then Trip Value

Activation Period: The Activation Period determines the minimum time span during which a trip will remain active. When set to anything other than 0, the trip will stay active for at least the given amount of time. If after this time the trip is still active, another span of the same duration is entered.

5.2 Add StrainBUSter Network

Use this command to create a new StrainBUSter network. A new StrainBUSter network will be added and the associated configuration dialog will be shown. In this dialog, you can specify which CAN interface should be used and at which speed the communication should take place. After that, devices on the CAN bus can be detected automatically.

5.2.1 Network Settings



Use this dialog to configure the CAN network. Select the CAN interface and parameters and set the correct speed, then press 'Scan Bus for Devices' to detect which devices are connected to the CAN bus. After detecting the devices, proceed to the Channels Configuration Page to configure the individual channels.

The individual items in this configuration page described:

CAN Interface: Select a CAN interface from this box. The box only shows properly installed CAN interfaces. For the PEAK Dongle (parallel port interface), the preferred option is 'Peak Dongle EPP'. Only select the slower alternative, 'Peak Dongle', if your parallel port does not support EPP.

Board Number: If you have multiple CAN interface boards of the same type, you can enter a specific board number here. Otherwise, use the default value 0.

Port Number: If your CAN interface board has multiple CAN connectors, you can enter the CAN port number here, numbered from 0 upwards. Otherwise, use the default value 0.

Connection: For the parallel port dongle, select the parallel port to use. If your parallel port uses special IRQ or I/O settings, enter them manually.

I/O Port: Only change this value if your parallel port uses a special I/O port. You can select the I/O address from the list or type your own value.

IRQ: Only change this value if your parallel port uses a special IRQ (Interrupt ReQuest). Select the appropriate value from the list.

Speed (kbps): Select the communication speed to use on the CAN network. **This speed must match the speed selected on the StrainBUSTer hardware!** The maximum speed of the CAN network depends on the total cable length.

Scan Bus for Devices: Press this button to scan the configured CAN network for connected devices.

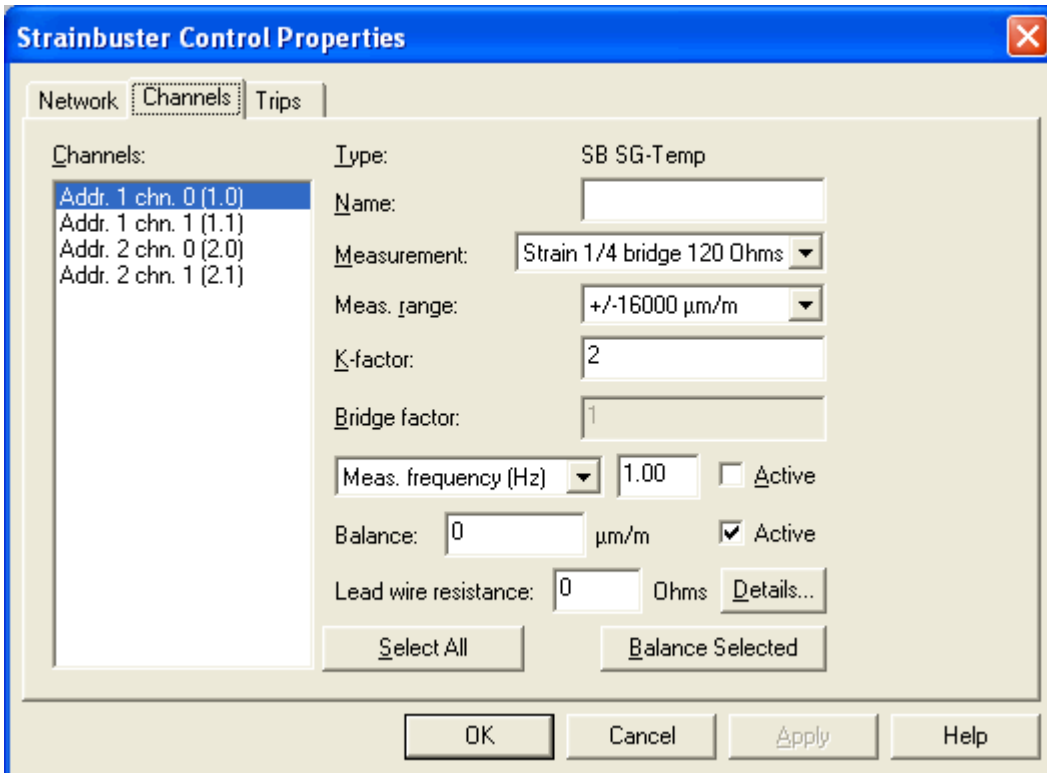
Driver Information: This box shows driver specific information from the CAN interface hardware driver.

5.2.2 CAN Cable Length

Maximum CAN cable length: This table shows maximum cable length depending on communication speed:

Busconfiguration		Max. number of channels at several measurement speeds			
Speed	Total cablelength	100 Hz	20 Hz	10 Hz	1 Hz
1 Mbit /s	<30 m	91	120	120	120
800 kbit/s	<50 m	73	120	120	120
500 kbit /s	<100 m	45	120	120	120
250 kbit /s	<250 m	22	110	120	120
125 kbit /s	<500 m	11	55	110	120
50 kbit /s	<1000 m	4	22	54	120
20 kbit /s	<2500 m	1	9	18	120

5.2.3 Channel Settings



Use this dialog to configure the individual channels. You can select one or more channels to configure from the list on the left side. If this list does not show any channels, go to the Network Configuration Page to configure the CAN network and press the 'Scan Bus for Devices' button.

To select ranges of channels, click on the first channel, then press and hold the SHIFT key and click on the last channel. To select multiple individual channels, press and hold the CTRL key and click on the channels.

The configuration items on the right side of the dialog show the settings for the currently selected channel(s). When multiple channels are selected and an item is blank, it means that the channels have different settings for this item. When you select a new value for the item, it will apply to all selected channels.

The individual items in this configuration page described:

Channels: Using the SHIFT and CTRL keys in combination with the left mouse button, you can select one or more channels from this list to configure.

Name: Sets the name of the selected channel(s). If the same name is assigned to multiple channels, the channel names will automatically be made unique by appending a number. You can use the name to make the channels easier to identify.

Measurement: Sets the type of measurement performed by the selected channel(s). **Beware: this setting must match the setting on the StrainBUSTER device.**

Gain: Sets the gain to be used by the selected channel(s).

Scaling: Check this box to allow the measured value to be scaled to your desired physical units.

Scaling configuration: Press this button to show the scaling dialog.

Units: Sets the freely configurable physical units after scaling.

k-Factor: For strain measurements, enter the k-factor depending on material type.

Bridge factor: For strain measurements, enter the bridge factor. For quarter bridge measurements, this value will usually be 1.

Meas. frequency: Sets the measurement interval for the selected channel(s). For higher intervals (slower measurements), the measurement value is averaged over a longer period of time and will therefore usually be more stable. Depending on your preference, you can show and enter the value as a frequency in Hz or an interval in milliseconds.

Active: Determines whether or not the selected channel(s) are active. Active channels produce measurement values using the configured measurement interval. Inactive channels do not produce measurement values.

Balance Value: This shows the current balance value. You can manually adjust this value if you wish. For Pt-100 measurements you can use this value to correct offsets measured at 0 degrees Celsius.

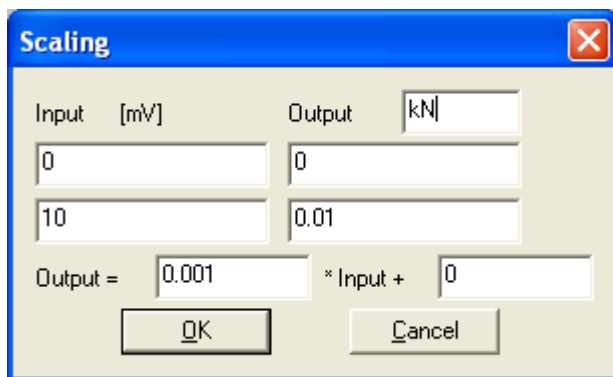
Active: Determines whether or not the balance value is actually used. If this box is unchecked, the measurement value will be calculated as if the balance value equals 0.

Lead wire resistance: Set this value to compensate for signal loss for quarter bridge measurements with long sensor cabling. Press the 'details' button to show the lead wire resistance dialog.

Select All: Press this button to select all available channels.

Balance Selected: Press this button to balance the selected channels.

5.2.4 Scaling



Use this dialog to enter scaling factors to convert a measurement value to physical units.

The individual items in this configuration page described:

Input: Shows the input units (measurement units)

Output: Freely configurable physical units to use for presentation.

Input values: You can choose two different input values for which you know the physical output value you desire.

Output values: When you change one of the output values, the new factors for scaling will be calculated and shown.

Scaling formula: The formula shows how the input values are converted to the output values. You can modify this formula to your liking.

5.2.5 Lead Wire Resistance

The screenshot shows a dialog box titled "Lead wire resistance calculation". It contains the following fields and values:

- Length of wire: 20 meters
- Cross-section area: 0.35 mm² (with a "Default" button)
- Spec. resistivity: 1.75 micro-Ohms * cm (with a "Default" button)
- Wire resistance: 50 Ohms / km
- Wire resistance: 1 Ohms

Buttons for "OK" and "Cancel" are located at the bottom of the dialog.

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.

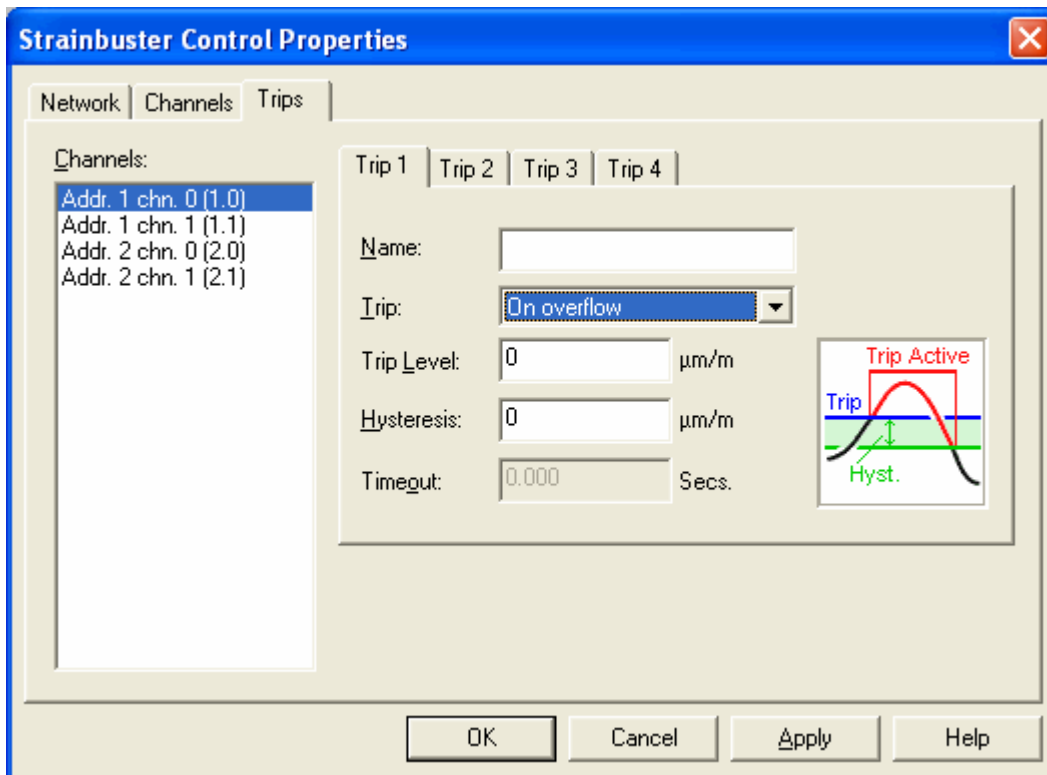
Cross-section area: The cross-section area of the conductive material of the wire in mm².

Spec. resistivity: The specific resistivity of the conductive material of the wire. The default value, 1.75 μ Ohms-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.

5.2.6 Trip Settings



Use this dialog to configure up to four trips for the individual channels. You can select one or more channels to configure from the list on the left side. If this list does not show any channels, go to the Network Settings to configure the CAN network and press the 'Scan Bus for Devices' button.

To select ranges of channels, click on the first channel, then press and hold the SHIFT key and click on the last channel. To select multiple individual channels, press and hold the CTRL key and click on the channels.

The configuration items on the right side of the dialog show the settings for the currently selected channel(s). When multiple channels are selected and an item is blank, it means that the channels have different settings for this item. When you select a new value for the item, it will apply to all selected channels.

The individual items in this configuration page described:

Channels: Using the SHIFT and CTRL keys in combination with the left mouse button, you can select one or more channels from this list to configure.

Trips tabs: From this tab strip, you can choose between the four different trips that can be configured per channel.

Name: Set a freely configurable name for the trip.

Trip: Sets the type of trip, unused trips are marked 'disabled'.

'On overflow' trips will activate as soon as the signal exceeds the trip level and deactivate as soon as the signal drops below 'trip level - hysteresis'.

'Retriggerable overflow' trips will activate as soon as the signal exceeds the trip level and deactivate when the signal remains below 'trip level - hysteresis' for at least 'timeout' seconds.

Trip Level: Sets the level at which the trip should activate. The value should be entered in the units displayed to the right of the input box.

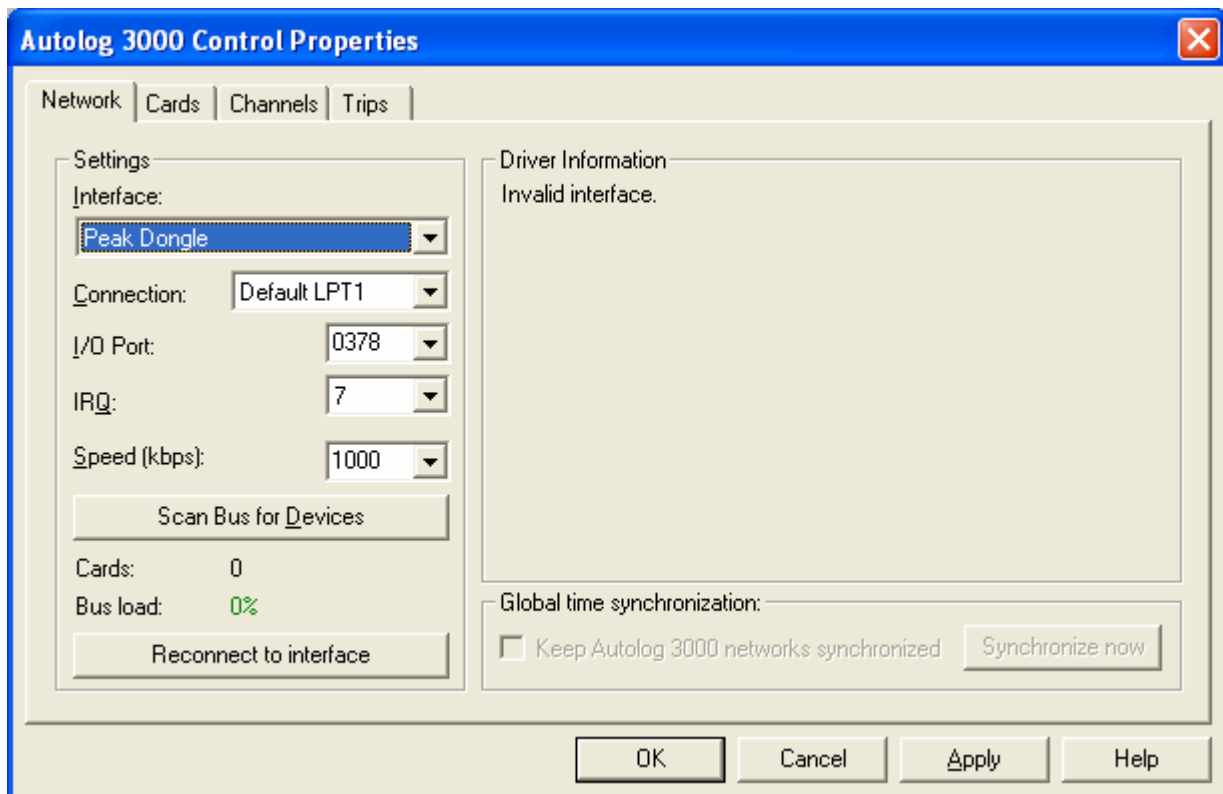
Hysteresis: Sets the hysteresis band around the trip level that determines when the trip should deactivate. The value should be entered in the units displayed to the right of the input box. It will be added to or subtracted from the trip level to find the level at which the trip will deactivate.

Timeout: Sets the timeout time in seconds for retriggerable trips.

5.3 Add Autolog 3000 Device

Use this command to create a new Autolog 3000. A new Autolog 3000 device will be added and the associated configuration dialog will be shown. In this dialog, you can specify which interface should be used and at which speed the communication should take place. After that, the cards available in the Autolog 3000 can be detected automatically.

5.3.1 Network Configuration



Use this dialog to configure the communication network. Select the communication interface and parameters and set the correct speed, then press 'Scan Bus for Devices' to detect which devices are connected to the communication bus.

After detecting the devices, proceed to the Channels Configuration Page to configure the individual channels.

The individual items in this configuration page described:

Interface: Select a communication interface from this box. The box only shows properly installed communication interfaces. For the PEAK Dongle (parallel port CAN interface), the preferred option is 'Peak Dongle EPP'. Only select the slower alternative, 'Peak Dongle', if your parallel port does not support EPP.

Board Number: If you have multiple communication interface boards of the same type, you can enter a specific board number here. Otherwise, use the default value 0.

Port Number: If your communication interface board has multiple communication connectors, you can enter the communication port number here, numbered from 0 upwards. Otherwise, use the default value 0.

Connection: For the parallel port CAN dongle, select the parallel port to use. If your parallel port uses special IRQ or I/O settings, enter them manually.

I/O Port: Only change this value if your parallel port uses a special I/O port. You can select the I/O address from the list or type your own value.

IRQ: Only change this value if your parallel port uses a special IRQ (Interrupt ReQuest). Select the appropriate value from the list.

Speed (kbps): Select the communication speed to use on the CAN bus network. The maximum speed of the CAN network depends on the total cable length.

Scan Bus for Devices: Press this button to scan the configured communication network for connected devices.

Bus load: Shows the amount of communication as a percentage of the available bandwidth on the communication interface between PC and device.

Reset interface: Press this button to reset the communication interface.

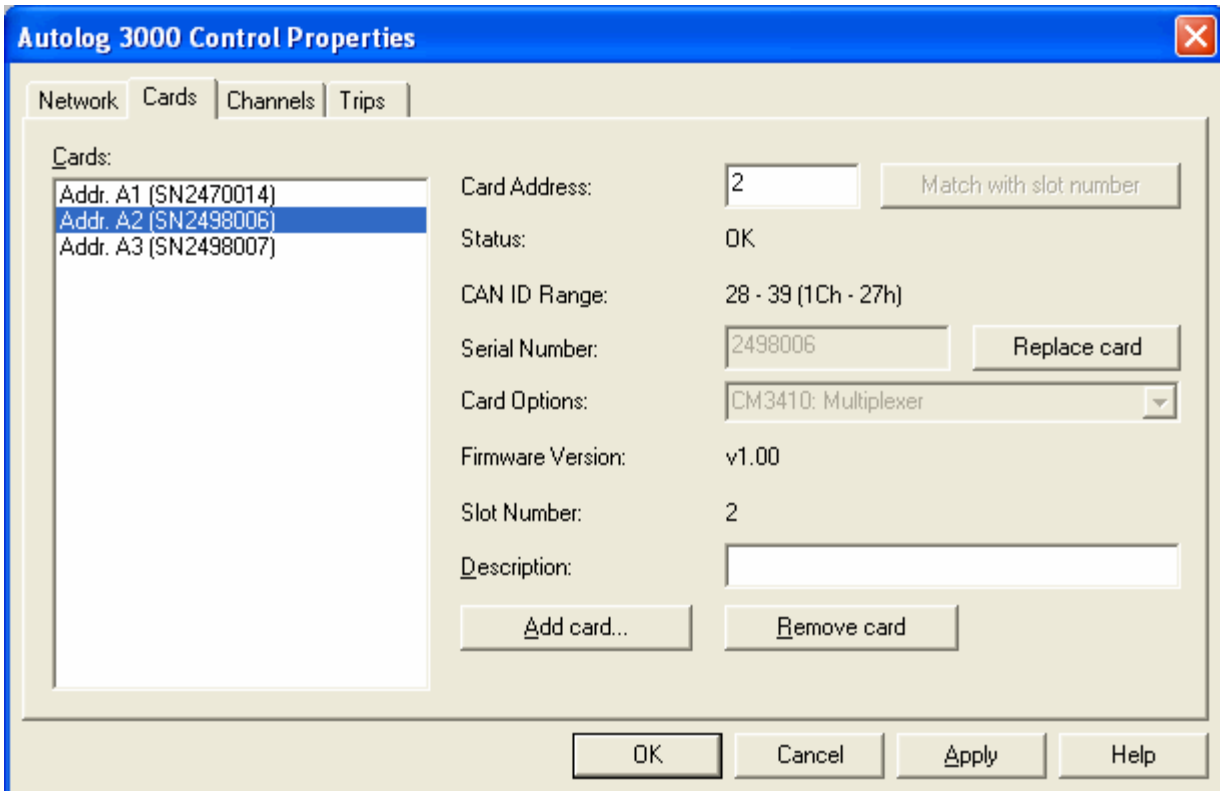
Driver Information: This box shows driver specific information from the communication interface hardware driver.

5.3.2 CAN Bus Cable Length

For CAN bus communication, this table shows maximum cable length depending on communication speed:

CAN Bus configuration		Max. number of channels at several measurement speeds			
Speed	Total cablelength	1000 Hz	100 Hz	10 Hz	1 Hz
1 Mbit /s	<30 m	7	70	700	7000
500 kbit /s	<100 m	3	35	350	3500
250 kbit /s	<250 m	2	17	175	1750
125 kbit /s	<500 m	1	8	87	875
50 kbit /s	<1000 m	-	3	35	350
20 kbit /s	<2500 m	-	1	14	140

5.3.3 Cards Configuration



Use this dialog to configure the individual cards. You can select the card to configure from the list on the left side. If this list does not show any cards, go to the Network Configuration Page to configure the communication network and press the 'Scan Bus for Devices' button.

The configuration items on the right side of the dialog show the settings for the currently selected card.

The individual items in this configuration page described:

Cards: Select a card from this list to show its information.

Card Address: Shows the logical card address of the card. This address determines the CAN ID range that the card uses for communication. If you change this value, the card will be reprogrammed to communicate using the new address.

Status: Shows the communication status of the card. The status is OK if all channels on the card respond as expected, DISCONNECTED if the card fails to produce measurement values for one or more channels on the card.

CAN ID Range: Shows the CAN ID range that the card uses for communication.

Serial number: Shows the serial number of the selected card.

Replace card: Allows you to replace a card at a specific address by another one (with a different serial number). For information, refer to the section on replacing cards below.

Card options: Shows the type of card and its option modules.

Firmware version: Shows the firmware version of the card.

Slot number: Shows which slot number in the Autolog 3000 the card occupies.

Description: Use this description to make the card easier to identify.

Add card: Use this button to manually add a new card to the configuration. You must know and specify the serial number of the card to be able to use this function.

Remove card: Use this button to remove a specific card and all its settings from the current configuration.

5.3.4 Replacing a Card

To replace a card in the Autolog 3000 system, use the Cards Configuration dialog. The exact procedure depends 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 your application and make a backup of the configuration/settings file, where applicable.
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 your software-configuration. Now go to the cards configuration dialog, and select the card that you replaced.
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.

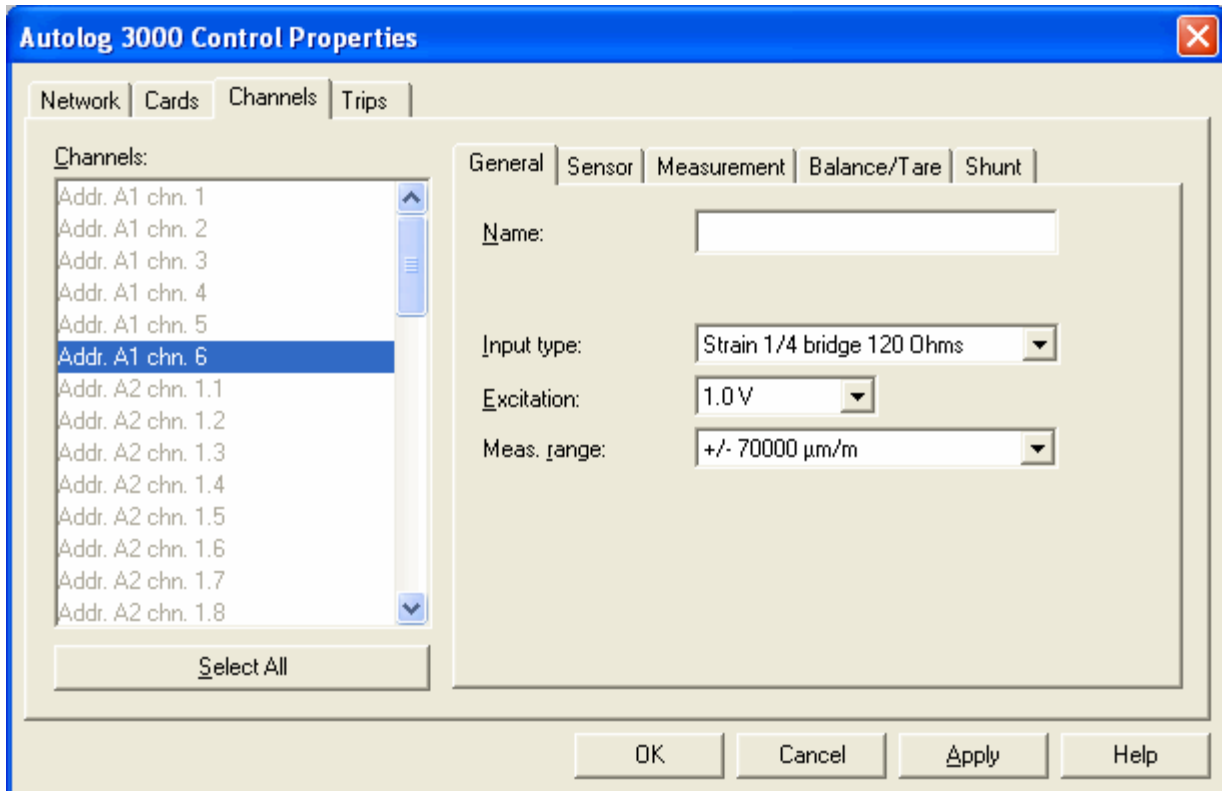
USB or Ethernet interface

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 your application and make a backup of the configuration/settings file, where applicable.
2. Switch off the device, and replace the card.
3. Switch the device back on and load your software-configuration. Now go to the cards configuration dialog, and select the card that you replaced.

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.

5.3.5 Channels Configuration



Use this dialog to configure the individual channels. You can select one or more channels to configure from the list on the left side. If this list does not show any channels, go to the Network Configuration Page to configure the communication network and press the 'Scan Bus for Devices' button.

To select ranges of channels, click on the first channel, then press and hold the SHIFT key and click on the last channel. To select multiple individual channels, press and hold the CTRL key and click on the channels.

The configuration items on the right side of the dialog show the settings for the currently selected channel(s). When multiple channels are selected and an item is blank, it means that the channels have different settings for this item. When you select a new value for the item, it will apply to all selected channels.

The individual items in this configuration page described:

Channels: Using the SHIFT and CTRL keys in combination with the left mouse button, you can select one or more channels from this list to configure.

Select All: Press this button to select all available channels.

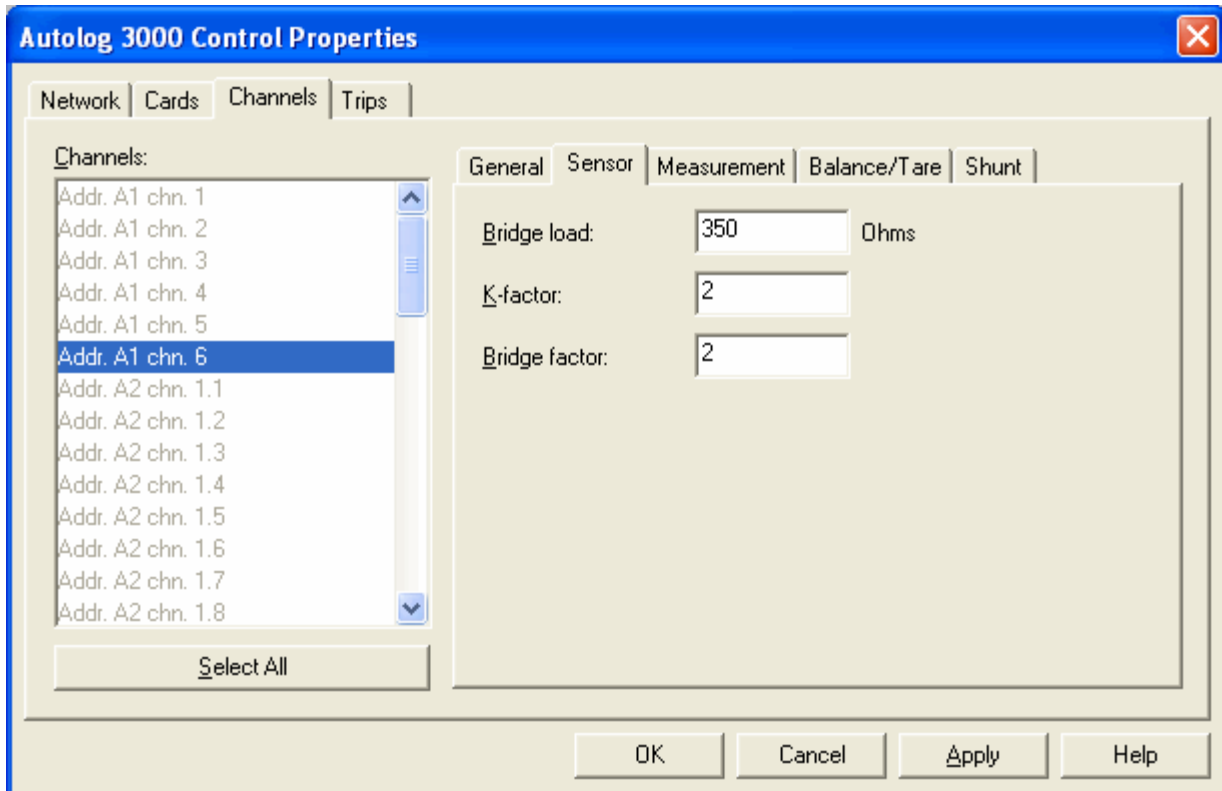
Name: Sets the name of the selected channel(s). If the same name is assigned to multiple channels, the channel names will automatically be made unique by appending a number. You can use the name to make the channels easier to identify.

Input type: Sets the type of measurement performed by the selected channel(s).

Excitation: Sets the excitation voltage as supplied to the sensor.

Meas.range: Sets the range to be used by the selected channel(s). Which ranges are available depends on the type of measurement.

5.3.6 Channels: Bridge Sensor



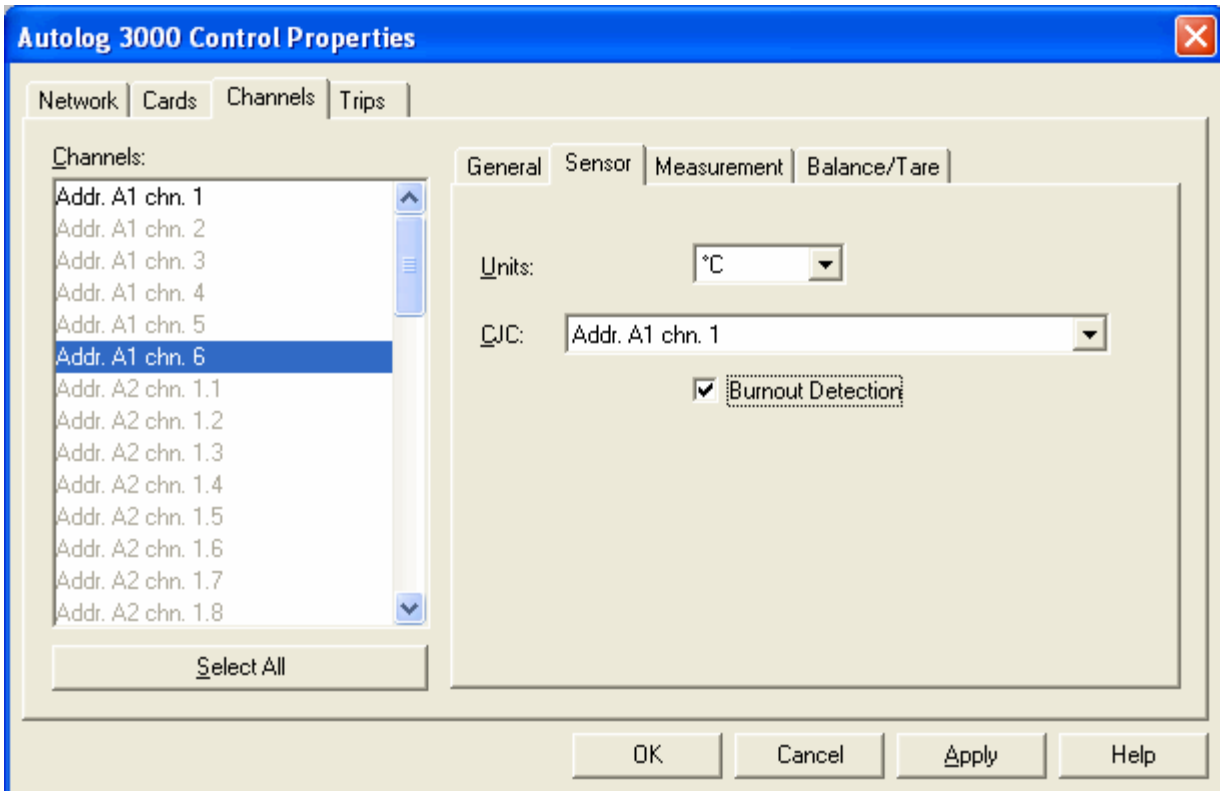
The sensor tab page of the channel configuration contains parameters specific to strain gage, pt-100 and thermocouple measurements.

Bridge load: For strain gage and transducer measurements, the sensor impedance. This value is used to determine the expected shunt measurement values. This value is not used during normal measurement.

K-factor: For strain gage measurements, the gage factor of the strain gage element, depending on material type.

Bridge factor: For strain gage measurements, the bridge factor for half and full-bridge measurements.

5.3.7 Channels: Thermocouple Sensor



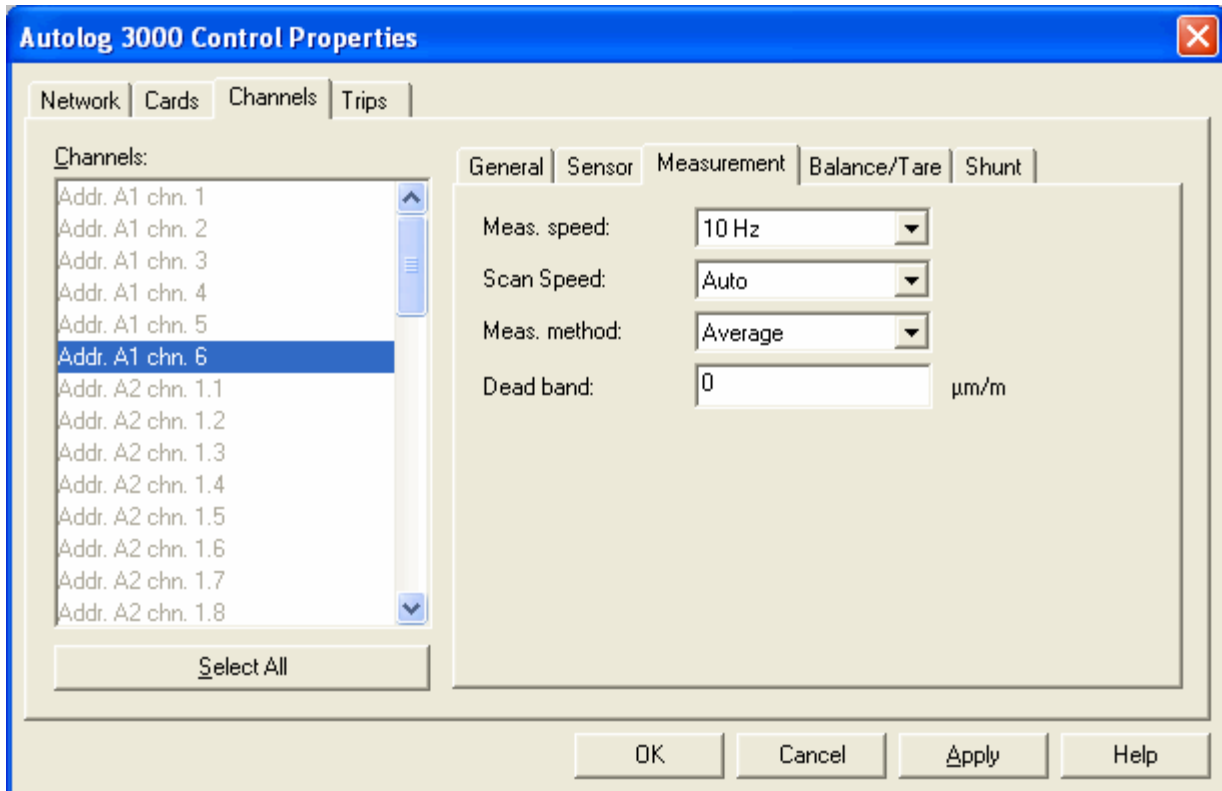
The sensor tab page of the channel configuration contains parameters specific to strain gage, pt-100 and thermocouple measurements.

Units: Selects the presentation units for temperature measurements (Celsius, Fahrenheit or Kelvin).

CJC: For thermocouples, selects the (Pt-100) input to use for cold-junction-compensation measurement.

Burnout detection: For thermocouples, burnout detection makes sure that a broken wire on a thermocouple measurement does not go undetected.

5.3.8 Channels: Measurement



The measurement tab page of the channel configuration determines the measurement speed and related parameters.

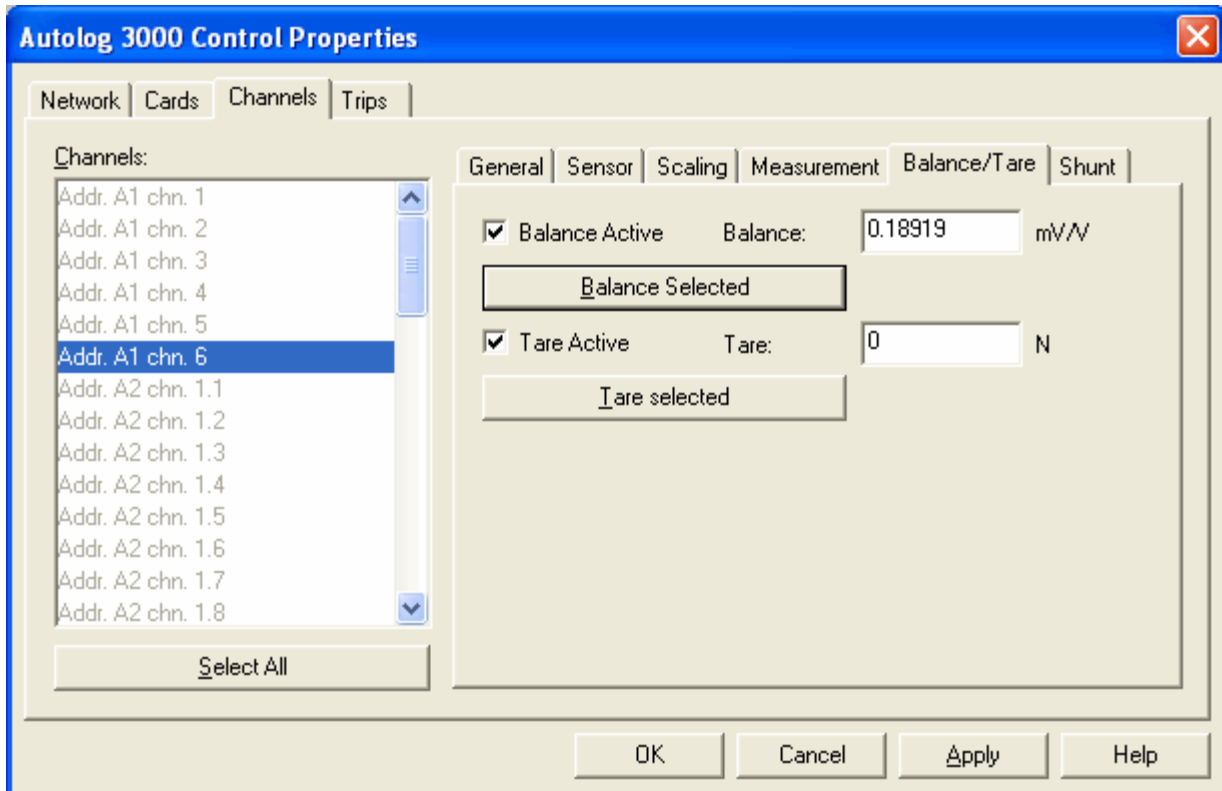
Meas. speed: The speed at which the channel should output measurement values.

Scan speed: The speed at which the card measures internally. If set to 'Auto', the optimum speed will be determined automatically.

Meas. method: For measurement speeds lower than 1000 Hz, this setting determines what operation the hardware should perform on the raw measurement values (acquired at the speed as defined under 'scan speed') to reduce it to the requested amount of data.

Dead band: If set to 0 (default), all measured values will be output at the requested speed. Otherwise, measured values will only be output if they differ from the last output value by more than the dead band setting. Regardless of the dead band setting, at least one measurement value will be output every second.

5.3.9 Channels: Balance/Tare



The balance/tare tab page of the channel configuration contains settings and commands for balance and tare functions.

Balance active: Determines whether or not the balance value is used for this channel.

Balance selected: Performs a balance command on the current selection of channels. This command averages the measured values over a period of 1 second to determine a stable balance value.

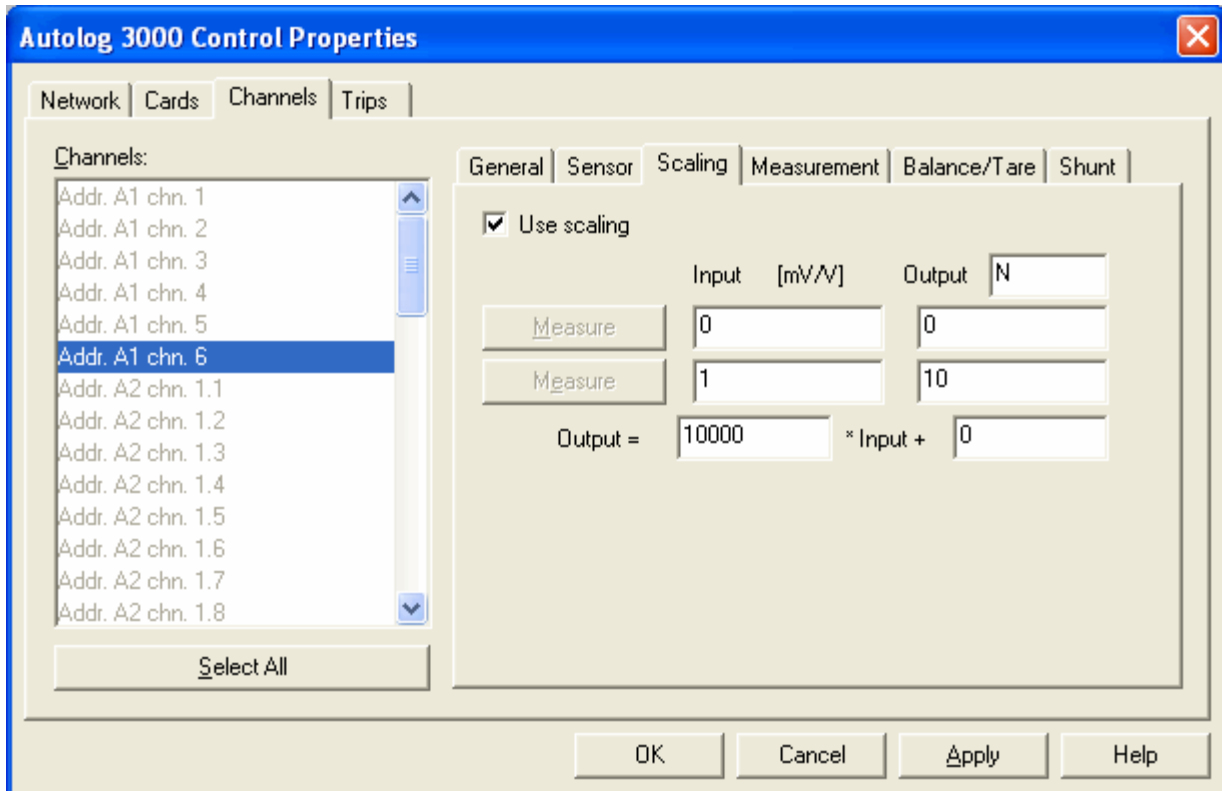
Balance value: The current balance value. It is possible to manually modify this value.

Tare active: Determines whether or not the tare value is used for this channel.

Tare selected: Performs a tare command on the current selection of channels. This command averages the measured values over a period of 1 second to determine a stable tare value.

Tare value: The current tare value. It is possible to manually modify this value.

5.3.10 Channels: Scaling



The scaling tab page of the channel configuration contains scaling parameters to allow for a linear scaling from the input value to suitable engineering units.

Use scaling: Determines whether or not linear scaling is used.

Output units: Freely assignable engineering units in which the result of the linear scaling is expressed.

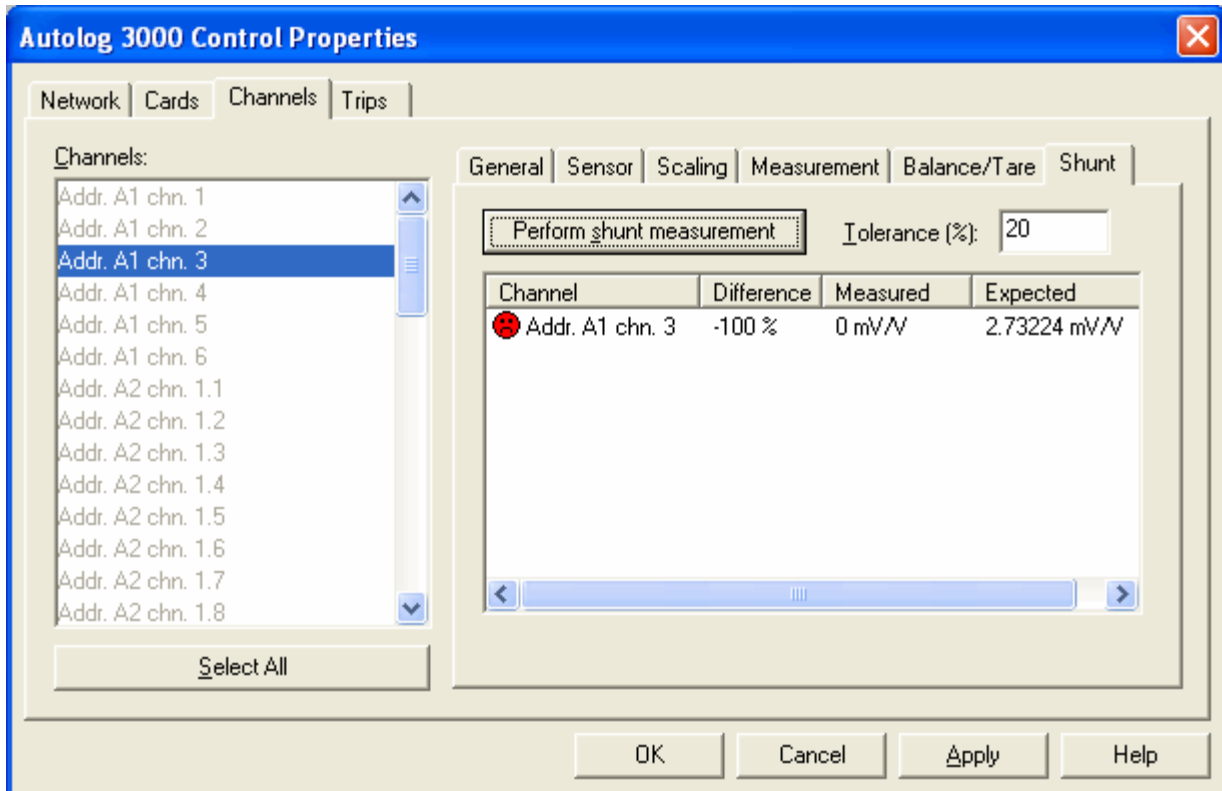
Measure: Press this button to obtain the latest measurement value for this channel as input value. You should first activate the channel and set a suitable measurement speed (slow to get a stable value) before using this function.

Input values: You can choose two different input values for which you know the physical output value you desire.

Output values: When you change one of the output values, the new factors for scaling will be calculated and shown.

Scaling formula: The formula shows how the input values are converted to the output values. You can modify this formula to your liking.

5.3.11 Channels: Shunt



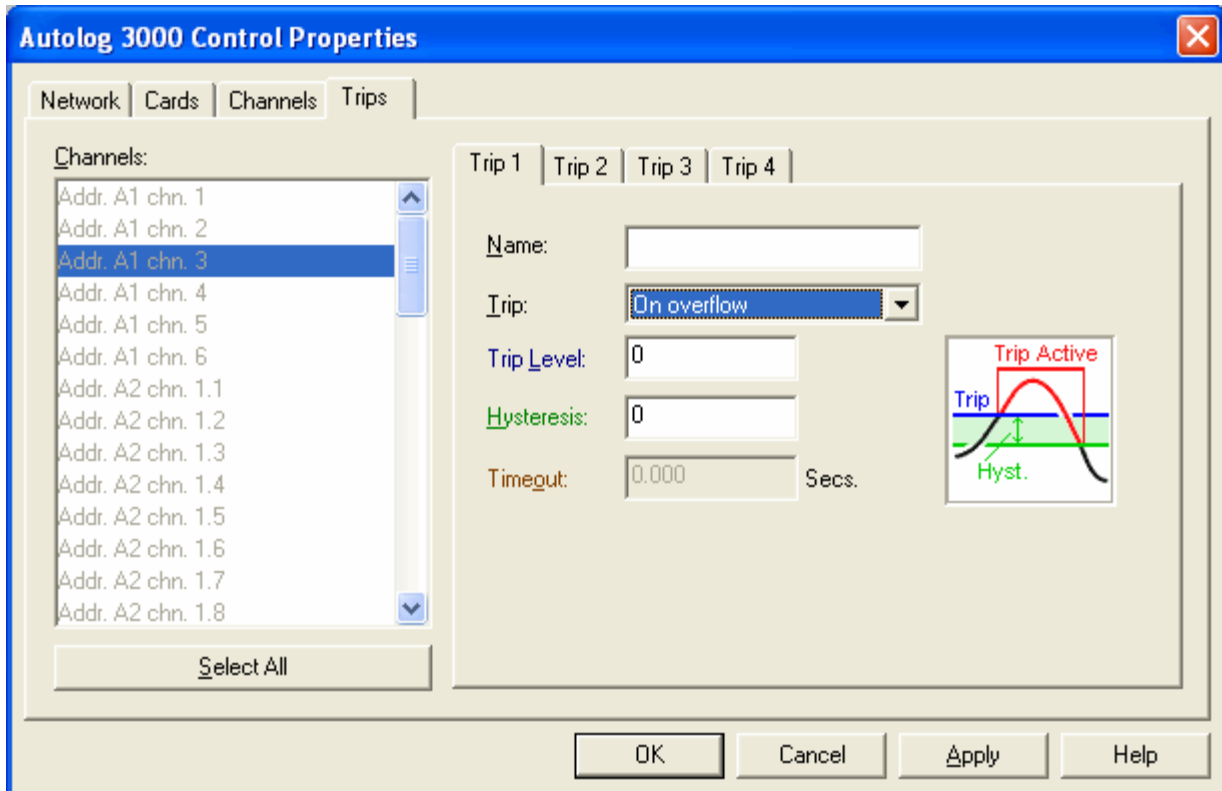
The shunt tab page of the channel configuration allows you to perform shunt measurements.

Perform shunt measurement: Press this button to perform a shunt measurement on the selected channels. A shunt measurement will only be performed for channels configured to a suitable input type (strain gage, transducer).

Shunt tolerance: Determines the maximum difference allowed between the expected and the measured value. If the difference is greater, a red smiley is shown in the measurement results.

Results: The results of a shunt measurement are shown in a list which includes the measured value in mV/V, the expected value and the difference in percent for each measured channel.

5.3.12 Trips Configuration



Use this dialog to configure up to four trips for the individual channels. You can select one or more channels to configure from the list on the left side. If this list does not show any channels, go to the Network Configuration Page to configure the communication network and press the 'Scan Bus for Devices' button.

To select ranges of channels, click on the first channel, then press and hold the SHIFT key and click on the last channel. To select multiple individual channels, press and hold the CTRL key and click on the channels.

The configuration items on the right side of the dialog show the settings for the currently selected channel(s). When multiple channels are selected and an item is blank, it means that the channels have different settings for this item. When you select a new value for the item, it will apply to all selected channels.

The individual items in this configuration page described:

Channels: Using the SHIFT and CTRL keys in combination with the left mouse button, you can select one or more channels from this list to configure.

Trips tabs: From this tab strip, you can choose between the four different trips that can be configured per channel.

Name: Set a freely configurable name for the trip.

Trip: Sets the type of trip, unused trips are marked 'disabled'.

'On overflow' trips will activate as soon as the signal exceeds the trip level and deactivate as soon as the signal drops below 'trip level - hysteresis'.

'Retriggerable overflow' trips will activate as soon as the signal exceeds the trip level and deactivate when the signal remains below 'trip level - hysteresis' for at least 'timeout' seconds.

Trip Level: Sets the level at which the trip should activate. The value should be entered in the units displayed to the right of the input box.

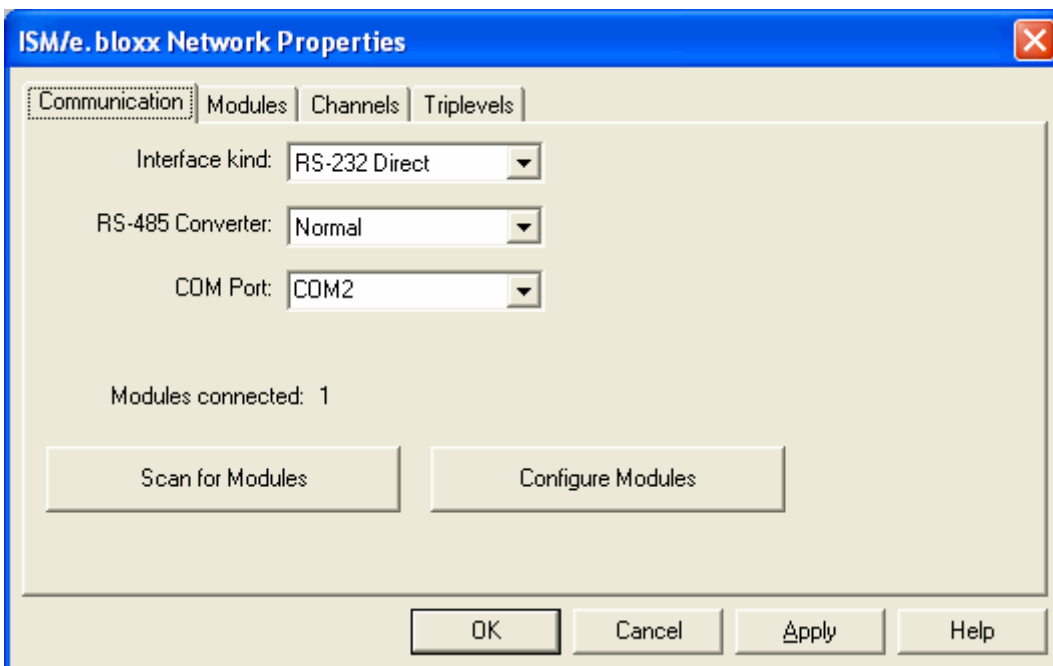
Hysteresis: Sets the hysteresis band around the trip level that determines when the trip should deactivate. The value should be entered in the units displayed to the right of the input box. It will be added to or subtracted from the trip level to find the level at which the trip will deactivate.

Timeout: Sets the timeout time in seconds for retriggerable trips.

5.4 Add ISM/e.bloxx Network

Use this command to create an ISM/e.bloxx network. Communication with a (strain of) ISM/e.bloxx Module(s) takes place over a serial port. You should configure the modules using the Gantner ICP software. After that, SignaSoft 6000 will automatically show the configured channels.

5.4.1 Communication Settings



On this property page you can configure the serial communications port and scan for available ISM or e.bloxx modules. 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!

Interface kind: Leave this setting to its default of 'RS-232 Direct' unless you have communication problems (usually because of a USB to RS-232 converter), in which case you should try 'RS-232 RF-Modem'.

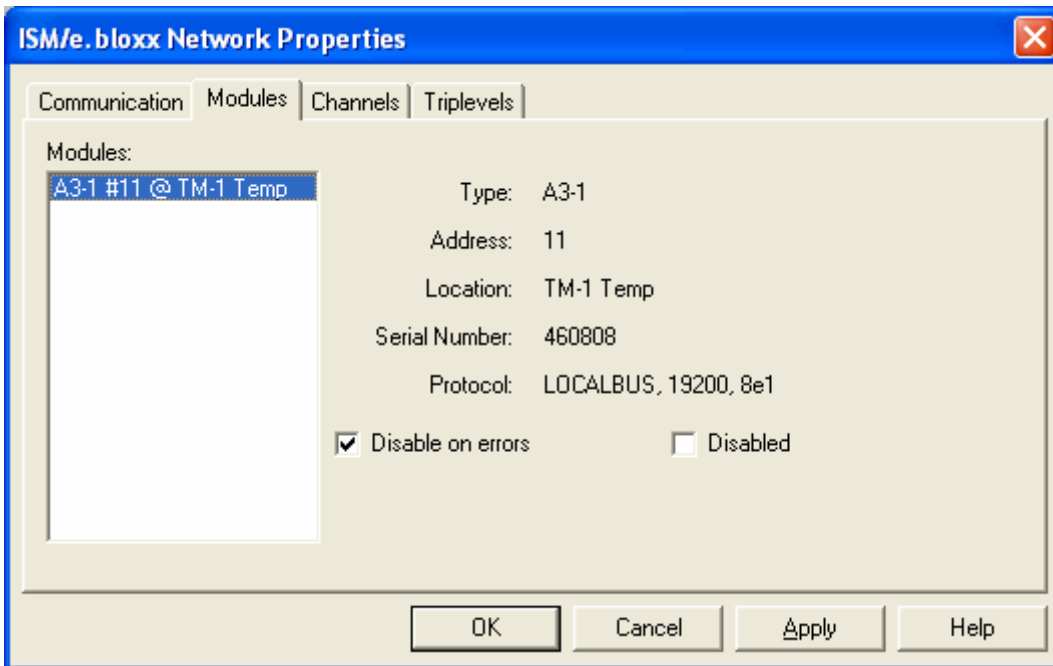
RS-485 Converter: Leave this setting to its default of 'normal' unless you use a switching RS-485 converter.

COM Port: Select a serial port between COM1 and COM8 where the ISM or e.bloxx modules are connected.

Scan for Modules: After setting the correct COM port, this button will start a scan to find the available ISM or e.bloxx modules. A scan can take up to 15 seconds.

Configure Modules: This button starts the ICP configuration software to allow you to configure the variables and settings of the modules.

5.4.2 Module Settings



On this property page you can view information about the connected modules. Also, you can reactivate disabled modules.

Modules: This box shows the available modules. To configure or show information on a module, first select a module from the list. If this box is empty, check the communication settings first.

Type: Show the type of ISM/e.bloxx module.

Address: Shows the unique module address.

Location: Shows the location of the module.

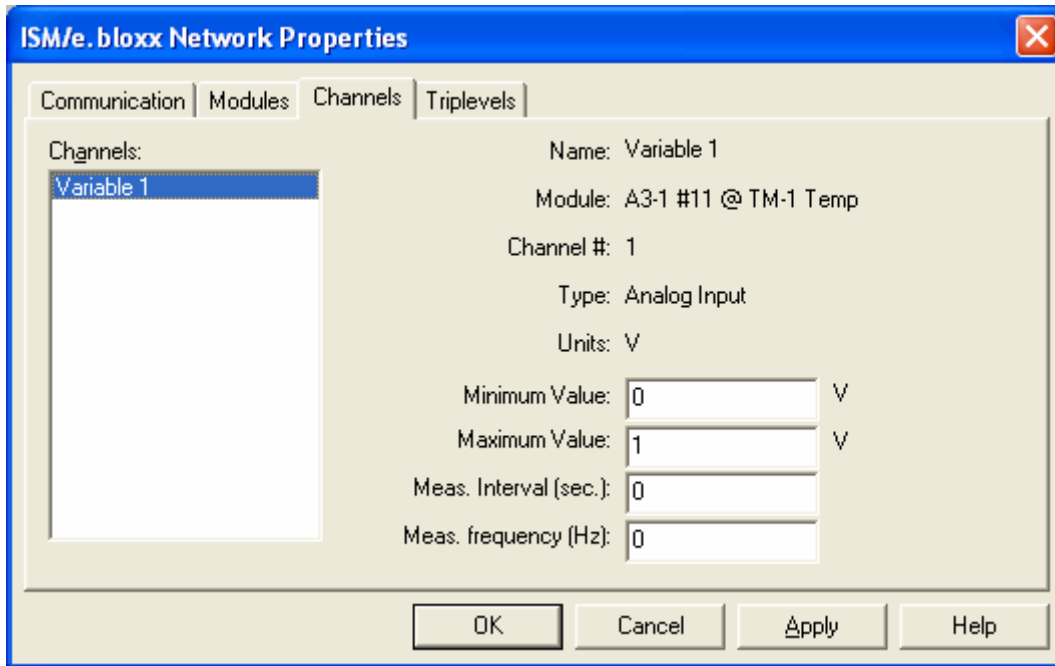
Serial number: Shows the unique serial number of the module.

Protocol: Shows the communication protocol and speed used by the module. The speed of e.bloxx modules must be 115200 or lower to make communication possible.

Disable on errors: 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.

Disabled: If this box is checked, the software will not attempt to communicate with this module. When the module has been disabled due to communication errors, you can uncheck this box to allow communication to start again.

5.4.3 Channel Settings



On this property page you can view information about the available channels. Also, you can set the measurement interval for each channel.

Channels: This box shows the available channels. To configure a channel, first select it from the list. You can also select multiple channels if you want to configure the same measurement interval on all of those channels. If this box is empty, check the communication settings first.

Name: Shows the name of the channel.

Module: Shows the module that contains this channel.

Channel #: Shows the channel number within the module.

Type: Shows the measurement type of the channel.

Units: Shows the units used to present measurement values.

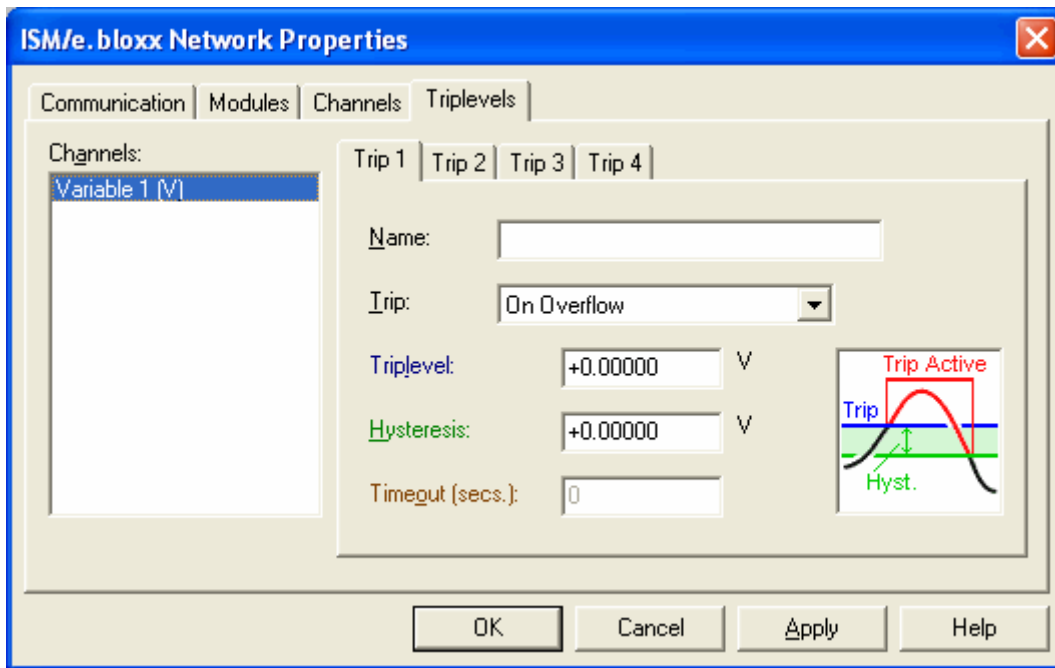
The settings mentioned above can not be changed from this dialog. To change these settings, go to the 'Communication Settings' and click the 'Configure Modules' button to start the ICP configuration software.

Minimum Value/Maximum Value: Use these fields to specify the expected range of the measured value. This is used in, for example, bar graph displays. The output value is NOT clipped to this range, so these settings have no influence on the actual measurement values produced.

Meas. Interval (sec.): Shows the requested measurement interval for the channel in seconds. Note that all channels in a module will be scanned at the same time. The channel with the highest configured measurement speed determines the actual scan frequency for a module. 'Slower' channels within the same module will be scanned at the same speed, but less measurement values will actually be shown and stored.

Meas. frequency (Hz): Shows the requested measurement frequency for the channel in Hz (the inverse of the measurement interval in seconds).

5.4.4 Trips Settings



On this property page you can configure up to 4 separate trips for each of the available channels.

Channels: This box shows the available channels. To configure trips, first select a channel from the list. You can also select multiple channels if you want to configure the same trip on all of those channels. If this box is empty, check the communication settings first.

Trip Selection: There are 4 different trips available for each channel, which can be configured separately.

Name: Each trip can have its own name, which is freely configurable.

Type: Sets the type of trip, unused trips are marked 'disabled'.

'On overflow' trips will activate as soon as the signal exceeds the trip level and deactivate as soon as the signal drops below 'trip level - hysteresis'.

'Retriggerable overflow' trips will activate as soon as the signal exceeds the trip level and deactivate when the signal remains below 'trip level - hysteresis' for at least 'timeout' seconds.

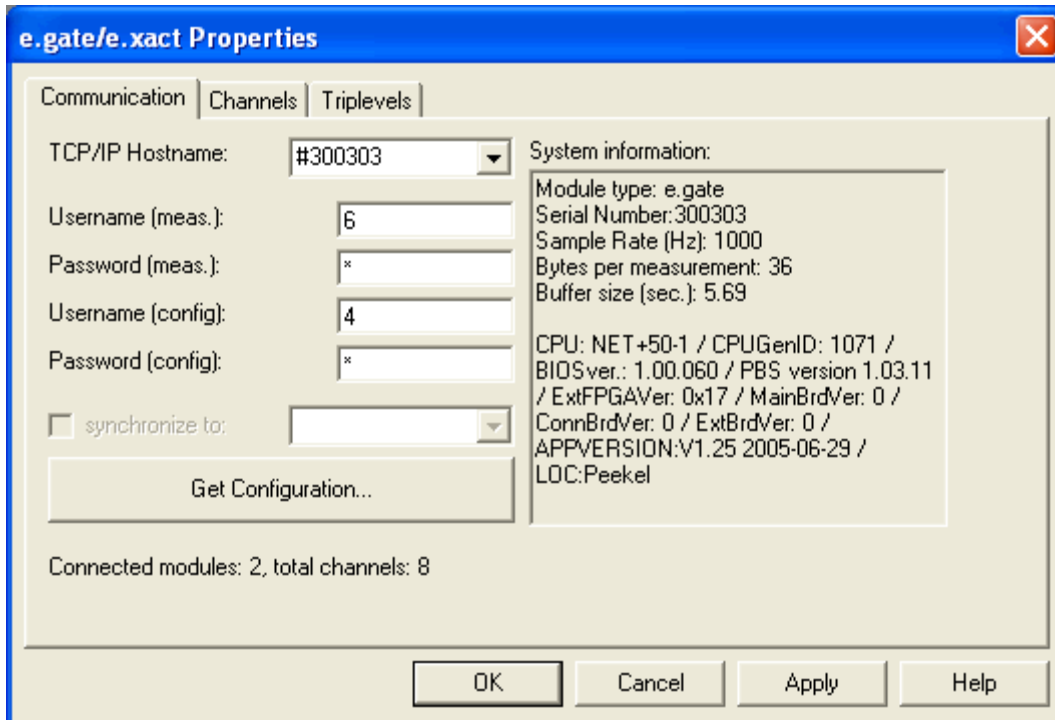
Trip level: Here you can specify the level at which the trip will activate.

Hysteresis: Sets the hysteresis band (as an absolute value) around the trip level that determines when the trip should deactivate. It will be added to or subtracted from the trip level to find the level at which the trip will deactivate.

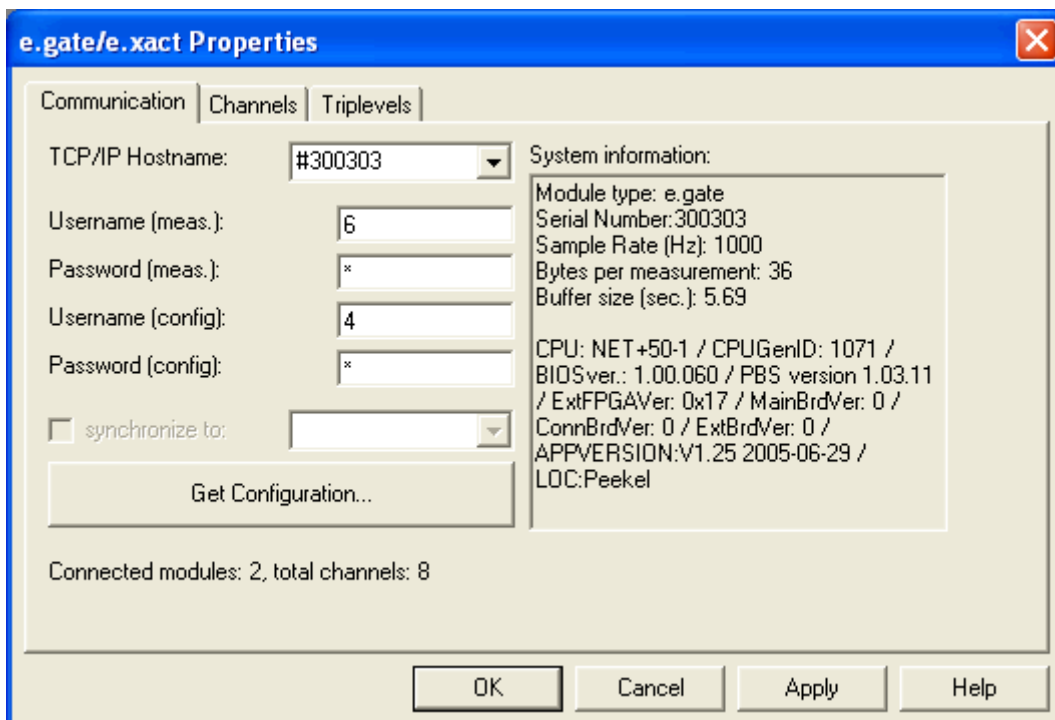
Timeout (secs.): Sets the timeout time in seconds, only for retriggerable trips.

5.5 Add e.gate/e.xact

Use this command to connect to a new e.gate or e.xact module. Communication with an e.gate takes place over a TCP/IP network connection. Use configuration documentation and tools supplied by Gantner to configure your e.gate and the modules that are connected to it. After that, SignaSoft 6000 will automatically show the configured channels.



5.5.1 Communication Settings



On this property page you can configure the TCP/IP connection and scan for available channels. You must first specify the TCP/IP address or hostname where the e.gate can be located and enter the correct username and password for configuration access and measurement access. Refer to your e.gate documentation for more information about configuring the e.gate TCP/IP settings, passwords and other parameters.

TCP/IP Hostname: Enter the TCP/IP address of the e.gate (example: '192.168.0.5'), or use a defined hostname if the network is configured to accept it (example: 'egate.peekel.nl').

Username (meas.): Sets the username used to connect to the e.gate in measurement mode. The default value '6' reflects the factory default setting in the e.gate.

Password (meas.): Sets the password used to connect to the e.gate in measurement mode. The default value '6' reflects the factory default setting in the e.gate.

Username (config.): Sets the username used to connect to the e.gate in configuration mode. The default value '4' reflects the factory default setting in the e.gate.

Password (config.): Sets the password used to connect to the e.gate in configuration mode. The default value '4' reflects the factory default setting in the e.gate.

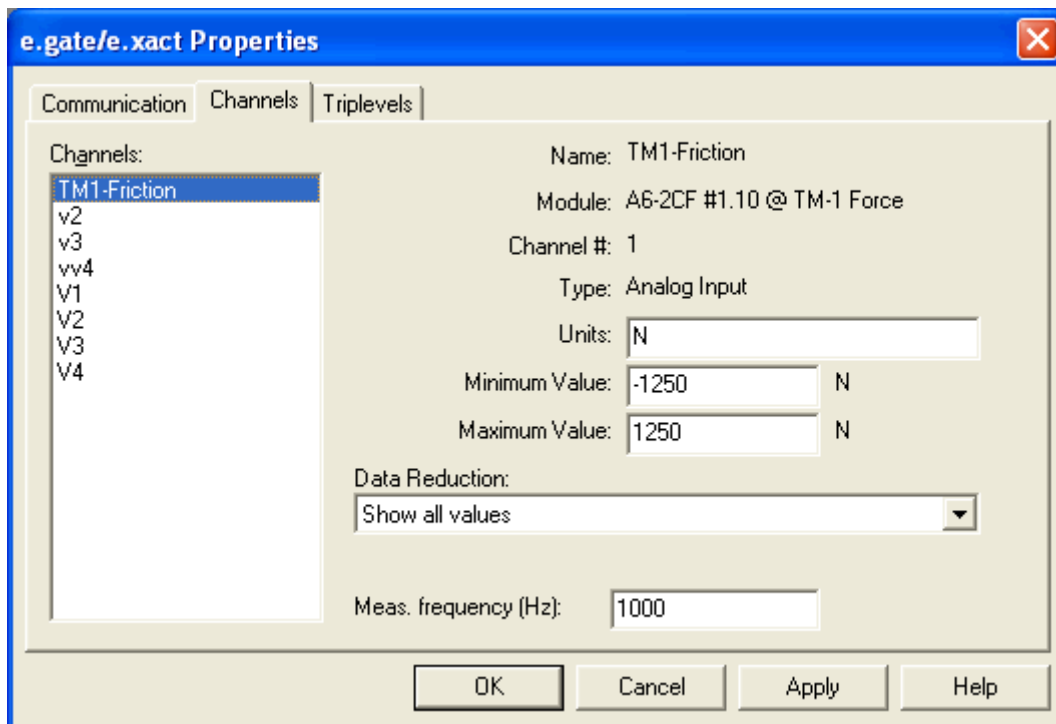
Synchronize to: Synchronizes this e.gate to another e.gate in the configuration. This option is only available if the e.gates are correctly configured.

Get Configuration...: After setting the correct TCP/IP hostname, usernames and passwords, this button will retrieve the current configuration from the e.gate. The process can take some time, the status line will keep you informed of progress.

Status line: The status line shows progress information while retrieving the configuration from an e.gate. Afterwards, it shows the number of modules connected to the e.gate, as well as the total number of channels configured in those modules.

System Information: This box shows system information on a connected e.gate. The most important information shown here is the overall sample rate at which the e.gate measures and the size of the internal buffer, measured in seconds of measurement data.

5.5.2 Channel Settings



On this property page you can view information about the available channels. Also, you can set the measurement interval for each channel.

Channels: This box shows the available channels. To configure a channel, first select it from the list. You can also select multiple channels if you want to configure the same measurement interval on all of those channels. If this box is empty, check the communication settings first.

Name: Shows the name of the channel.

Module: Shows the module that contains this channel. Also shows address (#UART.Address) and module location.

Channel #: Shows the channel number within the module.

Type: Shows the measurement type of the channel.

Units: Shows the units used to present measurement values.

Minimum Value: Shows the minimum acceptable value for the channel. This value is only used for presentation, not for range checking or limit warnings.

Maximum Value: Shows the maximum acceptable value for the channel. This value is only used for presentation, not for range checking or limit warnings.

Data Reduction: The e.gate retrieves measurement values for all channels at the same speed. The data reduction option allows you to change the data collection rate for individual channels. The options are:

Show all values: no data reduction, use the global e.gate sample rate.

Disable channel: show no measurement values for this channel at all.

Average over X values: build an average value over X measurement values.

Keep only 1 out of every X values: reduce sample rate by keeping only some of the measured values.

Minimum over X values: for each X values, keep only the minimum value measured.

Maximum over X values: for each X values, keep only the maximum value measured.

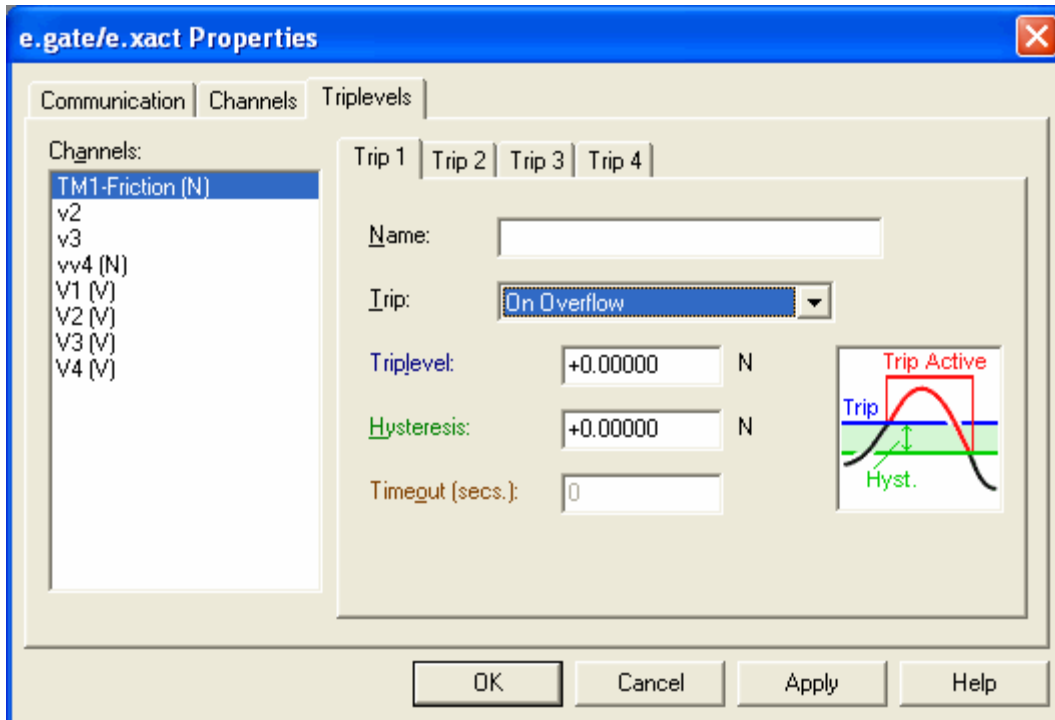
Keep after value difference of X or more: keep a new measurement value only if the difference with the previous kept value is more than X.

Keep after value diff. * seconds of X or more: keep a new measurement value only if the difference with the previous kept value, multiplied by the time interval in seconds, is more than X.

Value for X: Specify the value of X as described for data reduction.

Meas. frequency (Hz): Shows the requested measurement frequency for the channel in Hz. This value depends on the type of data reduction, and the value for X. You can also specify the preferred measurement frequency and have the software determine the appropriate data reduction needed.

5.5.3 Trips Settings



On this property page you can configure up to 4 separate trips for each of the available channels.

Channels: This box shows the available channels. To configure trips, first select a channel from the list. You can also select multiple channels if you want to configure the same trip on all of those channels. If this box is empty, check the communication settings first.

Trip Selection: There are 4 different trips available for each channel, which can be configured separately.

Name: Each trip can have its own name, which is freely configurable.

Type: Sets the type of trip, unused trips are marked 'disabled'.

'On overflow' trips will activate as soon as the signal exceeds the trip level and deactivate as soon as the signal drops below 'trip level - hysteresis'.

'Retriggerable overflow' trips will activate as soon as the signal exceeds the trip level and deactivate when the signal remains below 'trip level - hysteresis' for at least 'timeout' seconds.

Triplevel: Here you can specify the level at which the trip will activate.

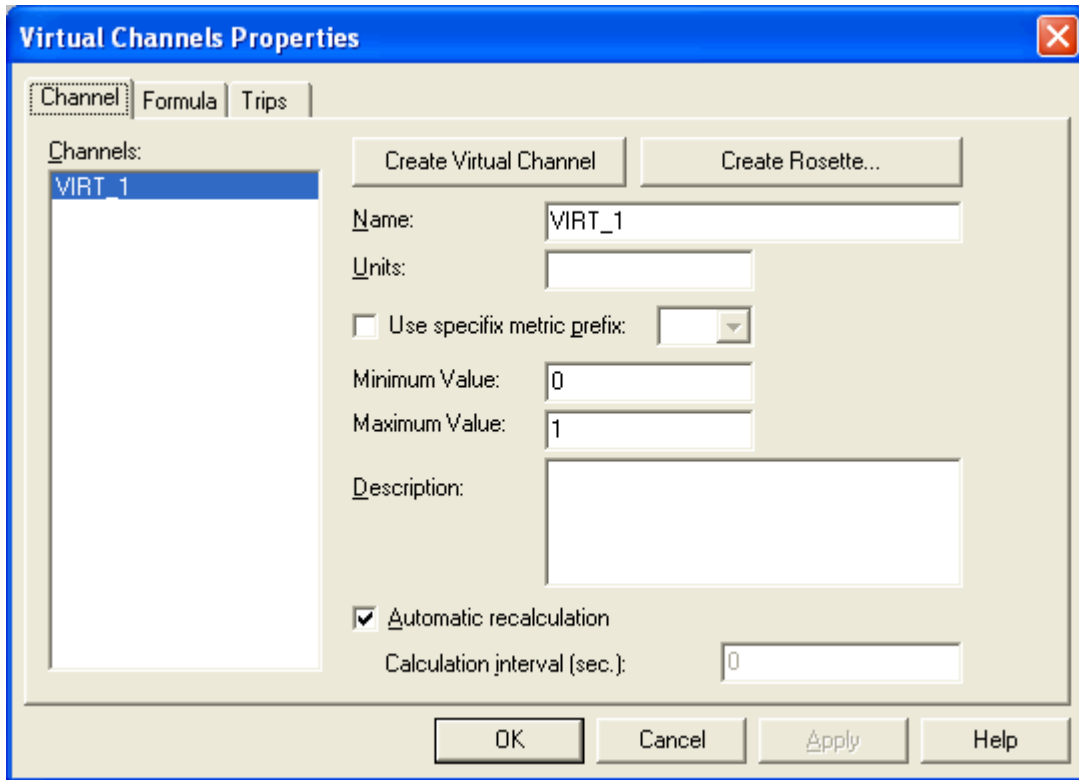
Hysteresis: Sets the hysteresis band (as an absolute value) around the trip level that determines when the trip should deactivate. It will be added to or subtracted from the trip level to find the level at which the trip will deactivate.

Timeout (secs.): Sets the timeout time in seconds, only for retriggerable trips.

5.6 Add Virtual Channel

Use this command to create a new virtual channel.

5.6.1 Channel Settings Property Page



On this property page you can view and configure general information on the available channels. Also, you can create new virtual channels from here.

Create New Channel: Press this button to create a new virtual channel.

Channels: This box shows the available channels. To configure a channel, first select it from the list. You can also select multiple channels if you want to configure some of their parameters identically. If this box is empty, create a new virtual channel first.

Name: Defines the name of the virtual channel, which uniquely identifies it.

Units: Defines the engineering units used to present values calculated by this channel.

Metric prefix: If you check 'use specific metric prefix', you can force the channel to present values using a specific prefix. For example, if your channel calculates results in V (Volts), but you want to present them as mV (millivolts), you can select the specific prefix 'm' here. If you uncheck the 'use specific metric prefix' option, values will be presented using the most suitable prefix at any given time.

Minimum Value: Shows the minimum acceptable value for the channel. This value is only used for presentation, not for range checking or limit warnings.

Maximum Value: Shows the maximum acceptable value for the channel. This value is only used for presentation, not for range checking or limit warnings.

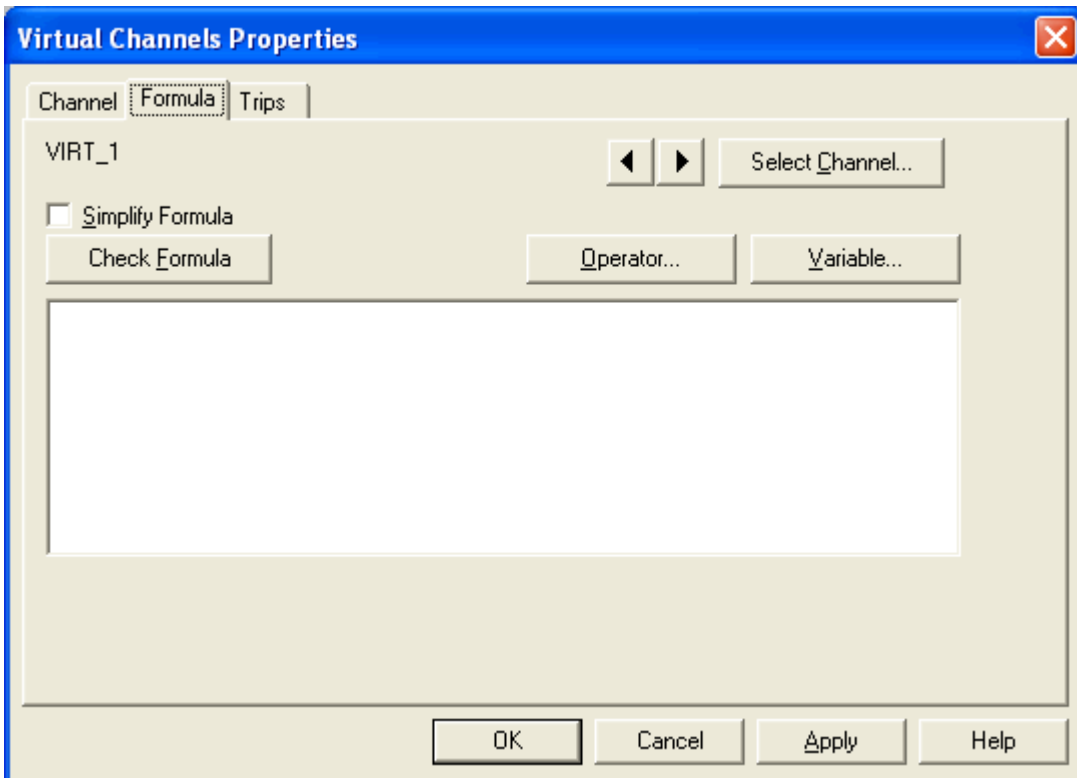
Description: Room for entering a descriptive text associated to a virtual channel.

Automatic Recalculation: When checked, the virtual channel will be recalculated as soon as one of the channels used in its formula receives a new measurement value.

When unchecked, the virtual channel will be recalculated on a time interval basis. This option is most useful for 'pure virtual' channels, that do not rely on physical measurement values as input for their calculations.

Calculation interval (sec.): Specifies the interval in seconds to use for recalculating a virtual channel that does not use 'automatic recalculation'.

5.6.2 Formula Property Page



On this property page you can configure the formula for a virtual channel.

Channel Selection: Here you can select the channel to configure.

Simplify Formula: Select this option to simplify the formula as far as possible. Constant expressions will be converted to numbers and extraneous brackets will be removed.

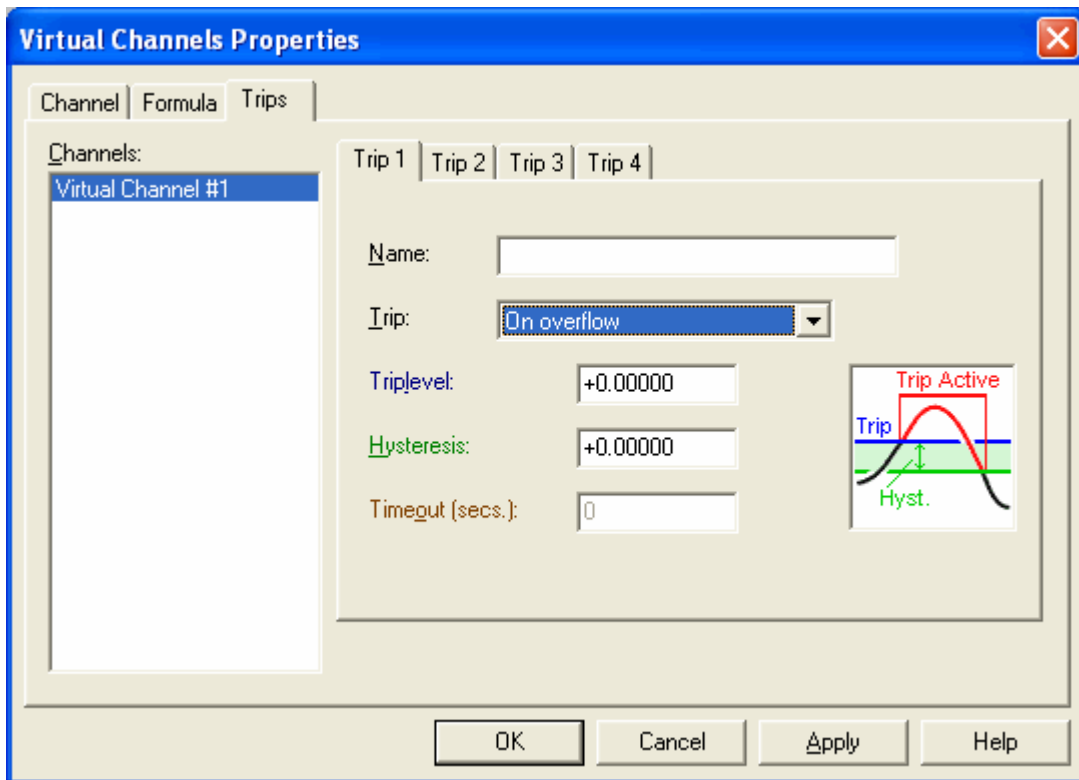
Check Formula: Press this button to check the formula for errors. If the formula is incorrect, problem will be described and highlighted.

Operator...: Shows a list of available operators that can be used in the formula.

Variable...: Shows a list of available channels that can be used in the formula.

Formula: Place to enter the formula. The text will be red when the formula is incorrect or incomplete.

5.6.3 Trip Levels Property Page



On this property page you can configure up to 4 separate trips for each of the available channels.

Channels: This box shows the available channels. To configure trips, first select a channel from the list. You can also select multiple channels if you want to configure the same trip on all of those channels. If this box is empty, check the communication settings first.

Trip Selection: There are 4 different trips available for each channel, which can be configured separately.

Name: Each trip can have its own name, which is freely configurable.

Type: Sets the type of trip, unused trips are marked 'disabled'.

'On overflow' trips will activate as soon as the signal exceeds the trip level and deactivate as soon as the signal drops below 'trip level - hysteresis'.

'Retriggerable overflow' trips will activate as soon as the signal exceeds the trip level and deactivate when the signal remains below 'trip level - hysteresis' for at least 'timeout' seconds.

Trip level: Here you can specify the level at which the trip will activate.

Hysteresis: Sets the hysteresis band (as an absolute value) around the trip level that determines when the trip should deactivate. It will be added to or subtracted from the trip level to find the level at which the trip will deactivate.

Timeout (secs.): Sets the timeout time in seconds, only for retriggerable trips.

5.7 Add Rosette Channel

Use this command to create a new rosette channel.

Rosette Configuration

Type:

S/G A:

S/G B:

S/G C:

Poisson (ref.): Poisson (spec.):

Kt: % E-Modulus: N/mm²

Output values:

Stress A	Unused
Stress B	Unused
Stress C	Unused
Corr. Strain eps. A	Unused
Corr. Strain eps. B	Unused
Corr. Strain eps. C	Unused
Major Strain eps. 1	Unused
Minor Strain eps. 2	Unused
Angle alpha	Unused
Major Stress sigma 1	SIG_MAJ
Minor Stress sigma 2	SIG_MIN
Stress 0° sigma x	Unused
Stress 90° sigma y	Unused
Shear Stress tau xy	Unused
Equivalent Stress sigma e	Unused

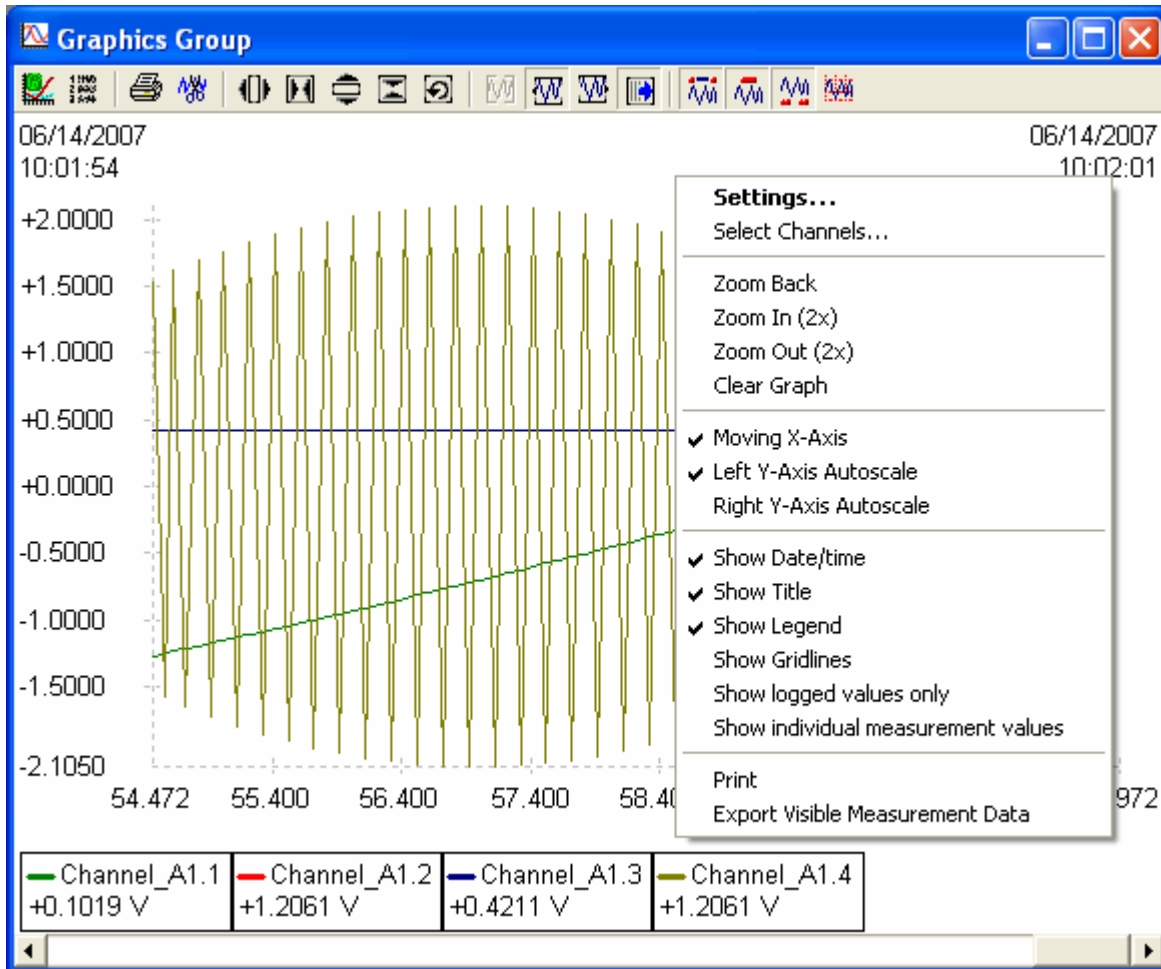
To configure a rosette, first select its **type** and the strain gauges to be used as inputs. Note that the input channel must be measured in 'm/m' units to be accepted as a strain gauge input.

You can set the **Poisson** parameters and **E-Modulus** (used for stress type outputs in N/mm²). The '**Kt**' is used to compensate for the transverse sensitivity of strain gauges.

For each of the available output types, you can select whether or not to actually create a visible output. Each output is actually a virtual channel using a special formula to calculate the requested value.

5.8 New Graphics Group

Use this command to create a new Graphics Group.



The graphics group shows measured data for a selection of channels in graphical form. In the legend, measurement values shown in red are invalid, either because they have not actually been measured or because the value is out of range.

When showing on-line measurement values, the graphics control will show at most 100 new values per second. For fast measurements (> 100 Hz per channel) this means that not all measurement values may be shown on-line. During historical analysis (by scrolling back in time), the graphics control will retrieve archived data from disk. At that point values that may have been ignored during on-line *will* be shown.

You can configure the contents and presentation of the graphics using the button bar at the top of the control.

An alternative way to determine which channels are shown and how the graphical data is formatted is clicking the right mouse button anywhere inside the window to show to context menu. The context menu contains the following options:

















- | | |
|--------------------|---|
| Settings... | Show the settings for the selected graphics group. |
| Select Channels... | Select which channels should be displayed in the graphics group. |
| Zoom Back | Go back to the previous zoom level, after zooming in or out. |
| Zoom In (2x) | Zoom in to the middle of the graphics. You can also click and drag in the graphics view to select a specific zoom area. |
| Zoom Out (2x) | Zoom out to show surrounding data. |
| Clear Graph | Remove all current measurement values from the graphics buffer, which clears the graph. |
| Moving X-Axis | Selects whether X-axis should automatically move in time to make sure the latest measurement values are visible. |

Left Y-Axis Autoscale	Select automatic scaling of the left y-axis to make all data visible.
Right Y-Axis Autoscale	Select automatic scaling of the right y-axis to make all data visible.
Show Date/Time	Option to show the date and time of start and end of graph on top.
Show Title	Option to show the title of the graph. The title can be entered freely under settings.
Show Legend	Option to show a legend under the graph.
Show Gridlines	Option to show gridlines in the graph.
Show logged values only	Option to hide measurement values, which are not specifically requested in the measurement logging configuration.
Print	Immediately print the contents of the graph to the default printer.
Export Visible Measurement Data	Allows you to export measurement data from the archives using the currently visible time range.

5.8.1 Graphic Button Bar

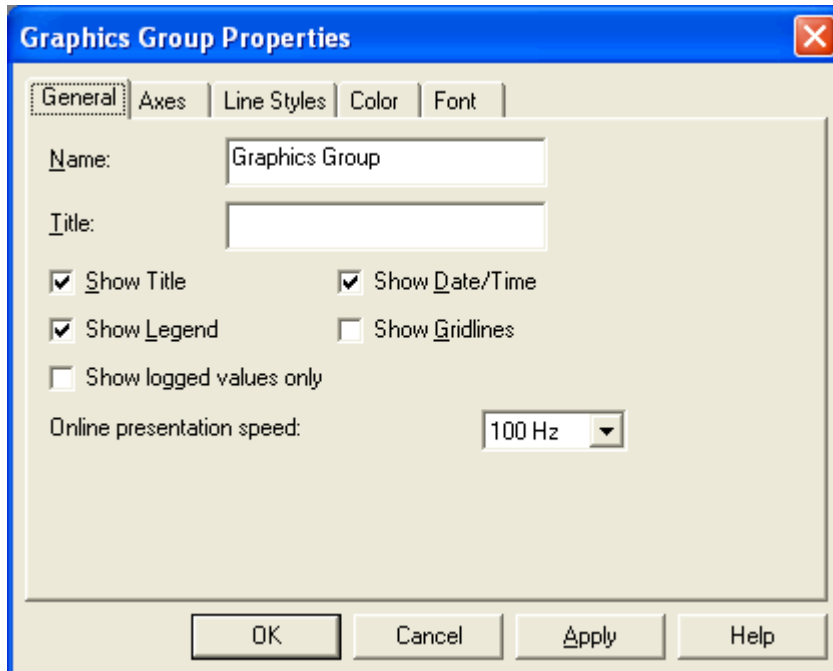


The button bar on top of the graphics control has the following buttons:

-  Settings: show the settings for the graphics group.
-  Select Channels: select which channels should be displayed in the graphics group.
-  Print: Make a hardcopy of the current display of the graphics group.
-  Clear Graph: Remove all measurement data from the graph and its history.
-  Zoom Out (2x): Show twice as much data in the horizontal direction.
-  Zoom In (2x): Show half the amount of data in the horizontal direction.
-  Zoom Out (2x): Show twice as much data in the vertical direction.
-  Zoom In (2x): Show half the amount of data in the vertical direction.
-  Zoom Back: Return to the previous zoom level.
-  X-Axis Autoscale: Scale X-axis to fit the data (for XY-plots).
-  Left Y-Axis Autoscale: Scale left Y-axis to fit the data.
-  Right Y-Axis Autoscale: Scale right Y-axis to fit the data.
-  Moving X-Axis: Automatically scroll the graph to make sure the most recent data is visible.
-  Show Date/Time: Show date and time at the top of the graph.
-  Show Title: Show title at the top of the graph.
-  Show Legend: Show legend at the bottom of the graph.

 Show Gridlines: Show gridlines in the graph.

5.8.2 Graphic Settings



In the general settings for a graphics control, the following items are available:

Name:

The name of the graphics control, used to identify it.

Title:

Title of the graphics, shown in the graphics output.

Show Title:

Determines whether the graphics title is shown.

Show Legend:

Determines whether the legend is shown.

Show Date/time:

Determines whether date and time is shown.

Show Gridlines:

Determines whether gridlines are shown.

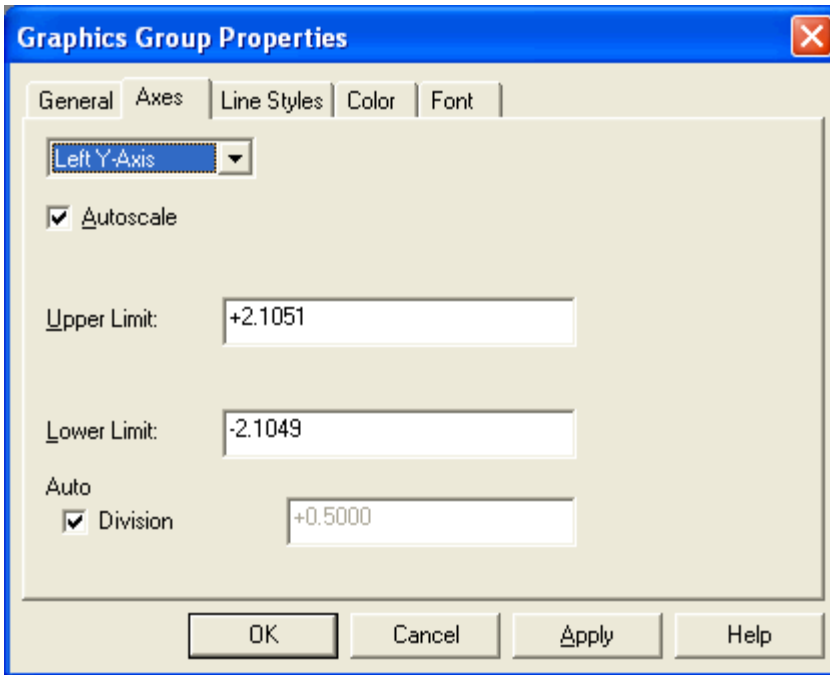
Show logged values only:

Determines whether or not to hide measurement values, which are not specifically requested in the measurement logging configuration.

Online presentation speed:

Determines how many measurement values per second are shown in the graphics group during online presentation.

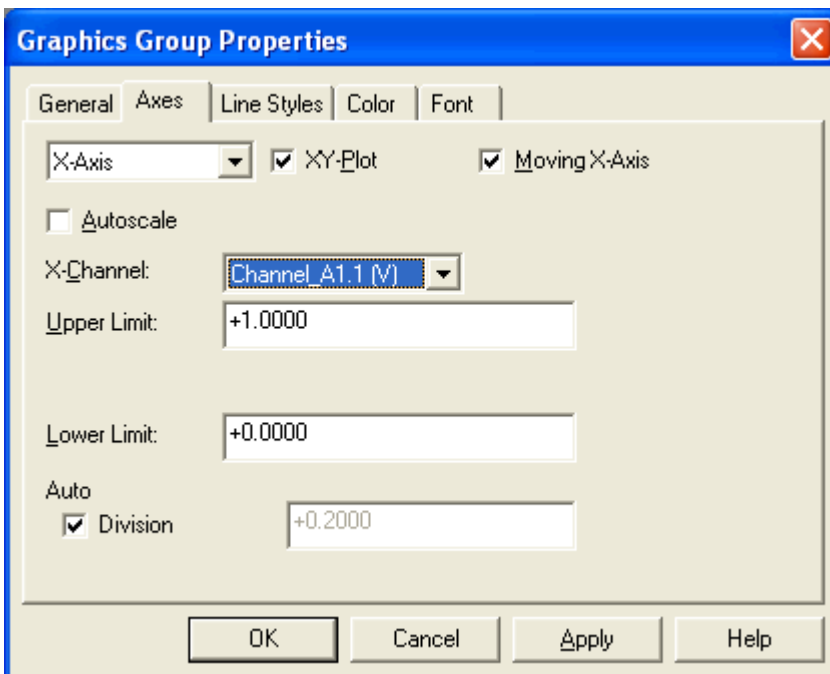
5.8.3 Graphic Axes



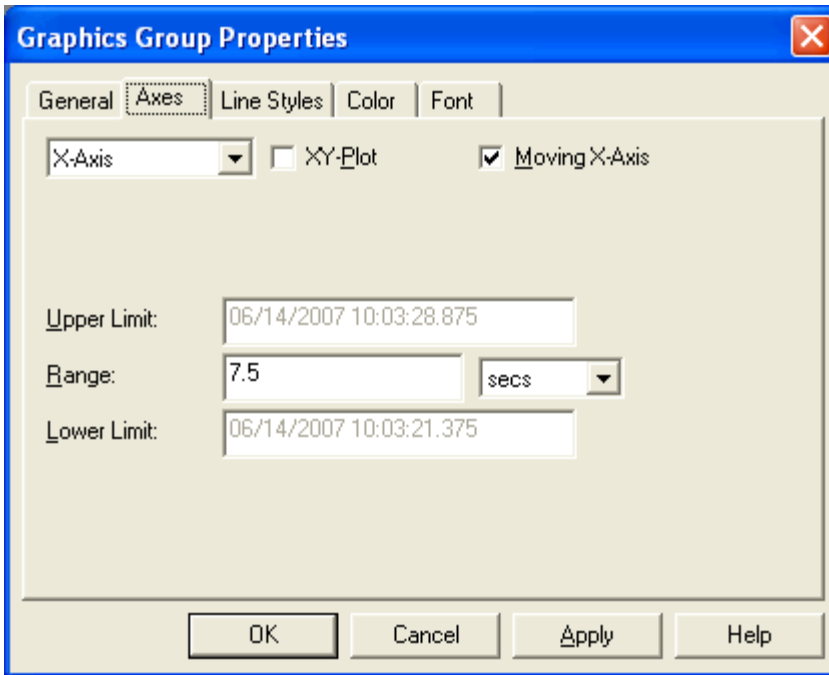
In this window, you can set the limits for the X-axis and left and right Y-axes.

For the left and right Y-axes, you can set the upper and lower limit. If you check the '**Autoscale**' box, these values are ignored and the limits are calculated to make the measurement data fit.

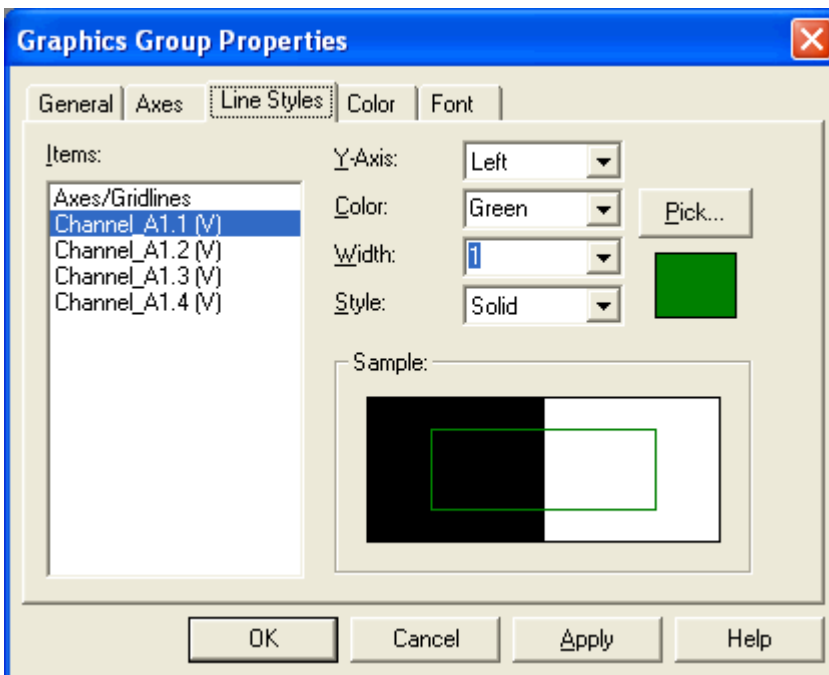
If you check '**XY-plot**', you can choose a channel to be used for the X-axis. Upper and lower limit for the X-axis can be set just like those for the Y-axes.



If you leave '**XY-plot**' unchecked, a time-range can be entered for the x-axis. If the '**Moving X-Axis**', you can also determine the start of the presentation as an absolute date and time.



5.8.4 Graphic Line Style



In the line style settings for a graphics control, you can select one or more channels on the left side of the window. To select several channels, press and hold the <ctrl> key, then select individual channels using the left mouse button, or press and hold the <shift> key, then select a range of channels using the left mouse button. For the selected channels, the following items can be set:

Y-Axis:

Choose left or right axis, or 'none/hide' to hide this channel in the output.

Color:

Choose the color for the line. You can either pick one of the standard colors from the list, or click on the 'Pick' button to show a color selection window.

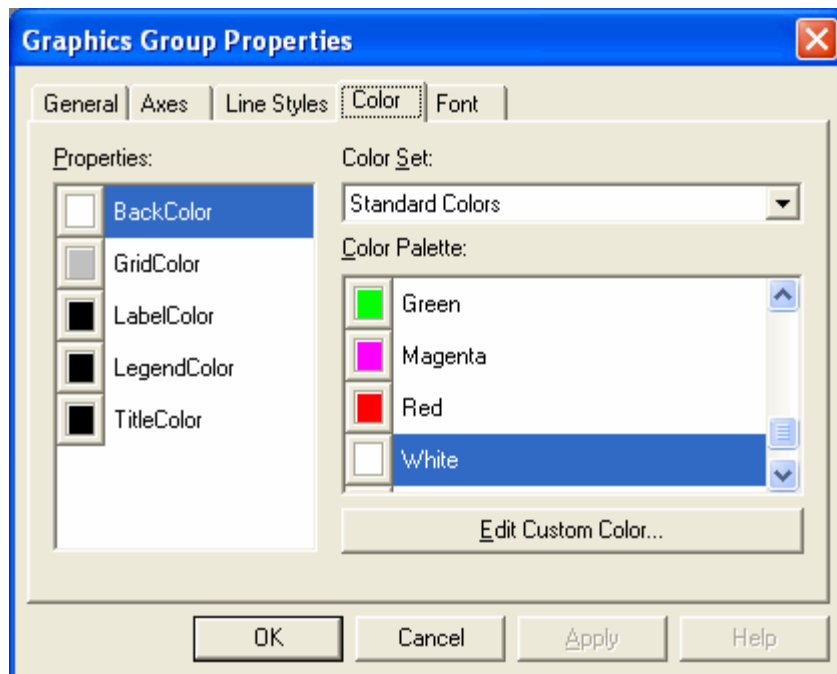
Width:

Choose the thickness of the line. If you select a width greater than 1, you can no longer select a style.

Style:

Choose the style of the line, if it has width 1. You can choose for solid, dotted or dashed lines.

5.8.5 Graphic Colors



In this window, you can choose the color for several parts of the graphics output. The following colors can be changed here:

BackColor: the background color of the graphics output.

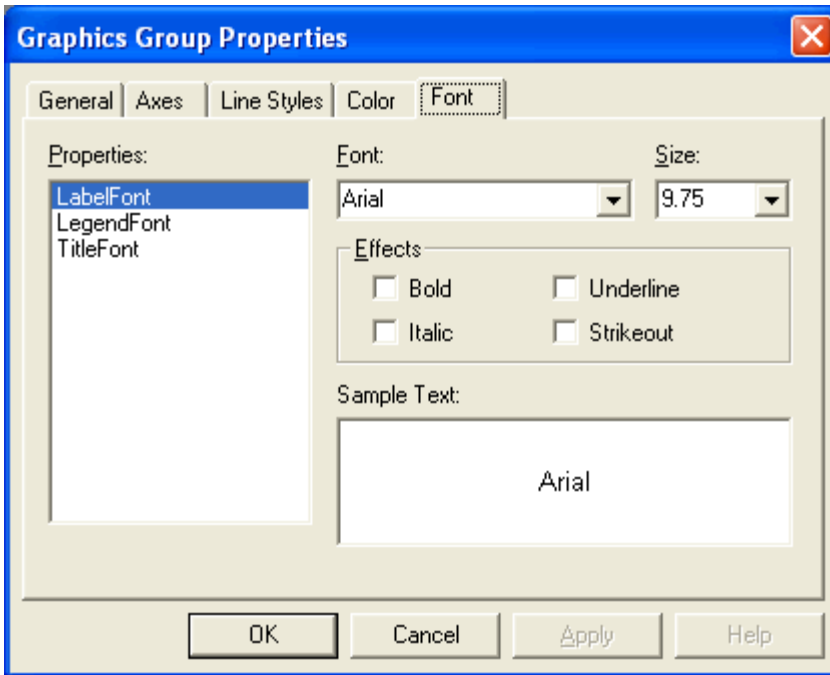
GridColor: the color of the gridlines.

LabelColor: the color of the labels printed next to the axes.

LegendColor: the color of the legend.

TitleColor: the color of the title text.

5.8.6 Graphic Font

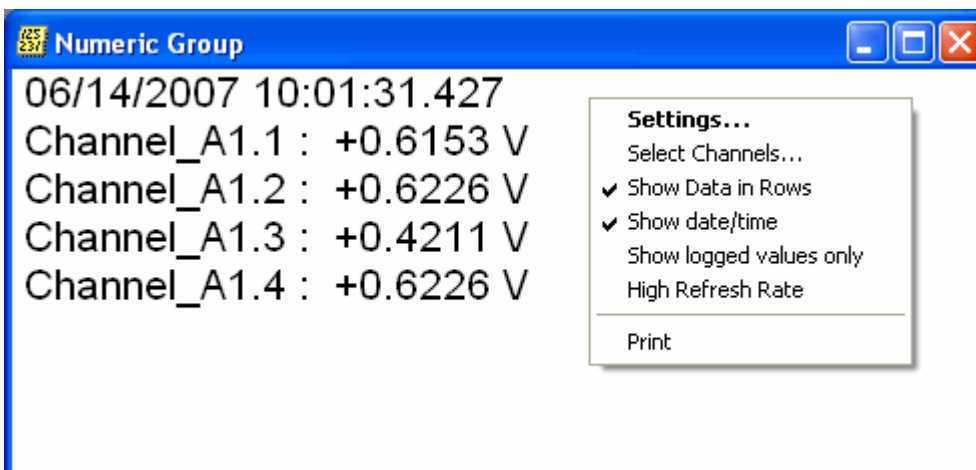


In this window, you can choose the font for several parts of the graphics output. The following fonts can be changed here:

- LabelFont:** the font of the labels printed next to the axes.
- LegendFont:** the font of the legend.
- TitleFont:** the font of the title text.

5.9 New Numeric Group

Use this command to create a new Numerics Group.



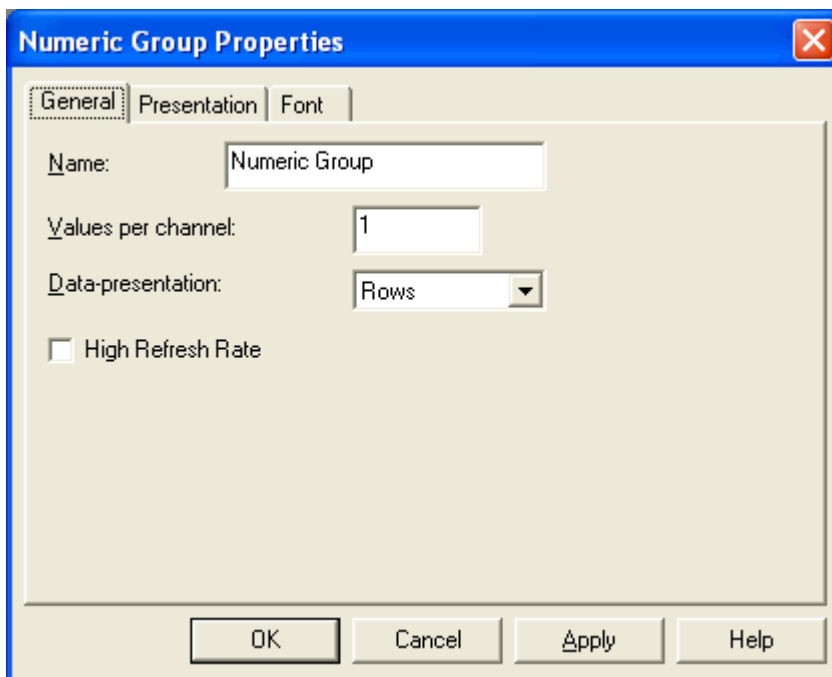
The numerics group shows measured data for a selection of channels in numerical form. The name of each channel is shown, followed by one or more measurement values including the units. Measurement values shown in red are invalid, either because they have not actually been measured or because the value is out of range.

The header line always shows the date and time of the latest measurement value received for any channel in the numeric control.

To determine which channels are shown and how the numerical data is formatted, click the right mouse button anywhere inside the window to show to context menu. The context menu contains the following options:

Settings...	Show the settings for the selected numerics group.
Select Channels...	Select which channels should be displayed in the numerics group.
Show Data in Rows	Option to show the measurement values in a row next to the channel name. By default, they are shown in a column under the channel name.
Print	Immediately print the contents of the numerics view to the default printer.

5.9.1 Numeric Settings



In the general settings for a numerics control, the following items are available:

Name:

The name of the numeric control, used to identify it.

Values per channel:

Determines how many measured values are displayed per channel.

Data-presentation:

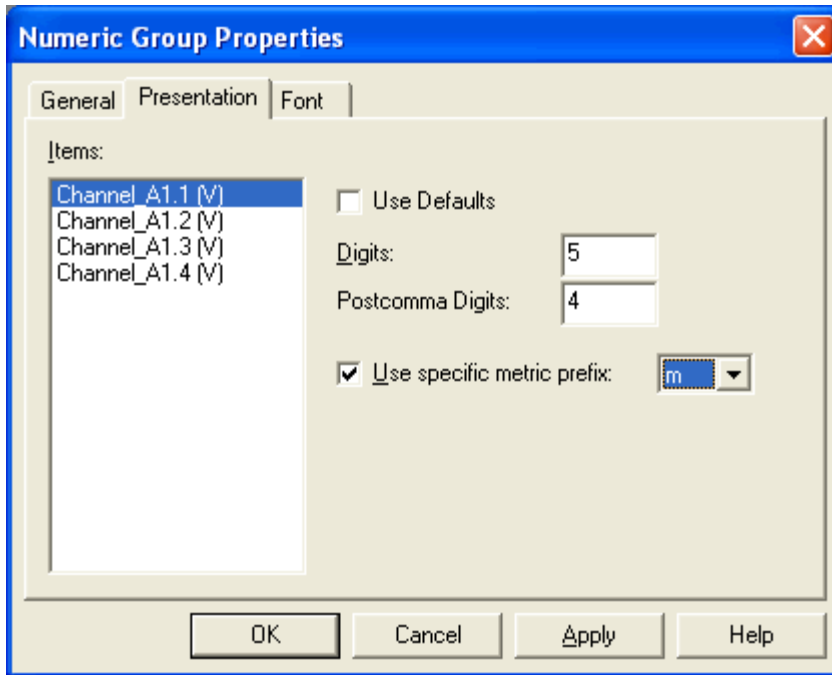
If data is presented in rows, the values for each channel are shown in a left-to-right direction, with the most recent measurement value on the right side.

If data is presented in columns, the values for each channel are shown under its name, in a top-to-bottom direction, with the most recent measurement value on top.

High Refresh Rate:

Check this box to update the numeric display at the maximum possible speed. Normally, it is only refresh 1-2 times per second to show a more stable display.

5.9.2 Numeric Presentation



In the presentation settings for a numerics control, you can select one or more channels on the left side of the window. To select several channels, press and hold the <ctrl> key, then select individual channels using the left mouse button, or press and hold the <shift> key, then select a range of channels using the left mouse button. For the selected channels, the following items can be set:

Use Defaults:

If selected, the default settings from the channel are used, otherwise, specific settings can be specified for this numeric group.

Digits:

Determines the total number of digits shown for the measurement value. If the value cannot be displayed using the specified amount of digits, it will be indicated by '###' signs.

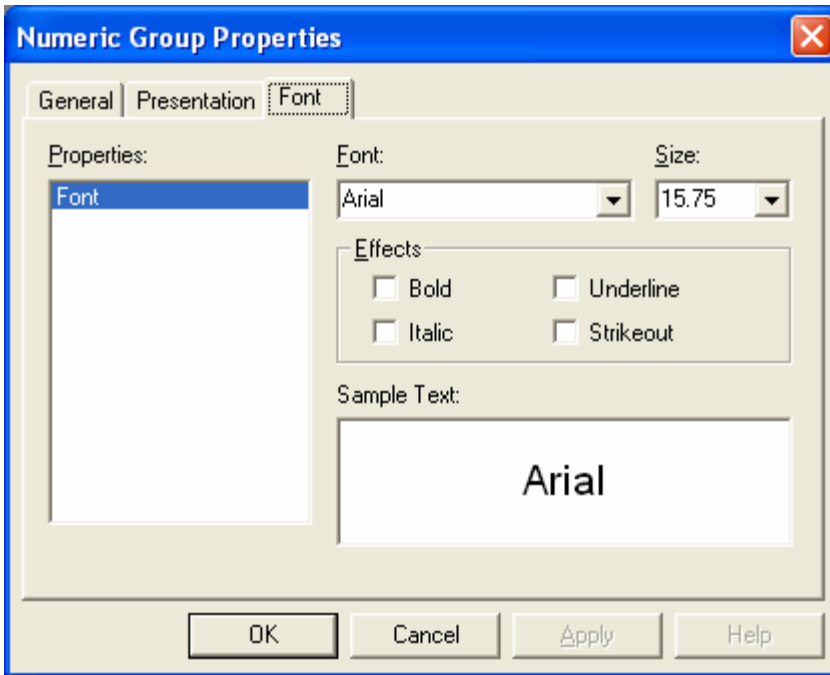
Postcomma Digits:

Determines the number of digits shown after the decimal point/comma.

Use specific metric prefix:

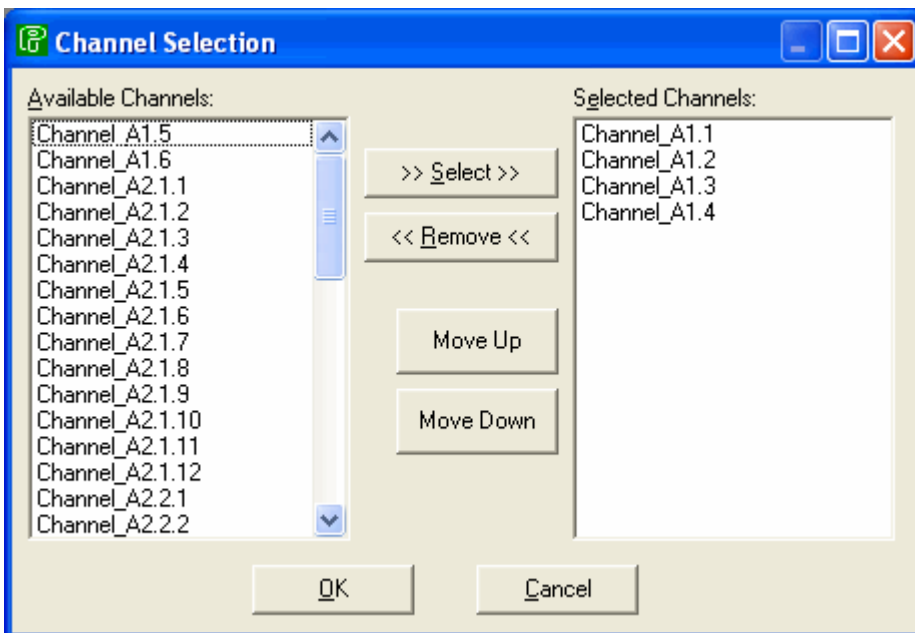
If this box is unchecked, the metric prefix (k, M, G, m, etc.) will automatically be selected to best match the current measurement value. If the box is checked, you can select a metric prefix to be used regardless of the measurement value.

5.9.3 Numeric Font



In the font selection box for a numerics control you can select the font used to present the numeric output on screen and on printer.

5.9.4 Channel Selection



To add channels to a group, first select one or more channels from the list of available channels on the left side of the window.

To select several channels, press and hold the <ctrl> key, then select individual channels using the left mouse button, or press and hold the <shift> key, then select a range of channels using the left mouse button.

Press the **'Select'** button to add the selected channels to the group.

To remove channels from the group, first select them from the list of selected channels, then press the **'Remove'** button to remove them from the group.

5.9.5 Print Function

When you select 'Print' from the right mouse button menu (context menu) of any control, the contents of the control are printed immediately on the default printer using the default settings. Data will be formatted to make best use of the available space on paper.

6. Measurement Menu Commands

The Measurement menu offers the following commands:

Logging Rules	Shows a dialog in which you can configure the rules that determine which measured channels should be archived and when archiving should start or stop.
Log Extraction	Allows you to convert archived data to a format suitable for further processing (ASCII text file). You can make a selection of channels and specify a start and stop time for the extraction.
Clean Archives	Allows you to remove data you no longer need from the archive directory, to save disk space. You can make a selection of channels and specify a time interval wherein data can be erased. Because of the archive structure, the software can not guarantee that all data within this time interval will actually be removed.
Name Manual Log...	Allows you to supply a name for manual log data. When you start and stop logging manually, this name will be given to the block of data archived during this period. Afterwards, you can easily access this data by using the 'Retrieve Manual Log' command.
Retrieve Manual Log...	Allows you to retrieve data from a specific time range. When you supply a meaningful name for all manual log actions, this function can later supply you with a list of archived data. From this list, you can select the data to retrieve. You can show this data in an existing graphics group or export it from the archives to a data file for offline processing.
Perform Loadstep	Perform datalogging in Loadstep mode. Details are described below.
Start Logging	Starts datalogging for all log groups that have a 'manual activation' rule, as defined using the Logging Rules command. If no groups or rules are defined, the program will create a temporary group containing all channels. This is the easiest way to manually start and stop archiving of all measured data.
Stop Logging	Stop datalogging for all log groups that have a 'manual activation' rule, as defined using the Logging Rules command. If a temporary group was created, it will be removed at this time.

6.1 Perform Loadstep

The 'Perform Loadstep' command start datalogging in the same way 'Start Logging' does. The difference is that a loadstep measurement will automatically stop as soon as a single measurement value is received and stored for each of the measured channels.

When performing loadstep measurements, the software will only ask for a manual log name at the first loadstep. All following loadsteps are considered to belong to the same measurement series. To complete a loadstep measurement series, do one of the following:

- 'Name Manual Log' to designate a new name for the log data that follows.
- 'Retrieve Manual Log' to convert the loadstep information to an external format.
- Close the configuration.

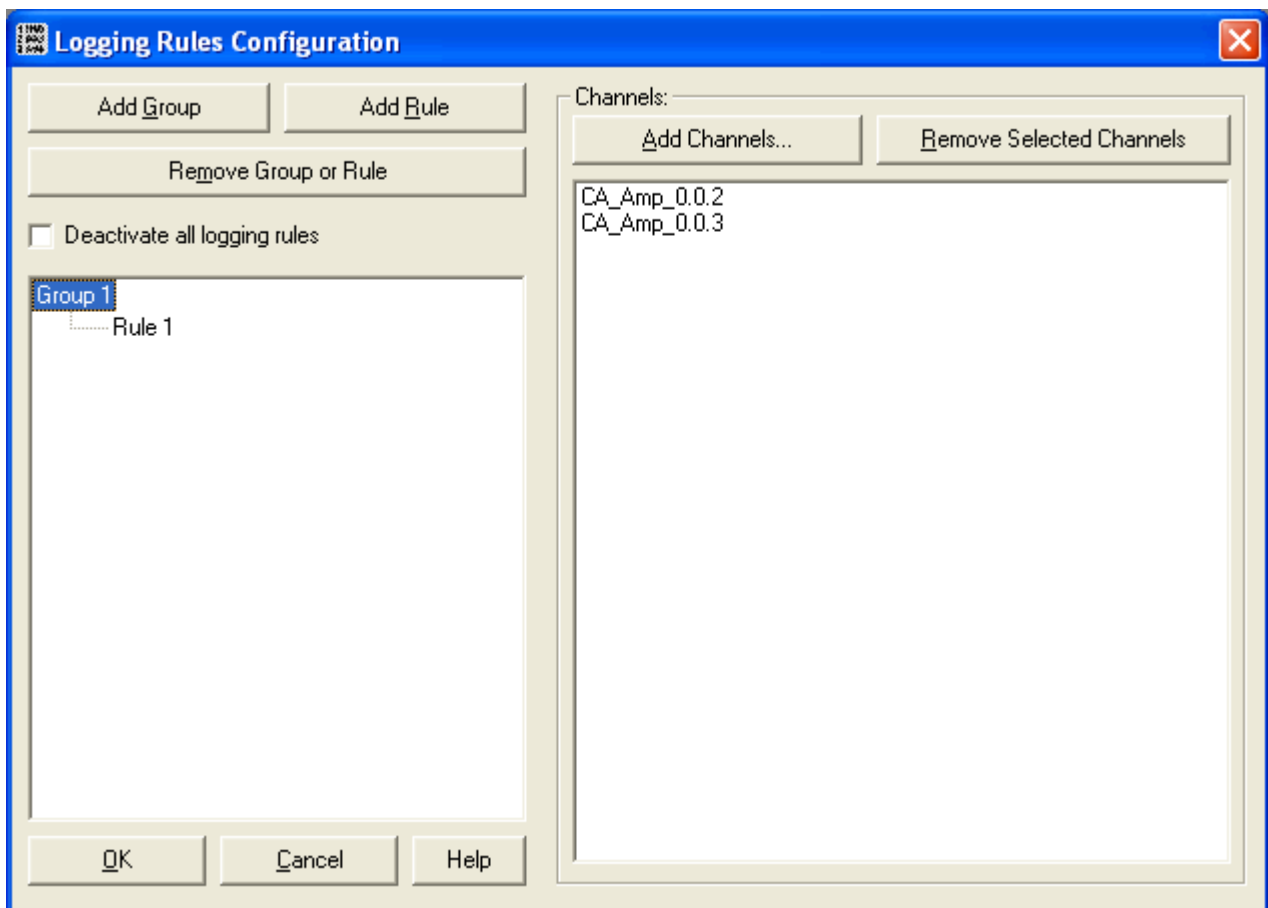
Each of these actions marks the end of a named collection of loadstep measurements.

6.2 Logging Group Configuration

The logging rules configuration dialog allows you to create a set of rules which dictate when measurement data for any channel should be stored to disk. To make a set of rules, you must first create a logging group and assign one or more channels to this group.

Then you can add one or more rules to the logging group. A rule determines when logging should be active, based on a time interval, trip activation, or manual activation. If multiple rules are assigned to the same group, the group is active as soon as one or more rules say that it should be active.

It is possible to create several logging groups, each with its own set of rules. It is not possible to assign a channel to more than one group.



The individual items in this configuration page described:

Add Group: Use this button to add a new logging group to the configuration, up to a maximum of 10.

Add Rule: Use this button to add a new rule to the selected group. A maximum of 30 rules can be divided among the logging groups.

Remove Group or Rule: Removes the currently selected group or rule from the configuration. If you remove a group, all rules assigned to this group are also removed.

Group and rule tree: The tree on the left side of the dialog shows the available groups and rules. When you click on a group or rule, the right side of the dialog shows the settings of the item.

OK button: Press this button to store the new configuration.

Cancel button: Press this button to leave the dialog without saving changes. All modifications to groups and rules are discarded.

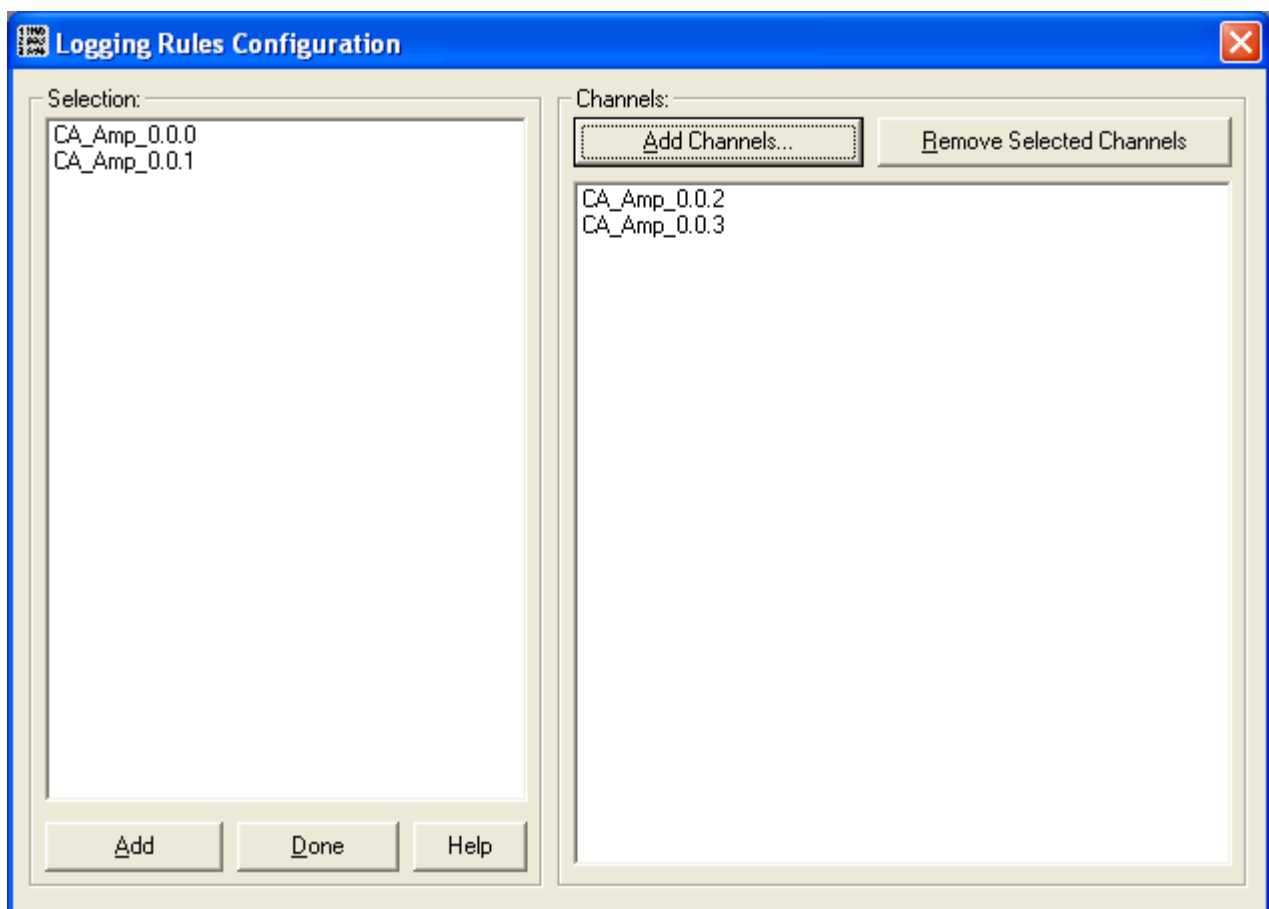
Channel list: This list shows all channels assigned to the currently selected logging group. Press 'Add Channels...' to add new channels, or select one or more channels and press 'Remove Selected Channels' to remove them.

Add Channels: When you press this button, the list of available channels will appear on the left side of the dialog. From this list, you can select one or more channels to add to the selected group.

Remove Selected Channels: After selecting one or more channels from the list, press this button to remove them from the currently selected group.

6.2.1 Channel Selection

When you press the 'Add Channels' button, the left side of the dialog will show a list of available channels. The list will not show any channels that are already assigned to a group. Press the 'Done' button to close the channel selection list.



The individual items in this configuration page described:

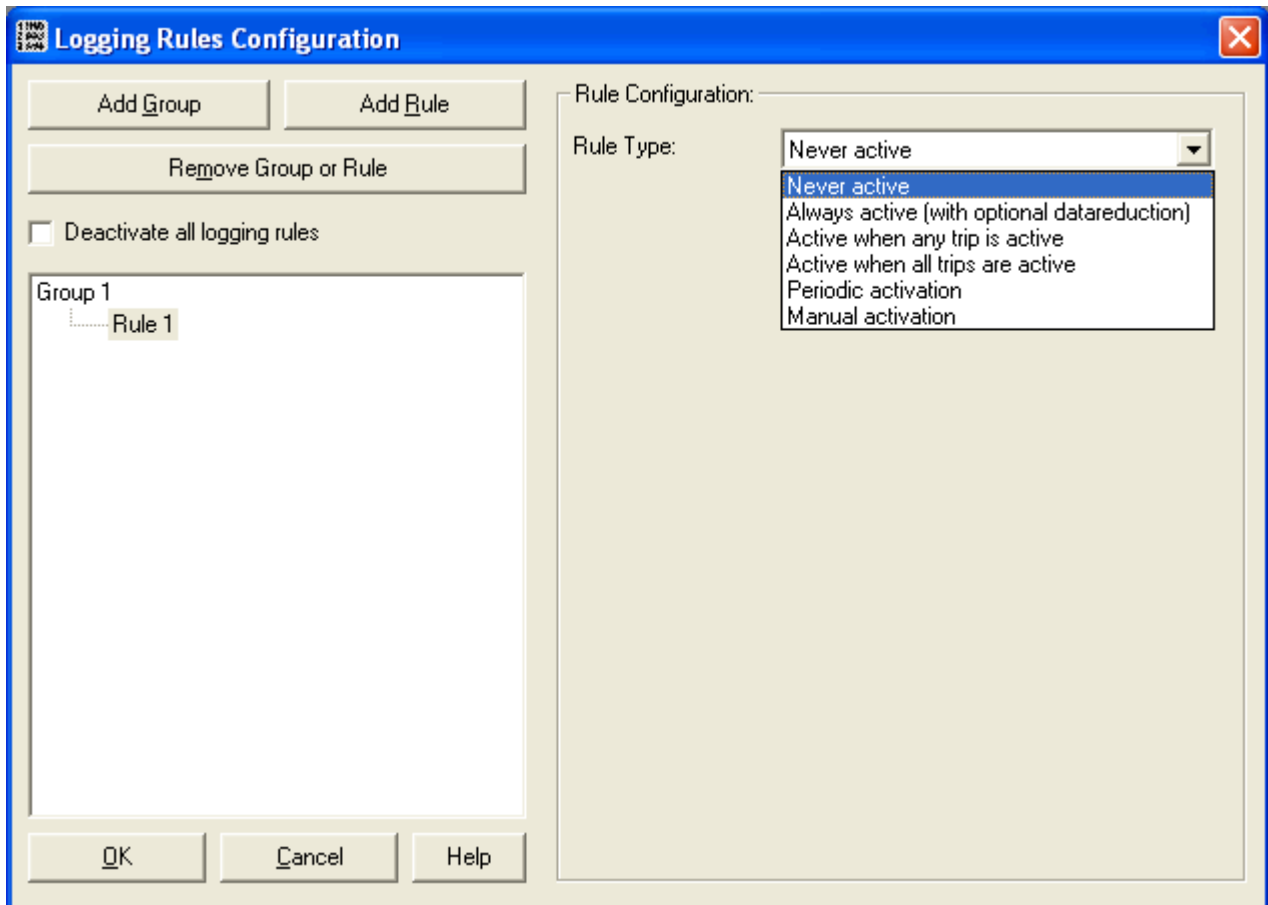
Item selection list: Shows available items. Click and drag to select one or more items. You can then drag them to the list on the right, or use the 'Add' button to add them manually.

Add: Click this button to add the currently selected items to the group or rule.

Done: Click this button to close the selection list, to be able to work on other groups or rules.

6.2.2 Rule Configuration

For each logging group, you can create one or more rules which determine when the group should be active.



Rule type: For each rule you assign to a group, you can select the type. Rules can be activated based on trips, time intervals or manual activation.

'Never' rule: If you set a rule to 'never', it is effectively disabled. The group will act as if the rule is not there.

'Always' rule: A rule set to 'always' is always active. Any other rules assigned to the same group will have no influence, because the group is active when at least one rule is active.

'Active when any trip active' rule: This rule is active when one or more trips are active. The trips to be considered must be added to the trip list of the rule.

'Active when all trips active' rule: This rule is active when all trips that are assigned to this rule are active.

'Periodic' rule: This rule is activated using specified time intervals.

'Manual' rule: This rule can be activated manually.

6.2.3 Periodic Rule

A periodic rule is activated using specified time intervals.

Loadstep: Check this to deactivate the rule after a single measurement value is stored. This is that during each period of activation, only the first measured value gets stored. Set the 'duration' long enough to make sure that at least one value will be measured in that time period, otherwise no value may be stored at all.

Interval: The rule will become active at the start of each interval.

Duration: The rule will remain active for the specified duration after the start of each interval. If 'duration' equals 'interval', then the rule will remain active from start to stop date/time without interruptions.

Start date/time: The rule will never become active before the specified start date/time. The first activation (start of the interval) is determined by the start date/time.

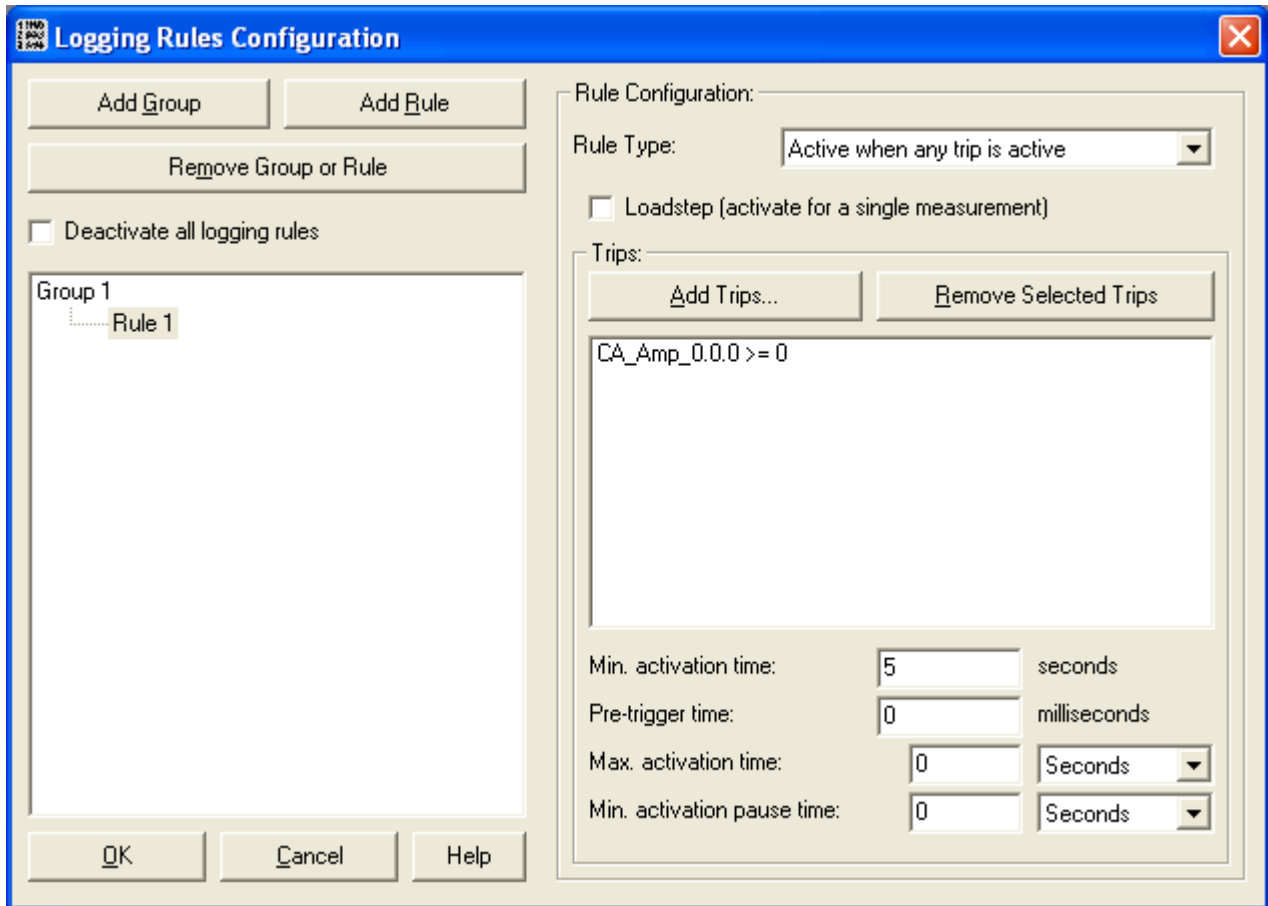
Stop date/time: The rule will never become active after the specified stop date/time.

6.2.4 Trip-based Rule

A trip-based rule is activated depending on trip activation. There are two types of trip-based activation:

'Active when any trip active' rule: This rule is active when one or more trips are active. The trips to be considered must be added to the trip list of the rule.

'Active when all trips active' rule: This rule is active when all trips that are assigned to this rule are active.



The individual items in this configuration page described:

Trip list: This list shows all trips assigned to the currently selected rule. Press 'Add Trips...' to add new trips, or select one or more trips and press 'Remove Selected Trips' to remove them.

Min. activation time: Specifies the minimum amount of time the rule should remain active after one (any trip active rule) or all (all trips active rule) trips become active.

Pre-trigger time: Using this field, for which a maximum of 2000 milliseconds is allowed, you can activate the rule before the trip becomes active. This allows you to store a small amount of measurement data measured before a certain event causes a trip to trigger.

Max. activation time: Use this field in combination with the 'Min. activation pause time' field to set a limit on the duration of the activation of the selected rule. If the rule activates during a long time, this option will force the rule to become inactive for at least 'min. activation pause time' before starting again. Leave the default value 0 to make sure the rule is always active whenever a any or all of the trips are active.

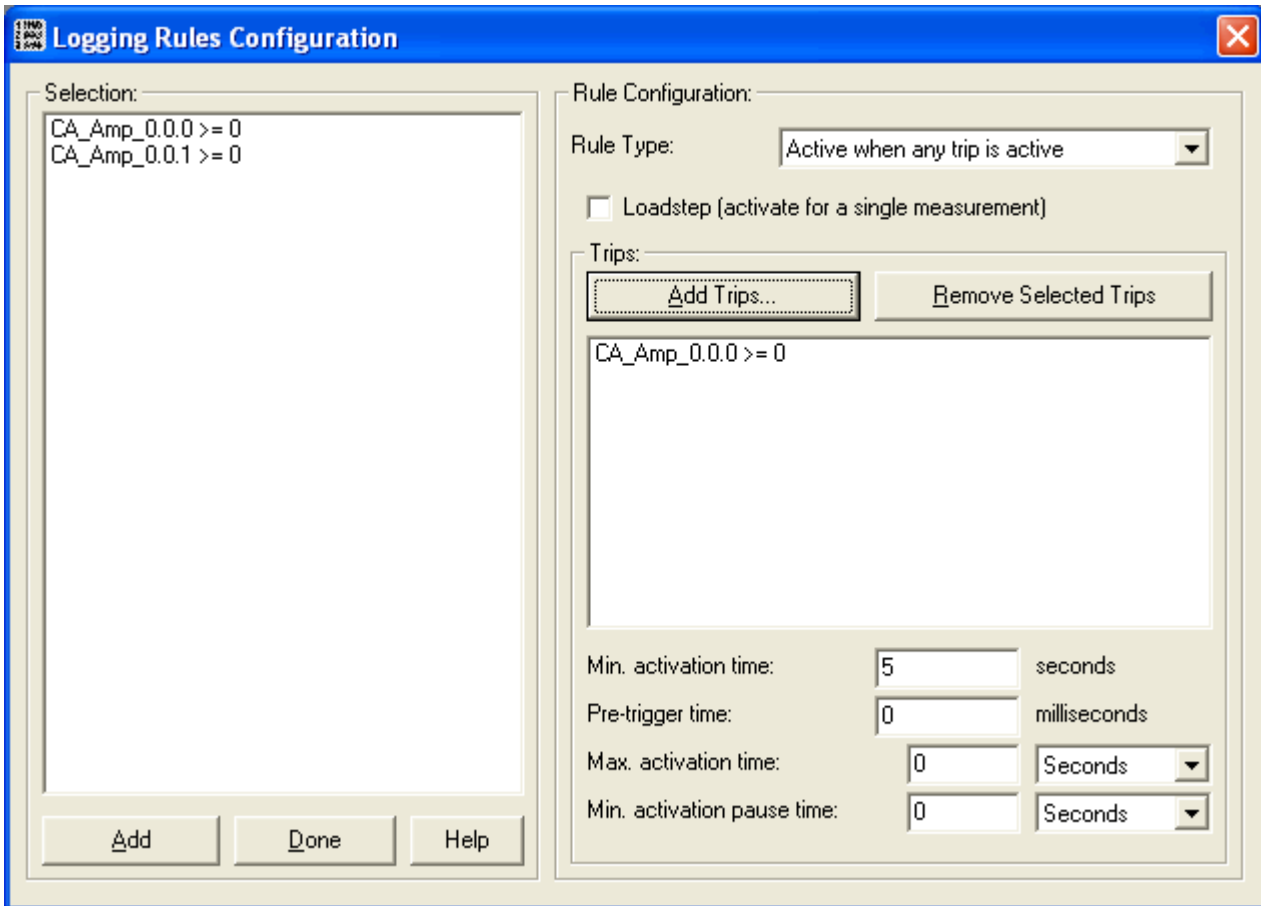
Min. activation pause time: Use this field in combination with the 'Max. activation time' field to set a minimum amount of time wherein the rule must remain inactive after the 'max. activation duration' time has elapsed. Leave the default value 0 to make sure the rule is always active whenever a any or all of the trips are active.

Add Trips: When you press this button, the list of available trips will appear on the left side of the dialog. From this list, you can select one or more trips to add to the selected rule.

Remove Selected Trips: After selecting one or more trips from the list, press this button to remove them from the currently selected rule.

6.2.5 Trip Selection

When you press the 'Add Trips' button, the left side of the dialog will show a list of available trips. Press the 'Done' button to close the trip selection list.



The individual items in this configuration page described:

Trip selection list: Shows available trips that are not yet assigned to a group. Click and drag to select one or more trips. You can then drag them to the trip list on the right, or use the 'Add' button to add them to the current rule.

Add: Click this button to add the currently selected trips to the rule.

Done: Click this button to close the trip selection list, to be able to work on other groups or rules.

6.3 Measurement Export Configuration

The extraction dialog allows you to retrieve data from the measurement archive. You can combine any selection of channels, specify a time interval and choose the file format for the output.

The output file will contain rows of measurement values. Each row has a date/time stamp and a measurement value for each channel. The measurement values on a row may not have been measured at the exact same time. Therefore, the date/time stamp indicates the first measured value on the row. All other values on the same row were measured between this time and the time of the next row. If, for a certain channel, no measurement value is available within the interval, the field for that channel will be empty.

The time interval between rows is automatically determined based on the amount of measurement values available for the selected channels. It is selected in such a way that all measurement values are shown using a minimum amount of rows.

The individual items in this configuration page described:

Select Channels: Use this button to make a selection of channels.

Start date/time: Sets the start of the time interval to use.

Stop date/time: Sets the end of the time interval to use.

Storage file: This field contains the name of the output file. When the output file exists, a warning is displayed. If you continue, the existing file will be overwritten.

Output format: Selects the format in which data will be stored.

The **ASCII** file format is a plain text format using columns separated by the 'list separator' character. The first two columns are reserved for the date and time of each row of measurement data.

The **DIAdem** file format produces a text file with .DAT-extension, which contains configuration information, and a binary file with .R32 extension, which contains the measurement values. The first channel in the DIAdem format contains the time in seconds since the specified 'start date/time'.

List separator: Only used for ASCII output files. Determines the character to use as a separator between columns in the output file. The default selection 'Tab' makes it easiest to import ASCII files in spreadsheets like Excel.

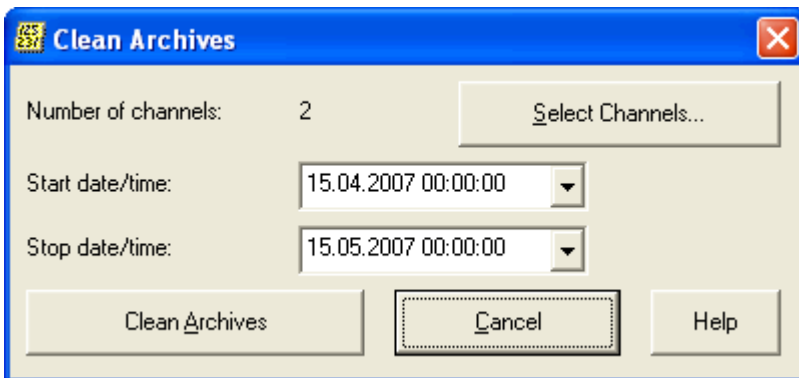
Perform extraction: Press this button to start the extraction from the archive. If the specified 'storage file' already exists, a warning will be shown before overwriting it. During extraction, an indicator shows the progress. If you cancel the operation during this progress, partial output will be available.

6.4 Clean Archives

The clean archives dialog allows you to remove data from the measurement archive. You can make any selection of channels, and specify a time interval wherein data may be removed.

When you clean the archives, no data will be removed outside the specified time interval, or for other than the specified channels.

There is no guarantee that data within the time interval will always be removed. The archive consists of multiple files for each channel. When cleaning up, only files containing data entirely inside the specified time interval are removed. The most recent file for each channel will never be removed.



The individual items in this configuration page described:

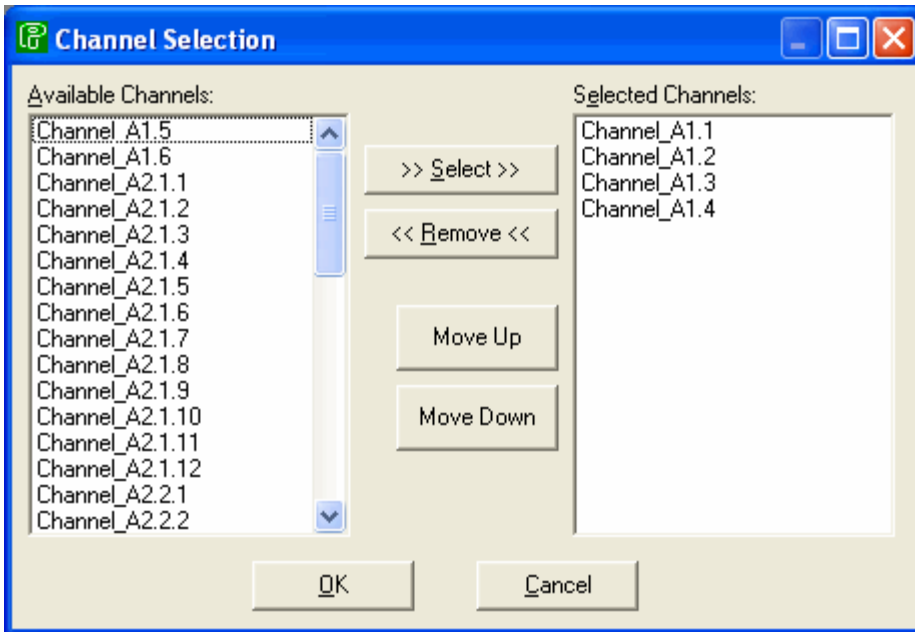
Select Channels: Use this button to make a selection of channels.

Start date/time: Sets the start of the time interval to use.

Stop date/time: Sets the end of the time interval to use.

Clean Archives: Click this button to clean the archives for the specified selection of channels. The operation will be performed immediately.

6.4.1 Channel Selection



To add channels to a group, first select one or more channels from the list of available channels on the left side of the window.

To select several channels, press and hold the <ctrl> key, then select individual channels using the left mouse button, or press and hold the <shift> key, then select a range of channels using the left mouse button.

Press the **Select** button to add the selected channels to the group.

To remove channels from the group, first select them from the list of selected channels, then press the **Remove** button to remove them from the group.

7. Device Commands

The Device menu offers the following commands:

Device Settings...	Show the settings for the selected device.
Delete	Removes the selected device from the configuration.
Store Measurement Data	Stores log data from a device on disk.
Central Balance	Performs a central balance command on the device.
Central Calibrate	Performs a central calibration command on the device.
Peak Value Active	Switches peak value detection on or off for all channels in the device. Peak values are shown in the channel overview window
Reset Peak Values	Resets the detected peak values.
Get Actual Slot Configuration from Device	Retrieves the slot configuration (which cards are in which slot) from a Signalog device.
Load All Channel Settings from Flash Memory	Retrieves the channel settings for all channels in the selected device from the flash memory in the device, overwriting the current channel settings in SignaSoft 6000.
Store All Channel Settings to Flash Memory	Stores the channel settings in SignaSoft 6000 for all channels in the selected device to the flash memory in the device.

7.1 Store Measurement Data

This option allows you to store measurement data from a device to disk. The option is only available when the device is connected and contains unsaved measurement data.

When you select this option, the device will send the measurement data from its buffer to the PC and SignaSoft 6000 will store the data using the configured settings.

8. Channel Menu Commands

The Channel menu offers the following commands (note that not all commands apply to all device/channel types):

Modify...	Show the settings for the selected channel(s).
Balance	Performs a balance command on the selected channel(s).
Tare	Performs a tare command on the selected channel(s).
Calibrate	Performs a calibration on the selected channel(s).
Measure Sensor	Measures the sensor of the selected channel(s).
Copy Settings	Copies the settings of the selected channel to the clipboard.
Paste Settings	Pastes the settings from the clipboard to the selected channel, overwriting the current settings.
Store Settings	Stores the settings of the selected channel in a file.
Load Settings	Loads settings from a file and stores them in the selected channel, overwriting the current settings.
Store Settings to Flash Memory	Stores the settings of the selected channel in the flash memory of the device.
Load Settings from Flash Memory	Loads settings for the selected channel from the flash memory of the device.
Restore Default Settings	Restores the default settings of the selected channel, overwriting the current settings.

9. Graphic Menu Commands

The Graphics menu offers the following commands:

Settings...	Show the settings for the selected graphics group.
Select Channels...	Select which channels should be displayed in the graphics group.
Zoom Back	Go back to the previous zoom level, after zooming in or out.
Zoom In (2x)	Zoom in to the middle of the graphics. You can also click and drag in the graphics view to select a specific zoom area.
Zoom Out (2x)	Zoom out to show surrounding data.
Clear Graph	Remove all current measurement values from the graphics buffer, which clears the graph.
Moving X-Axis	Selects whether X-axis should automatically move in time to make sure the latest measurement values are visible.
Left Y-Axis Autoscale	Select automatic scaling of the left y-axis to make all

	data visible.
Right Y-Axis Autoscale	Select automatic scaling of the right y-axis to make all data visible.
Show Date/Time	Option to show the date and time of start and end of graph on top.
Show Title	Option to show the title of the graph. The title can be entered freely under settings.
Show Legend	Option to show a legend under the graph.
Show Gridlines	Option to show gridlines in the graph.
Show logged values only	Option to hide measurement values, which are not specifically requested in the measurement logging configuration. Only affects Picas/Signalog devices.
Show individual measurement values	Show each single value as a cross on the graph, instead of a line connecting the values.
Export Visible Measurement Data	Allows you to export measurement data from the archives using the currently visible time range.

10. Numeric Menu Commands

The Numerics menu offers the following commands:

Settings...	Show the settings for the selected numerics group.
Select Channels...	Select which channels should be displayed in the numerics group.
Show Data in Rows	Option to show the measurement values in a row next to the channel name. By default, they are shown in a column under the channel name.
Show logged values only	Option to hide measurement values, which are not specifically requested in the measurement logging configuration. Only affects Picas/Signalog devices.

11. Trips Menu Commands

The Trips menu offers the following commands:

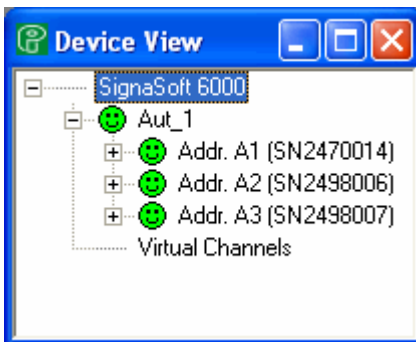
Settings...	Show the settings for the selected trips group.
Select Channels...	Select which channels should be displayed in the trips group.
Show as History	Option to show the trips as a chronological list of trip events. By default, an overview is shown indicating which trips are currently active and which aren't.
Hide channels without trips	Option to hide all channels without configured trips in the trips overview display.
Generate Logfile	Option to store trip events in a logfile. Use the trips settings to choose a filename.

12. View Menu Commands

The View menu offers the following commands:

Devices	Show the Device Overview window.
Channels	Show the Channel Overview window.
Bargraph	Show the Bargraph window.
Settings	Show the Settings window.
Trips View	Shows the Trips window.
Event Viewer	Shows the Event View window
Toolbar	Show or hide the toolbar.
Status Bar	Show or hide the status bar
Refresh	Refresh the contents of all windows in SignaSoft 6000.

12.1 Device Overview Window



From the device overview you can manage your devices and the cards and channels they contain. Information is displayed in a tree containing devices, cards and channels.

A device is shown by a line containing its type (Signalog or Picas) and its hardware address (001 up to 255).

A card is shown by a line containing its type (e.g. CA2CF or CA4CF). Empty slots are shown as 'Empty x', where x is the slotnumber.

A channel is shown by a line containing its name. By default, this names contains the type of channel (e.g. CA Amp), and an indication of device, slot and channel number (e.g. '001/2/3' for device address 001, slot 2, channel 3).

You can access settings for each of these items by clicking on them and pressing the right mouse button to open a context menu.

For the root item 'SignaSoft 6000', the context menu offers the options described under Measurement Menu Commands.

For a device item, the context menu offers the options described under Device menu commands.

For a card (or an empty slots), the context menu offers a list of possible cards to be placed in the slot. By selecting a card, channels are automatically created or deleted as needed.

For a channel, the context menu offers the options described under Channel menu commands.

To add channels to a numeric or graphics group you can select a channel from the device overview and drag it to the group you want it to be in. If you drag a device or slot to a group, all channels in that device or slot will be added to the group.

12.2 Channel Overview Window

Name	Log	Channel Type	Device	Slot #	Channel #	Frequency [Hz]	Current Value	Balance Va
Channel_A1.1	<input checked="" type="checkbox"/>	DC	Aut_1 (S/G)	1	1	+10.00	-1.5948 V	+0.00
Channel_A1.2	<input checked="" type="checkbox"/>	DC	Aut_1 (S/G)	1	2	+10.00	+0.3860 V	+0.00
Channel_A1.3	<input checked="" type="checkbox"/>	DC	Aut_1 (S/G)	1	3	+10.00	+0.4211 V	+0.00
Channel_A1.4	<input checked="" type="checkbox"/>	DC	Aut_1 (S/G)	1	4	+10.00	+0.3860 V	+0.00
Channel_A1.5	<input checked="" type="checkbox"/>	DC	Aut_1 (S/G)	1	5	+10.00	-1.5948 V	+0.00
Channel_A1.6	<input checked="" type="checkbox"/>	-	Aut_1 (S/G)	1	6	+0.00	+0.0000	+0.00
Channel_A2.1.1	<input checked="" type="checkbox"/>	-	Aut_1 (Mux S/G)	2	1.1	+0.00	+0.0000	+0.00

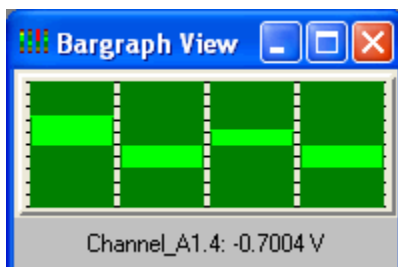
From the channel overview you can manage your channels. Information is shown in a grid containing the following columns:

- Name:** Shows the name of the channel. You can change this name to your liking by simply clicking on it and typing a new name. A user defined name is always shown in capitals and can not contain spaces or other special characters.
- Device:** Shows the device, which contains the channel.
- Slot:** Shows the slot number, which contains the channel.
- Channel:** Shows the number of the channel within the slot.
- Interval [sec]:** Shows the interval time in seconds, used for datalogging. You can change the interval time by clicking on it and typing a new value. To change intervals for multiple channels, click and drag to select them and then type a new value. If you set an interval time to 0, the channel will not be logged at all.
- Current Value:** Shows the most recent measurement value obtained for the channel.
- Balance Value:** Shows the balance value for the channel.
- Tare Value:** Shows the tare value for the channel.
- Low Peak, High Peak:** Show the lowest and highest value measured when peak value detection is active. Peak value detection can be switched on or off using the device menu, and is not available for every device type.
- Description:** Shows a description. You can enter any description you like by simply clicking on the description field and typing a new description.

If you click the right mouse button after selecting one or more channels, a context menu appears, which offers you the options described under Channel menu commands.

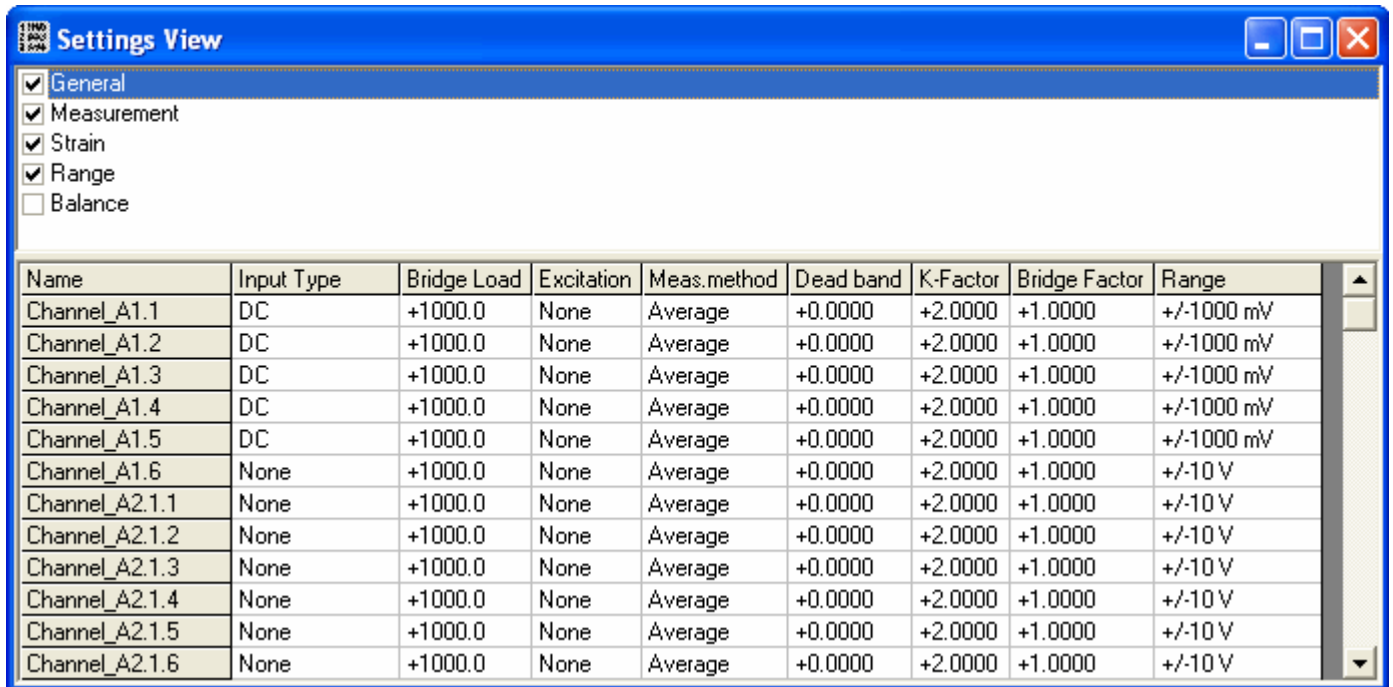
To add channels to a numeric or graphics group you can select one or more channels, then press the right mouse button on the selected area and drag it to the group you want the channel(s) to be in.

12.3 Bargraph View Window



This window shows a bar graph of a selection of channels. To add new channels to this window, drag and drop them from the device overview. You can change the order of the bar graphs by dragging them to a new position. To remove a bar graph, right click it and select the 'delete' option in the context menu. When you position your mouse over a bar graph, the channel name and its current measurement value are shown below the graphs. Double click the bar graph to show the channel configuration for that channel.

12.4 Settings View Window

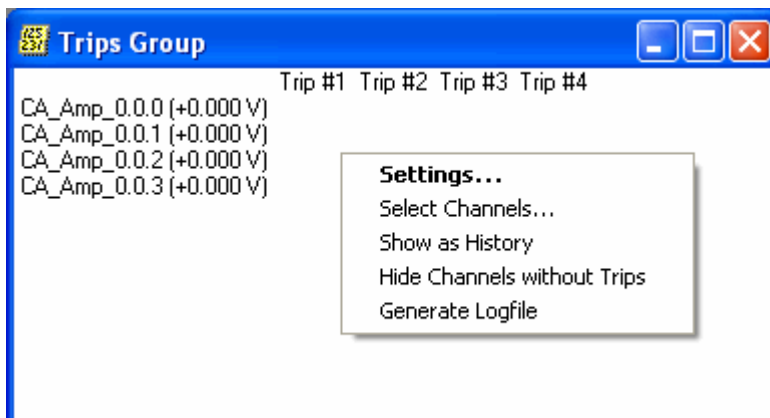


The settings view shows parameters for all Picas, Signalog 6000, Autolog 3000 and/or StrainBUSTer channels in a grid. You can use the checkboxes on the top to select the parameter groups you want to view and/or edit. The lower part of this window shows the selected parameters for all of the available channels.

To modify numbers in the grid, first select one or more items by clicking and dragging. Then type the new number, which will be assigned to all of the selected channels (if possible). For floating point numbers, you can use E-notation (eg. 1e-3 = 0.001) as well as the notation shown in the grid (eg. 1 m = 0.001). To modify a text item, select it and press the <enter> key (or click twice) to display a dropdown list containing the available options.

You can also copy and paste selected items from this grid to other applications, like Microsoft® Excel. After making modifications in Excel, you can copy the data back to this grid to change the channel parameters. If you modify text items in an Excel spreadsheet and then copy them to this grid, make sure the text matches one of the available options from the dropdown list **exactly**. You can also use numbers to indicate the dropdown options. The first (topmost) option has number 0.

12.5 Trips View Window

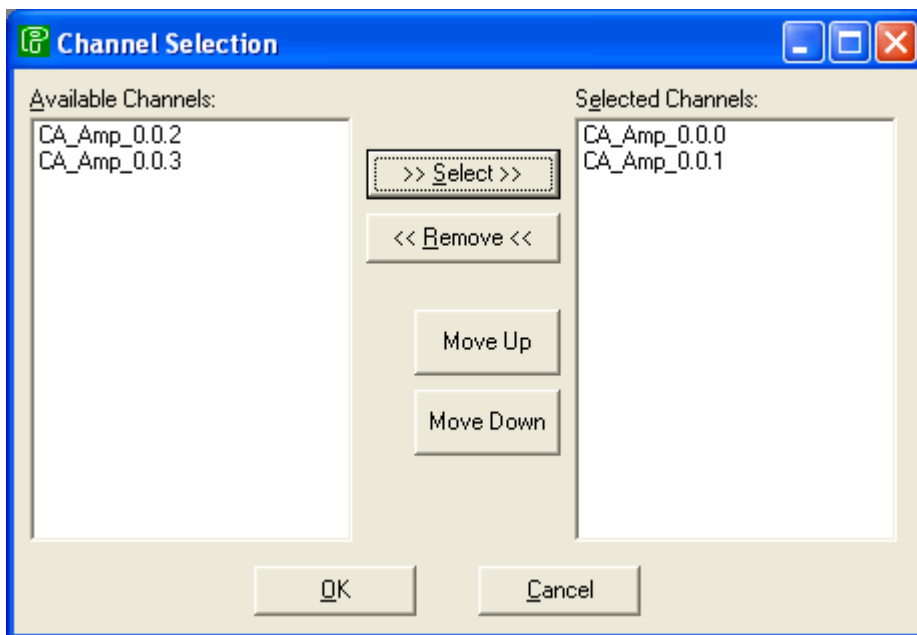


The trips view shows alarms or trips for all channels in numerical form. The name of each channel is shown, followed by the current trip state for 4 different trips. If the output is shown in historic mode, the trips window shows alarm lines (trip events) in chronological order.

To determine which channels have trips and how they are shown, click the right mouse button anywhere inside the window to show to context menu. The context menu contains the following options:

Settings...	Show the settings for the selected trips group.
Select Channels	Select which channels should be displayed in the trips group.
Show as History	Option to show the trips as a chronological list of trip events. By default, an overview is shown indicating which trips are currently active and which aren't.
Hide Channels without Trips	Option to hide all channels without configured trips in the trips overview display.
Generate Logfile	Option to store trip events in a logfile. Use the trips settings to choose a filename.

12.5.1 Channel Selection



To add channels to a group, first select one or more channels from the list of available channels on the left side of the window.

To select several channels, press and hold the <ctrl> key, then select individual channels using the left mouse button, or press and hold the <shift> key, then select a range of channels using the left mouse button.

Press the **Select** button to add the selected channels to the group.

To remove channels from the group, first select them from the list of selected channels, and then press the **Remove** button to remove them from the group.

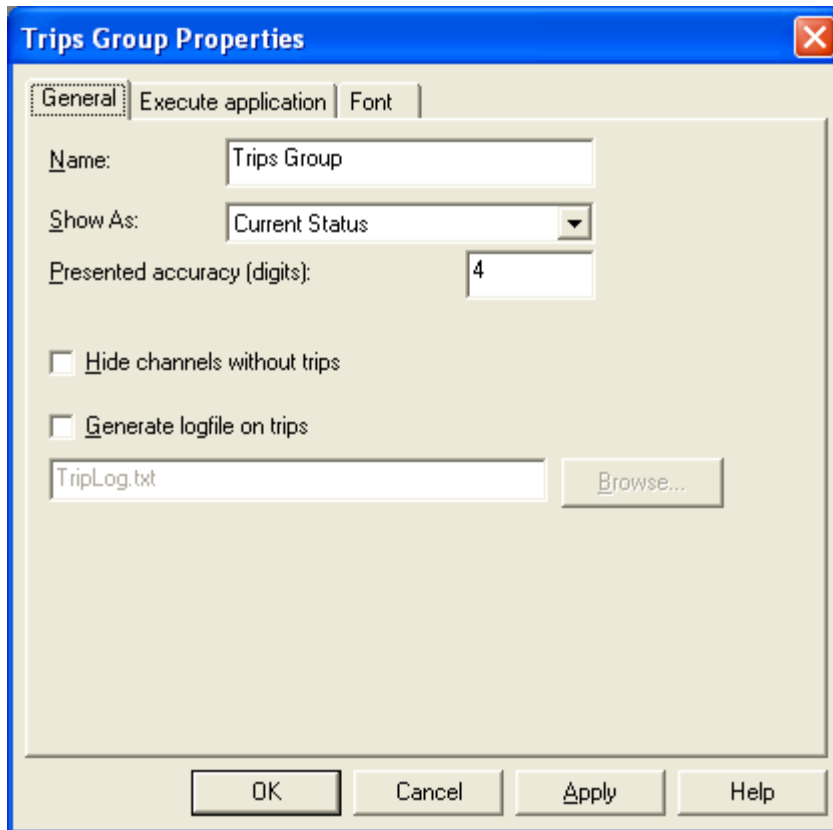
Use the **Move Up** and **Move Down** buttons to move selected channels in the group to a higher or lower position, to change the order.

12.5.2 Print Function

When you select **Print** from the right mouse button menu (context menu), the contents of the control are printed immediately on the default printer using the default settings.

Data will be formatted to make best use of the available space on paper.

12.5.3 Trips Settings



In the general settings for a trips control, the following items are available:

Name:

The name of the trips control, used to identify it.

Show As:

Choose '**Current Status**' to show the current status of all trips for all channels in an array. Choose '**History**' to show a chronological list of trip events.

Presented accuracy (digits):

Determines how many digits are shown in numeric values.

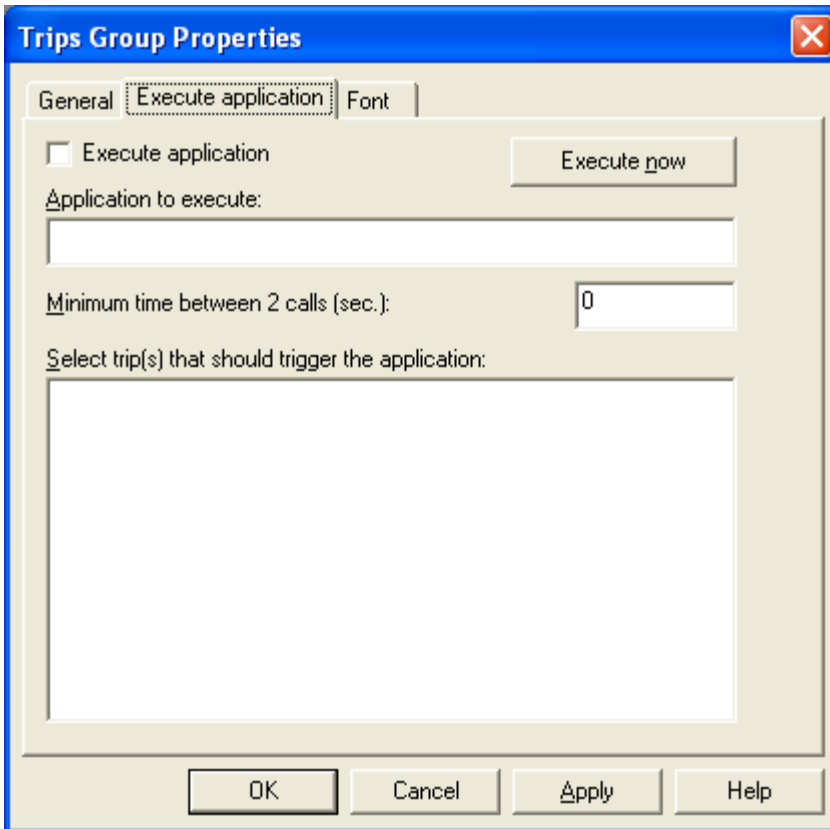
Hide channels without trips:

In 'Current Status' mode, only channels with one or more defined trips are shown if this box is checked.

Generate logfile on trips:

If this item is checked, all trip events are logged to the specified file. A single line in the file describes each event. If the file already exists, lines are appended to it.

12.5.4 Execute Application



In the 'execute application' settings for a trips control, you can configure an application to execute each time that any of a selection of trips occur.

Application to execute:

The name of the external application to execute. This should include the full path to the application and, optionally, any parameters to pass on to the application. If the application or its path contain space characters, make sure to surround it with quotes ("An Example.exe"), otherwise part of the name will be interpreted as a parameter.

Minimum time between 2 calls:

If the application takes a significant amount of time to execute, you can set a delay here to avoid executing the application multiple times.

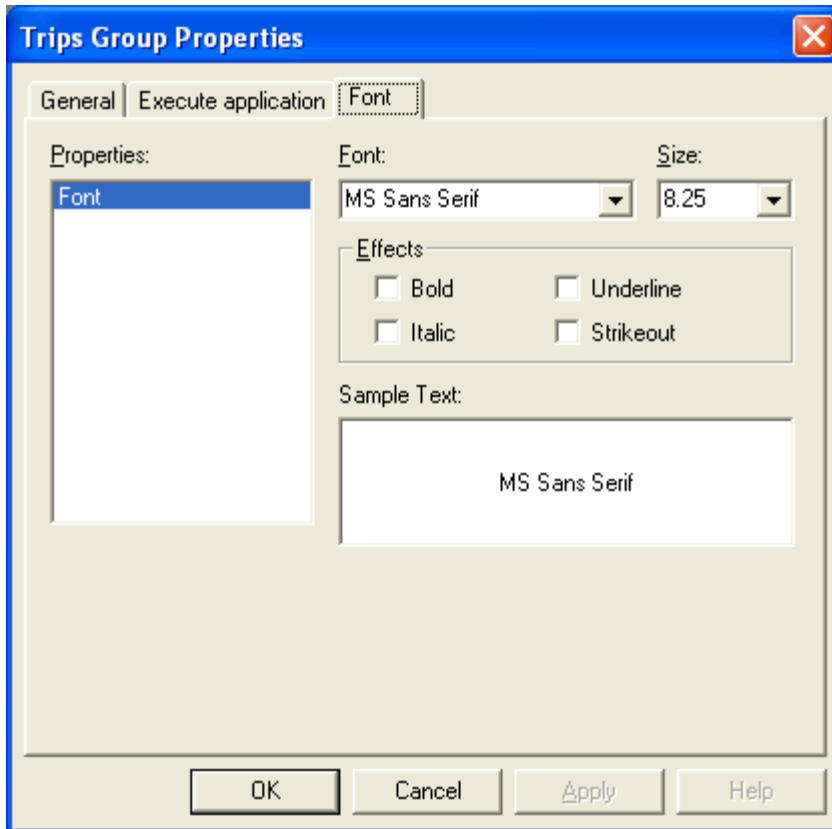
Select trip(s) that should trigger the application:

Here you can select one or more trips that will cause the application to execute when the trip level is reached.

Execute now:

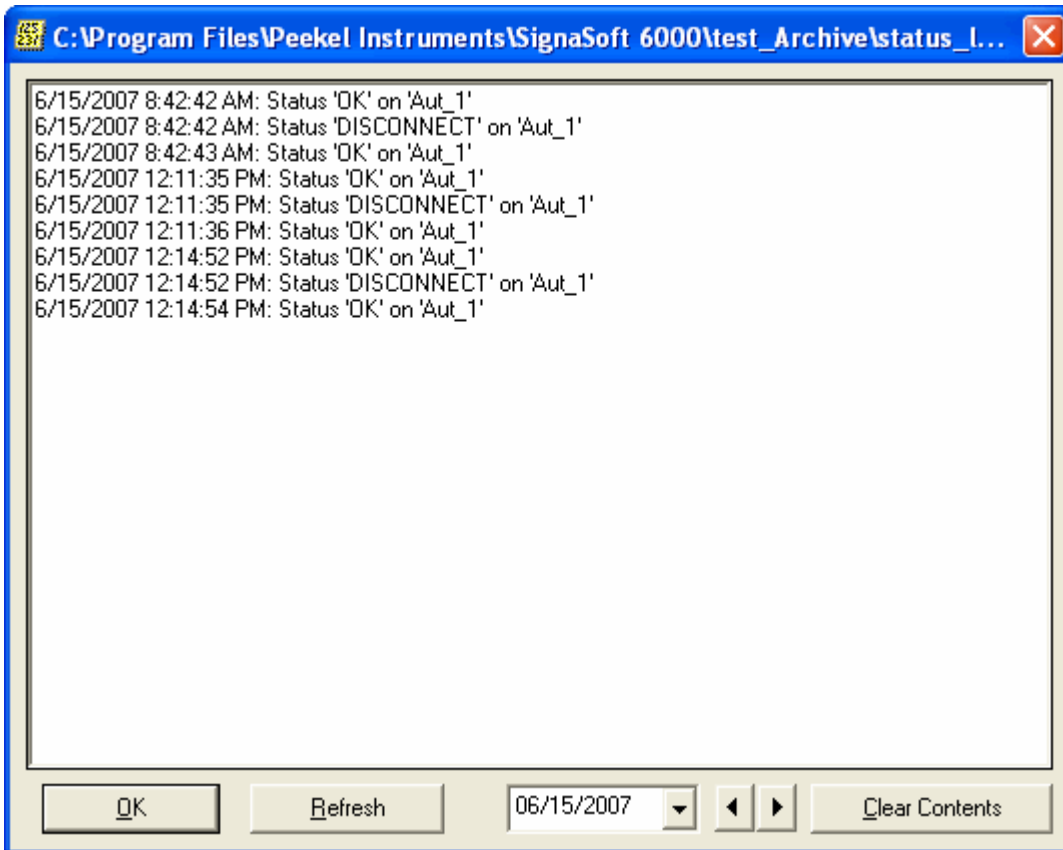
Press this button to execute the application immediately. Use this to test if your command works as expected.

12.5.5 Trip Font



In the font selection box for a trips control you can select the font used to present the output on screen.

12.6 Event View Window



This view shows a history of events. It can be accessed using the view menu or by clicking on the most recent event displayed in the status bar of the main window.

The event history shows trip activation information, logging status information and general alerts. You can use this information to find out when measurement data was stored to the measurement archives and which event caused measurement storage to start or stop.

The title of the event view shows the name of the (ASCII) file where events are stored. Each event-file contains information about one day. Use the date input field or left/right buttons to view events that happened in the past.

Press the **'refresh'** button to reread the contents of this file and refresh the window. Press the **'clear contents'** button to delete the event information for the currently displayed date and earlier. Before deleting the file(s), you are given the option of making a backup of the event history. If you make a backup of multiple event files (more than one day), the files will be merged into a single backup file.

13. Window Menu Commands

The Window menu offers the following commands, which enable you to arrange multiple views of multiple configurations in the application window:

Cascade	Arranges windows in an overlapped fashion.
Tile	Arranges windows in non-overlapped tiles.
Arrange Icons	Arranges icons of closed windows.
Window 1, 2, ...	Goes to specified window.

13.1 Cascade Command

Use this command to arrange multiple opened windows in an overlapped fashion.

13.2 Tile Command

Use this command to arrange multiple opened windows in a non-overlapped fashion.

13.3 Window Arrange Icons Command

Use this command to arrange the icons for minimized windows at the bottom of the main window. If there is an open configuration window at the bottom of the main window, then some or all of the icons may not be visible because they will be underneath this configuration window.

13.4 1, 2, ... Command

SignaSoft 6000 displays a list of currently open windows at the bottom of the Window menu. A check mark appears in front of the window name of the active window. Choose a window from this list to make it active.

14. Help Menu Commands

The Help menu offers the following commands, which provide you assistance with this application:

Contents	Offers you an index to topics on which you can get help.
About SignaSoft 6000	Displays version number and general information on SignaSoft 6000.