



# ***DI-770 and WINDAQ/Scope***

***Oscilloscope and supporting software***

## ***User's Manual***

***Manual Revision F***

***Software Release Level 1***

Copyright © 2005 by DATAQ Instruments, Inc. The Information contained herein is the exclusive property of DATAQ Instruments, Inc., except as otherwise indicated and shall not be reproduced, transmitted, transcribed, stored in a retrieval system, or translated into any human or computer language, in any form or by any means, electronic, mechanical, magnetic, optical, chemical, manual, or otherwise without expressed written authorization from the company. The distribution of this material outside the company may occur only as authorized by the company in writing.

DATAQ Instruments' hardware and software products are not designed to be used in the diagnosis and treatment of humans, nor are they to be used as critical components in any life-support systems whose failure to perform can reasonably be expected to cause significant injury to humans.

DATAQ, the DATAQ logo, and WINDAQ are registered trademarks of DATAQ Instruments, Inc. All rights reserved.



DATAQ Instruments, Inc.

241 Springside Drive

Akron, Ohio 44333 U.S.A.

Telephone: 330-668-1444

Fax: 330-666-5434

Designed and manufactured in the  
United States of America



# Warranty and Service Policy

## ***Product Warranty***

DATAQ Instruments, Inc. warrants that this hardware will be free from defects in materials and workmanship under normal use and service for a period of one year from the date of shipment. DATAQ Instruments' obligations under this warranty shall not arise until the defective material is shipped freight prepaid to DATAQ Instruments. The only responsibility of DATAQ Instruments under this warranty is to repair or replace, at its discretion and on a free of charge basis, the defective material.

This warranty does not extend to products that have been repaired or altered by persons other than DATAQ Instruments employees, or products that have been subjected to misuse, neglect, improper installation, or accident.

DATAQ Instruments shall have no liability for incidental or consequential damages of any kind arising out of the sale, installation, or use of its products.

## ***Service Policy***

1. All products returned to DATAQ Instruments for service, regardless of warranty status, must be on a freight-prepaid basis.
2. For in-warranty repairs, DATAQ Instruments will return repaired items to the buyer freight prepaid. Out of warranty repairs will be returned with freight prepaid and added to the service invoice.



---

# Table of Contents

<b>Warranty and Service Policy .....</b>	<b>iii</b>
<b>1. Introduction .....</b>	<b>1</b>
ATTENTION! .....	1
Safety .....	1
Sampling .....	2
Aliasing .....	2
Digitizing .....	3
The probes .....	3
<b>2. Specifications .....</b>	<b>5</b>
Oscilloscope .....	5
Voltmeter .....	6
Transient Recorder .....	6
Spectrum Analyzer .....	6
Arbitrary Waveform Generator .....	7
Acquisition System .....	7
Arbitrary Waveform Generator (independent from acquisition system) .....	7
Trigger System .....	8
Power Requirements .....	8
Physical .....	8
I/O Connectors .....	8
System Requirements .....	9
Operating Environment .....	9
Storage Environment .....	9
Certifications and Compliances .....	9
Package .....	9
<b>3. Installation .....</b>	<b>11</b>
Requirements .....	11
Driver Installation .....	11
Installing WinDaq/Scope Software .....	12
<b>4. Connections .....</b>	<b>13</b>
Extension connector .....	13
External power .....	13
<b>5. WinDaq/Scope Software .....</b>	<b>15</b>
Structure of the program .....	19
Starting the program .....	19
Help .....	19
Controlling the instruments .....	19
Keys that can be used .....	20
Controlling with the mouse .....	21
Program setup .....	21
Instrument taskbar always on top .....	21
Setting display colors .....	22
Setting the application language .....	22
Setting the location of files .....	22
Default text on the printer output .....	22
Device info .....	22

Setting the date and time style .....	22
Search hardware .....	22
Manually adjusting of calibration factors .....	22
Restore factory calibration factors .....	23
Loading a set of calibration values at start up .....	23
Selecting the type of duty cycle .....	23
Active instruments at start up .....	23
Saving instrument settings on disk .....	24
Restore instrument settings from disk .....	24
Storing measurement data on disk .....	24
Setting the data file type .....	24
Loading saved measurement data from disk .....	25
Setting the Waveform read method .....	25
Comparing signals using the reference channels .....	26
Fill the references .....	26
Switching references on or off .....	26
Scaling references .....	26
Update reference comment .....	27
Documentation of measurements .....	27
Add comment text to a measurement .....	27
Place a comment label .....	27
Edit a comment label .....	27
Deleting a comment label .....	28
Make a hard copy print out .....	28
Ending the program .....	28
<b>6. The Oscilloscope .....</b>	<b>29</b>
Aliasing .....	30
Displaying channels .....	30
Envelope mode .....	30
Averaging .....	31
Math mode .....	31
Setting up a channel .....	32
Setting the sensitivity of a channel .....	32
Setting the signal coupling of a channel .....	32
Setting the vertical position of a channel .....	33
Magnifying or reducing the vertical axis .....	33
Inverting a channel .....	33
The channel view button .....	34
Changing the units of the vertical axis .....	34
Mouse sensitivity for the vertical axis .....	35
Setting up the time axis .....	35
Setting the sample frequency .....	35
Setting the record length .....	35
Setting the pre trigger value .....	36
Mouse sensitivity pre trigger position adjustment .....	36
Magnifying the horizontal axis .....	37
Setting the horizontal position of the signal(s) .....	37
The record view scroll bar .....	37
Triggering .....	38
Selecting the trigger source .....	38

Selecting the trigger mode .....	39
Setting the trigger level .....	39
Setting the trigger hysteresis .....	39
The trigger symbol .....	39
The trigger time out .....	40
Performing measurements .....	40
Stopping the measurements .....	40
Perform Auto disk measurements .....	41
Auto setup .....	41
Zooming .....	42
Performing cursor measurements .....	42
Switching on cursors .....	42
Setting up the cursor measurements .....	42
Selecting the active channel .....	44
Showing or hiding the cursor readout .....	44
Switching off cursors .....	44
<b>7. The Voltmeter .....</b>	<b>45</b>
The measure system of the voltmeter .....	45
Setting the frequency range .....	46
Displaying channels .....	46
Setting up a channel .....	46
Setting the input range .....	46
Setting the input coupling .....	47
Setting a DC hardware offset .....	47
Configuring the displays .....	48
Switching displays on and off .....	48
Switching bargraphs on and off .....	48
Processing the measured value .....	49
Displaying the calculated value .....	50
Change the units of measure .....	51
Change the units per measurement unit .....	51
Relative measurements .....	51
Comparison measurements .....	52
Setting the reference impedance .....	52
Setting the number of digits after the decimal separator .....	52
Sound settings .....	52
Resetting the minimum and maximum values .....	53
Performing measurements .....	53
Start and stop measuring .....	53
Averaging .....	53
Fast measuring .....	53
One shot measurements .....	53
Storing measurement values on disk or paper .....	54
Setting the time between two measurements .....	54
Setting the hysteresis .....	54
Start an acquisition .....	54
<b>8. The Data Logger .....</b>	<b>55</b>
The data logger measuring system .....	55
Displaying channels .....	56
Setting up a channel .....	56

---

Setting up the time axis .....	56
Setting up the sample speed .....	56
Setting the record length .....	57
Setting the horizontal axis type .....	57
Perform measurements .....	57
Start measuring .....	57
Stop measuring .....	58
Scroll mode .....	58
Comparing signals using the reference channels .....	58
Performing cursor measurements .....	58
<b>9. The Spectrum Analyzer .....</b>	<b>59</b>
Description of FFT .....	60
Aliasing .....	60
Windowing .....	61
Displaying channels .....	61
Setting up a channel .....	62
Setting the input sensitivity .....	62
Logarithmic or linear vertical axis .....	62
Setting up the frequency range .....	63
Setting up the frequency axis type .....	63
Setting the frequency range .....	65
Setting the spectrum record length .....	65
Setting up the FFT .....	65
FFT window function .....	65
Performing measurements .....	66
Start and stop measuring .....	66
Single measurements .....	66
Averaging .....	66
Measuring maximum values .....	66
Measure the Total Harmonic Distortion of a signal .....	67
Performing cursor measurements .....	67
Setting up the cursor measurements .....	67
<b>10. The Function Generator .....</b>	<b>69</b>
Principle of the function generator .....	69
Structure of the function generator .....	69
Controlling the function generator .....	70
Signal shape .....	70
Frequency .....	70
Symmetry .....	70
Amplitude .....	71
DC offset .....	71
Numerical input .....	71
Presets .....	72
Storing .....	72
Selecting .....	73
Clearing .....	73
Performing a sweep .....	73
Generating a previous measured signal from a file .....	74
Measuring and storing a signal .....	74
Setting the function generator .....	74

---

# 1. Introduction

Many technicians investigate electrical signals. Though the measurement may not be electrical, the physical variable is often converted to an electrical signal, with a special transducer. Common transducers are accelerometers, pressure probes, current clamps and temperature probes. The advantages of converting the physical parameters to electrical signals are large, since several instruments for examining electrical signals are available.

The DI-770 is a two channel, 25 or 100 M samples/sec measuring instrument. With the accompanying software the DI-770 can be used as an oscilloscope, a storage oscilloscope, a spectrum analyzer, a true RMS voltmeter, or a data logger. All instruments measure by sampling the input signals, digitizing the values, process them, save them and display them.

## ATTENTION!

Measuring directly on the LINE VOLTAGE can be very dangerous. The OUTSIDE of the BNC CONNECTORS at the DI-770 are connected with the GROUND of the computer.

Use a good isolation transformer or a differential probe when measuring at the LINE VOLTAGE or at GROUND-ED POWER SUPPLIES!

In case this has not been considered and the GROUND of the DI-770 is connected to a positive voltage, a short-circuit current will be flowing. Because of this short-circuit current both the DI-770 and the computer can be damaged.

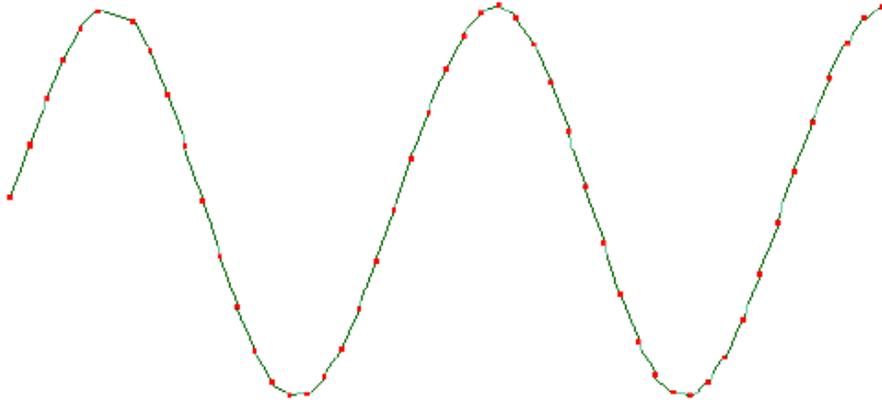
## Safety

Before you start working with the DI-770, first read these safety rules.

- Avoid working alone.
- Check the probes / test leads for damages. Do NOT use them if they are damaged.
- Take care when measuring at voltages higher than 25V AC or 60 V DC.
- Measuring directly on the LINE VOLTAGE can be very dangerous. The OUTSIDE of the BNC CONNECTORS at the DI-770 are connected with the GROUND of the computer.
- Use a good isolation transformer or a differential probe when measuring at the LINE VOLTAGE or at GROUNDED POWER SUPPLIES!
- In case this has not been considered and the GROUND of the DI-770 is connected to a positive voltage, a short-circuit current will be flowing. Because of this short-circuit current both the DI-770 and the computer can be damaged.

## Sampling

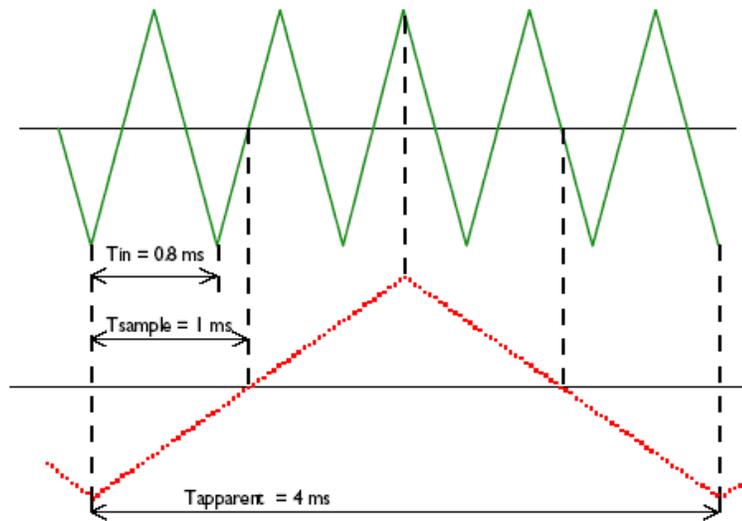
When sampling the input signal, samples are taken at certain moments. The frequency at which the samples are taken is called the sampling frequency. By taking a (large) number of samples, the input signal can be reconstructed.



The more samples are taken, the better the signal can be reconstructed. The sampling frequency must be higher than 2 times the highest frequency in the input signal. This is called the Nyquist frequency. Theoretically it is possible to reconstruct the input signal with more than 2 samples. In practice, 10 to 20 samples are necessary to be able to examine the signal thoroughly.

## Aliasing

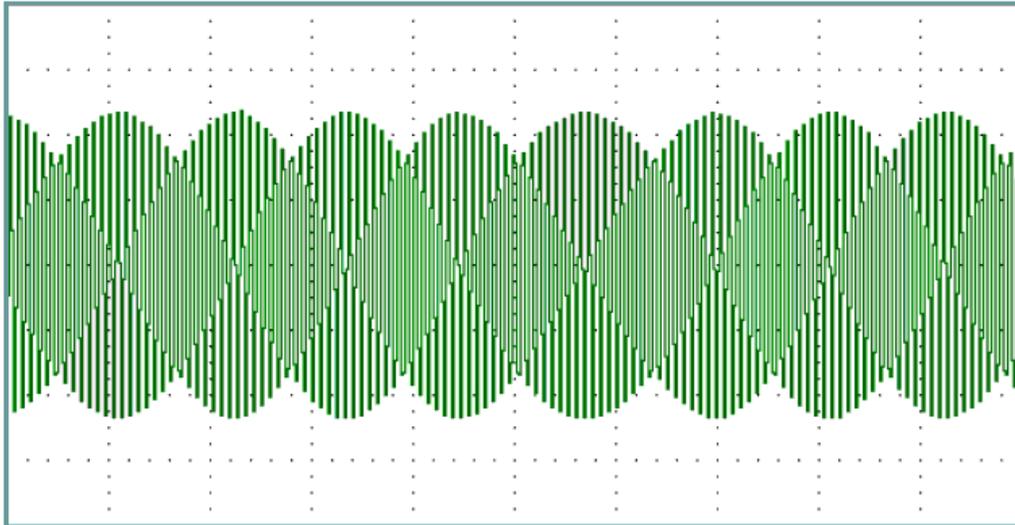
If the sampling frequency is lower than 2 times the frequency of the input signal, 'aliasing' will occur. The following illustration shows how aliasing occurs.



The input signal is a triangular signal with a frequency of 1.25 kHz (upper most in the illustration). The signal is sampled at a frequency of 1 kHz. The dotted signal is the result of the reconstruction. From that triangular signal the periodical time is 4 ms, which corresponds with an apparent frequency (alias) of 250 Hz (1.25 kHz - 1 kHz).

To avoid aliasing, the sample frequency must be higher than 2 times the maximum frequency of the input signal.

Aliasing is not always visible on an oscilloscope. In the latter illustration, it gives a 'good looking' picture. It is not apparent that aliasing occurs. The next illustration gives an example of visible aliasing.



This time it is a sine wave signal with a frequency of 25.7 kHz, which is sampled at a frequency of 5 kHz. The minimal sampling frequency should have been 51.4 kHz. For proper analysis, the sampling frequency should have been 500 kHz.

## Digitizing

After taking a sample of the input signal, it is digitized. This is done with an Analog to Digital Converter, ADC. The ADC converts the size of the signal to a digital number. This is called quantifying.

The first condition for accurate measurement is to have as many as possible quantifying steps. This can be realized by using an ADC with a resolution as high as possible.

The resolution of ADC's is often given in bits. The number of bits determines the number of quantifying steps according the formula:

$$\text{number of quantifying steps} = 2^{\text{number of bits}}$$

A 2 bits ADC has 4 quantifying steps. With an input range of 10 Volt, this ADC can divide the input range in 4 parts of each 2.5 Volt.

By increasing the number of bits, the resolution increases, the number of quantifying steps increases and the sub-divisions get smaller.

## The probes

The DI-770 is shipped with two probes. These are 1x/10x selectable passive probes. This means that the input signal is passed through directly or 10 times attenuated.

The x10 attenuation is achieved by means of an attenuation network. This attenuation network has to be adjusted to the oscilloscope input circuitry, to guarantee frequency independency. This is called the low frequency compensation. Each time a probe is used on an other channel or an other oscilloscope, the probe must be adjusted.

Therefore the probe is equipped with a setscrew, with which the parallel capacity of the attenuation network can be altered. To adjust the probe, switch the probe to the x10 and attach the probe to a 1 kHz square wave signal. Then adjust the probe for a square front corner on the square wave displayed. See also the following illustration.

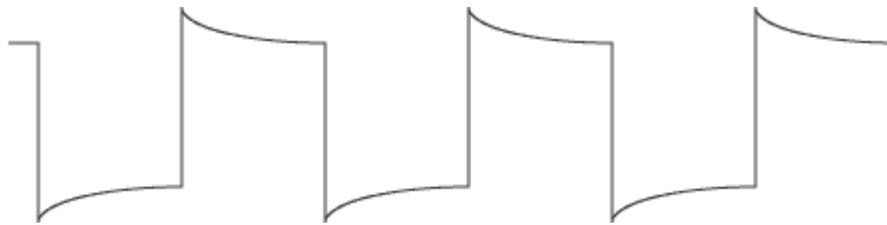
correct:



under compensated:



over compensated:



## 2. Specifications

### **Oscilloscope**

Display channels:	CH1, CH2, CH1 and CH2
Display Mode	
	Envelope: On/Off
	Averaging: 1.. 256 averaging
	Math mode: CH1+CH2, CH1-CH2, CH2-CH1
Polarity:	Normal, Invert
Vertical position setting:	200mVolt, 500mVolt, 1V, 2V, 5V, 10V, 20V, 40V 80 Volt full scale
Vertical gain setting:	0.25 to 8 magnifying
Input coupling:	AC/DC
Time base minimum:	20 day's/div
Time base maximum:	DI-770-25: 400nsec/div DI-770-100: 100nsec/div
Time base gain:	0.25 to 8 magnifying
Time base accuracy:	±100 ppm from 0°C to 70°C. Aging = ±5ppm per year.
Record length:	1 to 131060 samples per channel
Record view scroll bar gain:	1 to 125
Trigger mode:	free run, delayed run, auto, single, edge triggered, window, peak, external
Trigger system:	two digital trigger levels, 4096 positions
Trigger source:	CH1, CH2, External, Keyboard and Master
Trigger level:	0-100% full input range
Pre trigger:	1 to 131060 samples per channel
Post trigger:	1 to 131060 samples per channel
Trigger delay:	1 to 131060 samples per channel
Trigger hysteresis:	0 to 100% full input range
Trigger hysteresis resolution:	0.025% full scale
Auto disk measuring:	1 to 1000 file's
Auto setup:	connect and show
Zooming:	mouse window control, gain vertical and gain horizontal
Cursor system:	two cursors, auto tracking, auto correct points
Cursor mode:	large cursors, small cursors
Cursor measurements:	rise time left; rise time right; sample time left; sample time right; sample time difference; cursor frequency; Voltage left; Voltage right; Voltage difference; slew rate left; slew rate right; fase difference in Degrees, Radials or cos()

---

Color settings:	CH1, CH2, Ref CH1, Ref CH2, math, trigger, timebase, background, raster, cursors
Fonts:	user selectable

### ***Voltmeter***

Display channels:	CH1, CH2, CH1 and CH2
Display size:	user selectable
Number of displays:	1 to 6 user selectable
Frequency range:	DI-770-25: 10 Hz to 2 MHz DI-770-100: 10 Hz to 10 MHz
Full scale reading:	2000 counts
Accuracy DC coupled:	0.2% $\pm$ 10 counts
Accuracy AC coupled	0.3% $\pm$ 10 counts (10 to 1 MHz) 0.7% $\pm$ 20 counts (1 MHz to 5 MHz) 2.5% $\pm$ 20 counts (1 MHz to 5 MHz) low frequency roll of frequency is 10 Hz
Measurements:	True RMS, Peak-Peak, Mean, Max, Min, dBm, Power, Crest, Frequency, Duty cycle, Moment, value
Display mode:	CH1, CH2, CH1*CH2, CH1/CH2, CH1-CH2, CH2-CH1, CH1+CH2, >the Hi, <then Lo, ><COMP, <> HI LO,MAX, MIN, LOG(CH1/CH2), LOG CH2/CH1)
Measure units:	Volt, Amp, °C, °F, Watt, %, Meter, Colomd, Hertz, Bar, user defined
Relative:	CH1, CH2, user defined value
Sound settings:	No sound, 100Hz, 200Hz, 500Hz, 1kHz, 2kHz, and 10 kHz
Data storing:	direct on paper, on disk and on network
Acquisition hysteresis:	used defined value

### ***Transient Recorder***

Measuring channels:	CH1, CH2, CH1 and CH2
Measuring points:	1 to 131060
Measure time between to points:	0.01 sec to 500 sec
Measure time span:	21 min to 750 days
Measure mode:	scroll mode or scan mode
Cursor readout:	see oscilloscope
Time reference:	time of start, time of measuring, time difference

### ***Spectrum Analyzer***

Display channels:	CH1, CH2, CH1 and CH2
Display mode:	single shot, continuous
Measuring mode:	max mode, standard mode
Vertical position setting:	200mVolt, 500mVolt, 1V, 2V, 5V, 10V, 20V, 40V 80 Volt full scale
Vertical gain setting:	0.25 to 4 magnifying

---

Input coupling:	AC/DC
Frequency axis:	Logarithmic, linear, octaves or third octave
Octave range:	22.1Hz to 22.6 kHz
Frequency range:	DI-770-100: 0.01 Hz to 50 MHz DI-770-25: 0.01 Hz to 12.5 MHz
Record length:	32768 to 32 frequency components
Windows functions:	rectangular, Hanning, Hamming, Bartlett, Blackman, Parzen
Averaging:	1 to 256 measurements
Cursor readout:	see oscilloscope
Total Harmonic Distortion:	1 to 100 spectrums

### ***Arbitrary Waveform Generator***

Signal sources:	sine, triangle, square DC white noise, user defined
Sweep:	locked on spectrum analyzer
Frequency:	0.01 Hz to 2 MHz
Amplitude:	0 ± 12 Volt, max 50 mAmp
Symmetry:	0 to 99%
DC offset:	0 ± 12 Volt
Presets:	10 storage
Record length:	256 Kwords

### ***Acquisition System***

Resolution:	8 bit @ 100 MHz 12 bit @ 50 MHz 14 bit @ 3.125 MHz 16 bit @ 195 kHz
Bandwidth (-3dB):	DC to 50 MHz maximum
Sample rate:	DI-770-25: 25 MHz, 40 nsec DI-770-100: 100 MHz, 10 nsec
Sample source:	internal 0.01%, external
Number of input channels:	2 analog
Input sensitivity:	200 mVolt .. 80 Volt full scale
Input protection:	200 Volt (DC + AC peak < 10 kHz)
Input impedance:	1 MOhm / 30 pF
Input coupling:	AC/DC
Input accuracy:	0.2% ± 1 LSB
AC coupling cut off frequency (-3dB):	1 Hz with 1x probe
Memory:	0 - 131060 samples each channel

### ***Arbitrary Waveform Generator (independent from acquisition system)***

Resolution:	14 bit @ 50 MHz
-------------	-----------------

Sample rate:	50 MHz
Bandwidth:	DC to 2MHz
Impedance:	50 Ohm
Coupling:	DC
Output amplitude:	-12 Volt .. 12 Volt
Amplitude step:	0 - $\pm 0.1$ V range, 8192 steps $\pm 0.1$ - $\pm 0.9$ V range, 8192 steps $\pm 0.9$ - $\pm 12$ V range, 8192 steps
DC level:	0 - $\pm 12$ V in 8192 steps
Waveforms:	sine, triangle, square, DC, noise and user defined
Symmetry:	1 - 99%, 1% steps
Memory:	1 - 256 Kwords

### ***Trigger System***

System:	digital, 2 levels
Source:	CH1, CH2, AND, OR, digital external
Trigger modes:	rising slope, falling slope, inside window, outside window, peak
Level adjustment:	0 - 100% of full scale
Hysteresis adjustment:	0 - 100% of full scale
Resolution:	0.025% (12 bits)
Pre trigger:	0 - 131060 samples (0 - 100%, one sample resolution)
Post trigger:	0 - 131060 samples (0 - 100%, one sample resolution)
Digital trigger input level:	0 - 3.3 volt (5 volt max)

### ***Power Requirements***

Power from USB port:	500 mAmp max (2.5 Watt max)
Power via extension connector:	1500 mAmp max (7.5 Watt max)

### ***Physical***

Dimensions:	1.0 $\times$ 6.7 $\times$ 5.2 in. (25 $\times$ 170 $\times$ 140 mm)
Weight:	approx. 2 pounds 10 ounces (1190 grams)
Cord length:	1.8 meter (70 inch)

### ***I/O Connectors***

Input Channel 1:	BNC
Input Channel 2:	BNC
Output AWG:	BNC
USB:	USB 2.0 and USB 1.1 type A
Extension connector:	Sub-D 25 pins female

**System Requirements**

PC I/O connection: USB 2.0 or USB 1.1 port type A  
Operating system: Windows 98/2000/ME/XP

**Operating Environment**

Ambient temperature: 0 to 55 °C  
Relative humidity: 10 to 90% non condensing

**Storage Environment**

Ambient temperature: -20 to 70 °C  
Relative humidity: 5 to 95% non condensing

**Certifications and Compliances**

CE mark compliance: CE

**Package**

Instrument: DI-770  
Accessories: 2 oscilloscope probes 1:1 - 1:10 switchable  
Software: Windows 98/2000/ME/XP on CD  
Drivers: Windows 98/2000/ME/XP on CD  
Manual: printed on paper and digital on CD



---

# 3. Installation

The DI-770 is an external measuring instrument which can be connected to a PC.

The DI-770 is connected to a USB port of the PC using the attached cable.

The DI-770 does not need an external power supply, but is powered by the computer, through the USB.

The USB can deliver only a limited amount of power. In case the USB can not supply enough power, an extra cable is supplied which can be connected to the DI-770 and between the computer and the keyboard cable. In that case the DI-770 will be powered by the keyboard connection.

***Note:** The outside of the external power connector is connected to +5 Volt. In order to avoid shortage, first connect the cable to the DI-770 and then to the keyboard connector.*

## Requirements

To use the Windows software you need:

- An IBM compatible PC/AT with 386/25 processor or higher (mathematic coprocessor recommended)
- Microsoft Windows 3.1 or higher
- 32 MB RAM
- 4 MB free hard disk space
- A mouse or other pointing device
- A CD-ROM drive.

Optional

A sound card with wave file playback and/or general midi.

## Driver Installation

1. With your computer powered, Windows running, and all other applications closed, plug the appropriate end of the supplied USB communications cable into the USB port (labeled **USB**) on the rear panel of the instrument. Connect the other end of this cable to one of your computer's USB ports. The addition of this new piece of hardware will be "sensed" by Windows and the "Found New Hardware Wizard" will automatically be launched, anticipating the installation of a device driver for the new hardware.
2. Insert *The WINDAQ Resource* CD-ROM into your CD-ROM drive. It contains the device drivers for the DI-770. If your Windows auto play feature is enabled, the WINDAQ Software Installation will start. Click on the Exit button to leave the installation.
3. Click through the defaults to install the Driver.

***Note:** If you receive the Microsoft-issued warning that the software "has not passed Windows Logo testing to verify its compatibility with Windows XP" click "Continue Anyway" to continue with installation.*

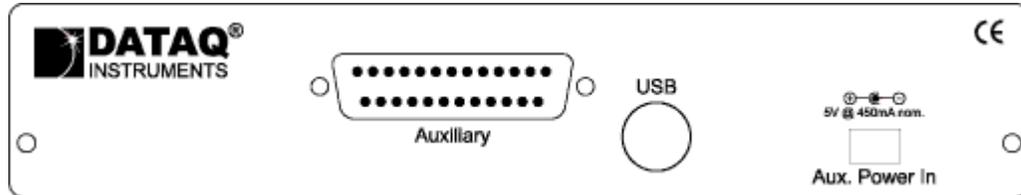
4. After the Driver is installed, click Finish to close the Found New Hardware Wizard. Windows should prompt you that the device has been installed properly.

## Installing WINDAQ/Scope Software

The WINDAQ Resource CD-ROM contains all the software required for use with your DI-770 instrument.

1. Re-insert the WINDAQ Resource CD for the auto play feature to run or run the Setup application.
2. In the “What do you want to do?” window, select “Install Software” and click OK.
3. In the “Installing Software” window, select “Install Software for all other products” and click OK.
4. From the list provided select “WinDaq/Scope for DI-770” and click OK.
5. In the Welcome! box, click OK to continue.
6. Read the License Agreement. If you accept the terms, click “Accept and Continue.” If you choose not to accept, this will end the installation.
7. When prompted, specify the directory where you want to install the software. It is recommended that you accept the default (C:\WinDaq770).
8. When prompted to Select a Program Manager Group specify a destination (or group window) in the Start Menu for your software icons. It is recommended that you accept the default.
9. Installation is complete - you will now see a Successful Installation box - click on OK to exit WINDAQ Installation.
10. To run WINDAQ/Scope software go to the appropriate program group (specified above—default is *Start > Programs > WINDAQ Scope*) and click on **WinDaq Scope**.

# 4. Connections



## Extension connector

To connect to the DI-770, a 25 pin female Sub-D connector is available, containing the following signals:

1 Ground	14 Ground
2 not connected	15 Ground
3 External power in 5V DC/ 500 mA	16 Ground
4 Ground	17 Ground
5 5 V out, 10 mA max.	18 Ground
6 External, sampling clock in	19 Ground
7 Ground	20 not connected
8 External trigger in (TTL)	21 not connected
9 Data OK out (TTL)	22 Ground
10 Ground	23 I <sup>2</sup> C SDA
11 Trigger out (TTL)	24 I <sup>2</sup> C SCL
12 Square wave generator out (TTL)	25 Ground
13 External sampling clock out	

## External power

The DI-770 is powered through the USB. If the USB cannot deliver enough power, an external power cable, to the keyboard connector, can be connected.

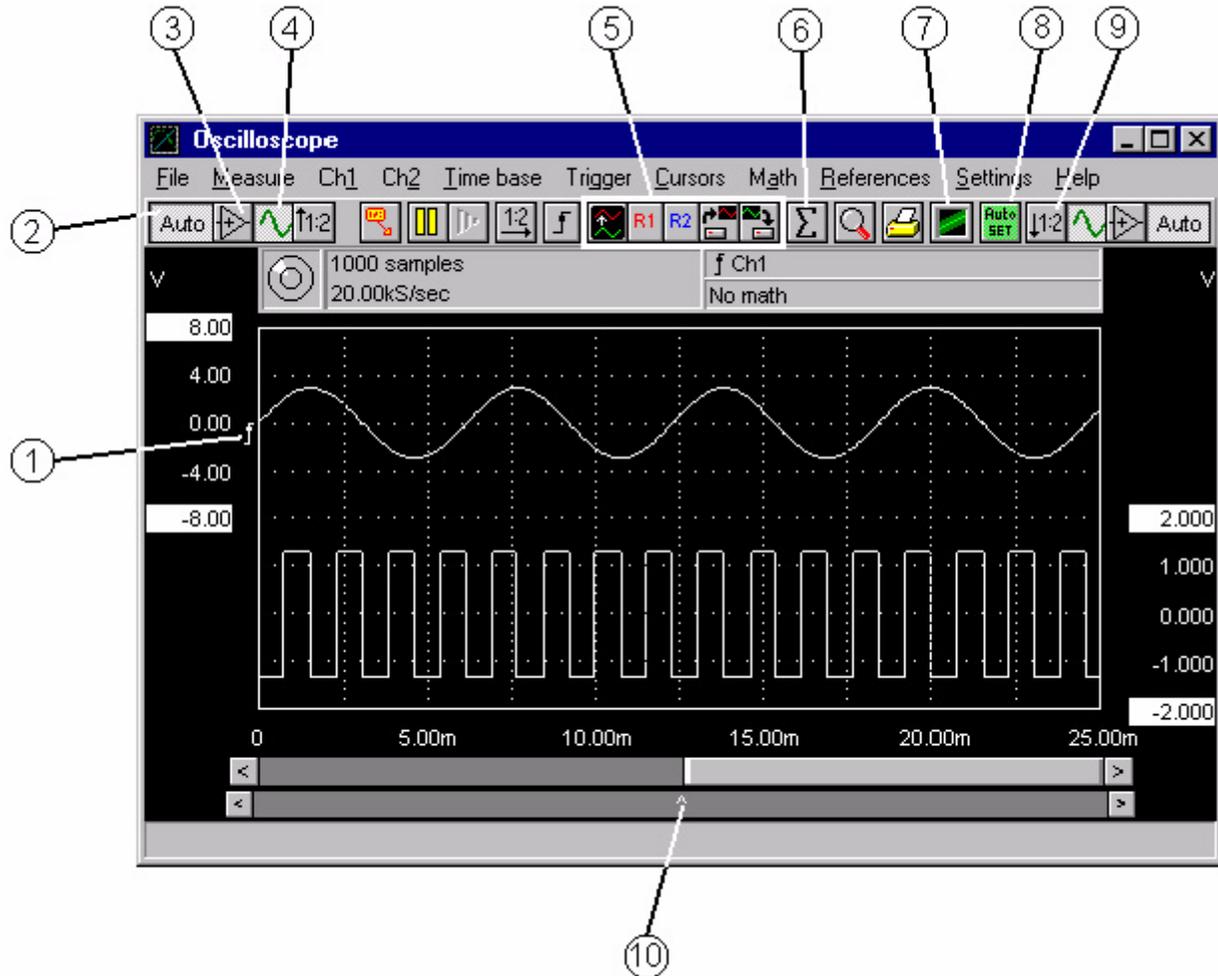
**Note:** The outside of the external power connector is connected to +5 Volt. In order to avoid shortage, first connect the cable to the DI-770 and then to the keyboard connector.

Center Pin	Ø 1.3 mm	Ground
Outside bushing	Ø 3.5 mm	+ 5 V DC



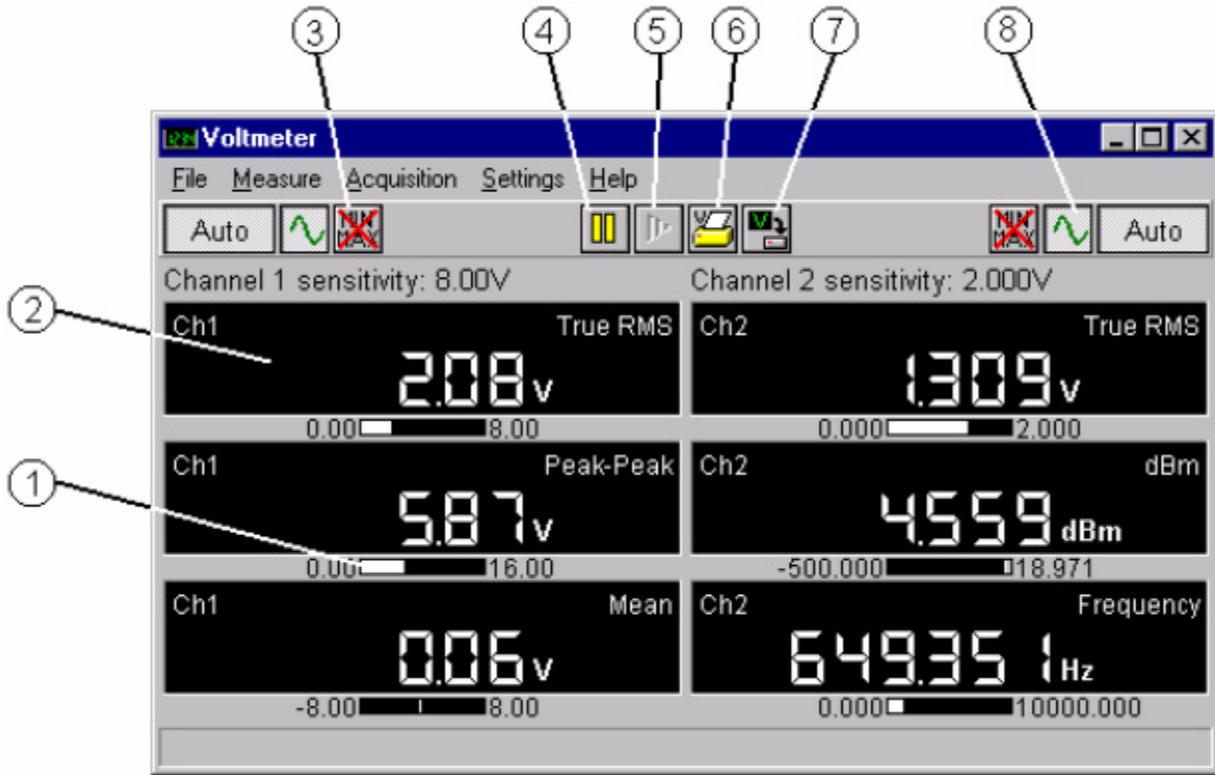
# 5. WINDAQ/Scope Software

This manual expects that the reader has a basic knowledge how to control applications running under windows, working with menus in windows applications, etc.

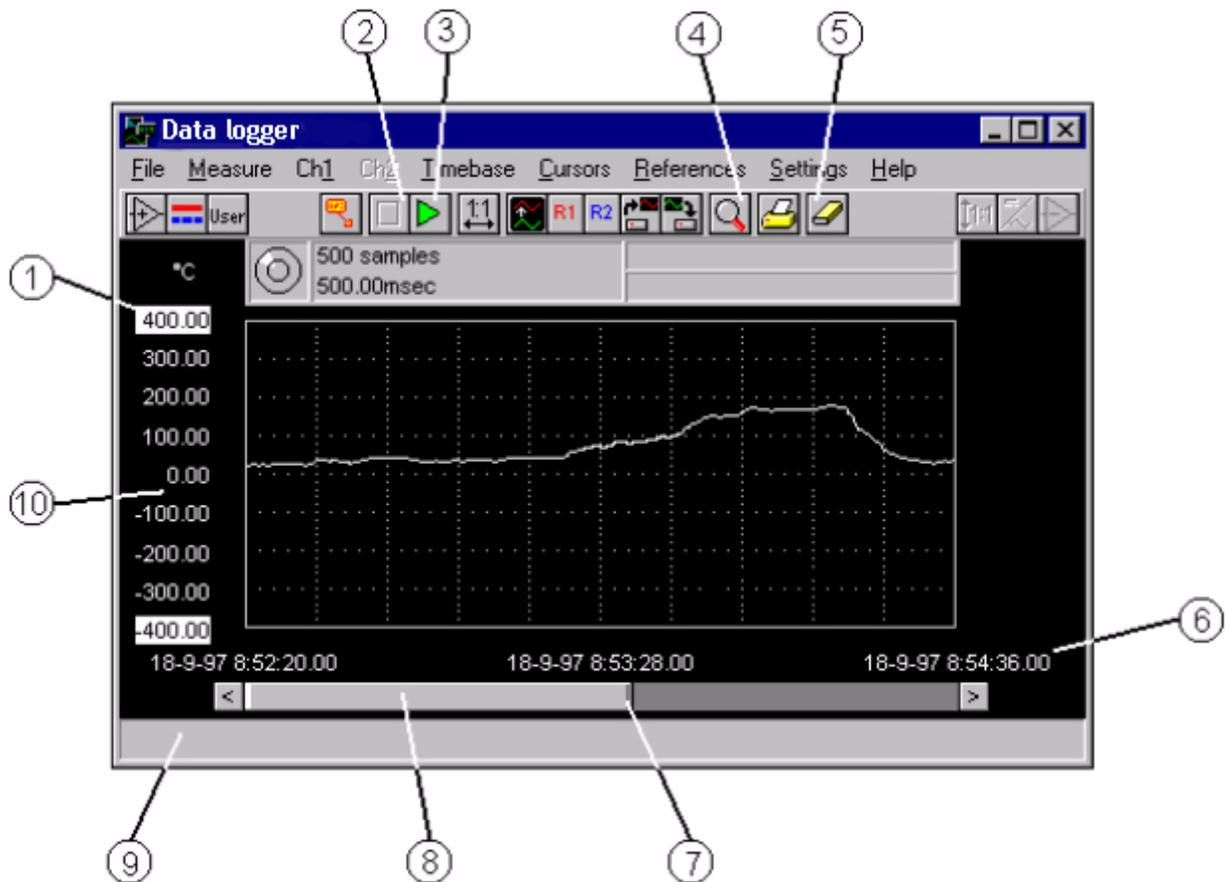


1. Set the trigger level, the trigger hysteresis and the trigger slope by dragging and double clicking. Press the right mouse button and setup the complete trigger system.
2. Toggle auto ranging for channel 1. Press the right mouse button and select an input sensitivity from a menu.
3. Toggle the invert for channel 1.
4. Set the signal coupling for channel 1 to AC or DC.
5. Copy, show, save and load reference signals.
6. Select a math channel.
7. Toggle envelope mode.

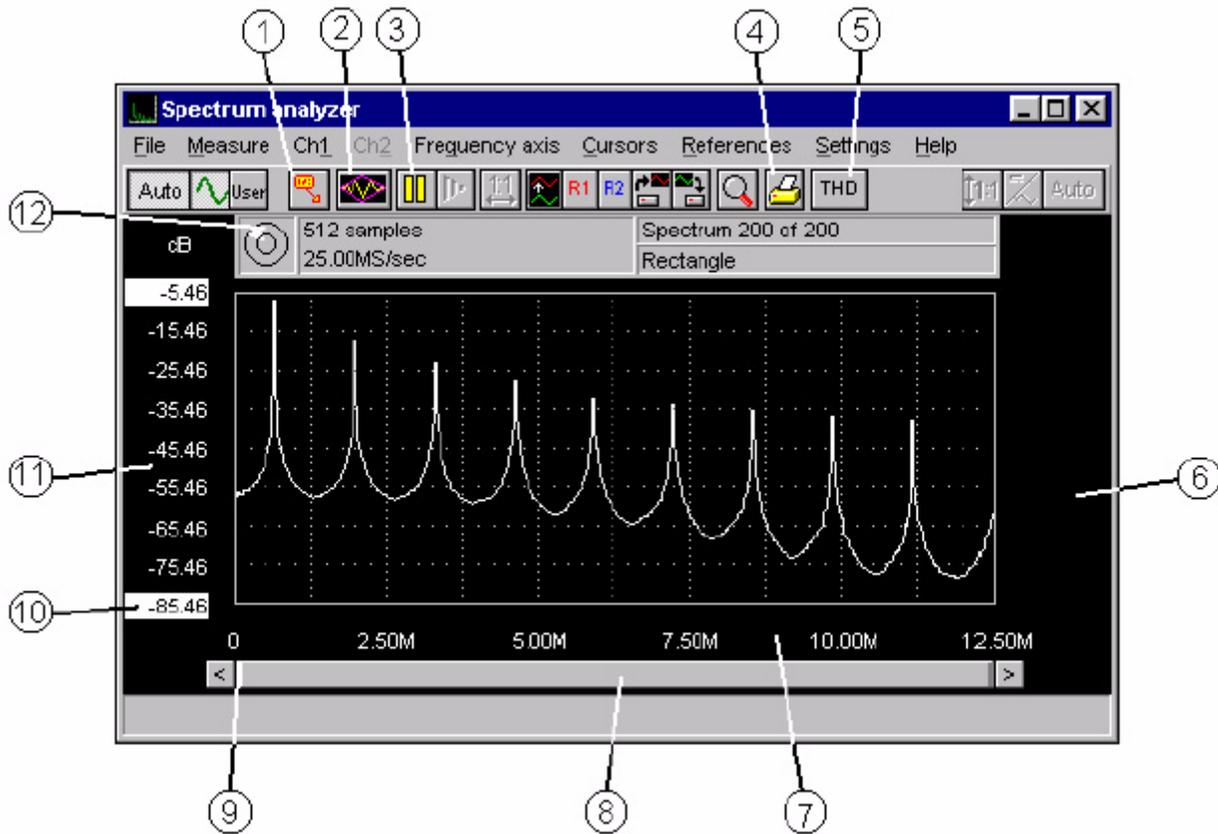
8. With the auto setup the oscilloscope sets itself according the input signal.
9. Switch between a number of pre defined offset and gain settings for channel 2. Press the right mouse button and select the settings from a menu.
10. Adjust the pre trigger point. Press the right mouse button and setup the complete time axis.



1. The bar graph of display 2 of channel 1, for a quick indication of the signal size.
2. Display 1 of channel 1, press the right mouse button to setup the display.
3. Reset the minimum and maximum values.
4. Stop the measurements with this button.
5. Perform a single measurement with this button.
6. Perform a measurement at settable times and send the result to the printer.
7. Perform a measurement at settable times and store the result on disk.
8. Switch the channel 2 signal coupling to AC or DC.



1. Drag this end of the axis and adjust the gain of channel 1. Press the right mouse button and setup channel 1.
2. With the Stop button the data logger is stopped.
3. Start a measurement with the data logger
4. Zoom in to a specific part of the signal
5. Erase the signal screen.
6. The time axis can display sample times and real time and date. Set this by opening the Settings menu. The right mouse buttons allows the complete time base to be set up.
7. Drag this end of the slider and change the horizontal magnification. Press the right mouse button and setup the time base.
8. Drag the slider and adjust the horizontal position of the signal. Press the right mouse button and setup the time base.
9. The hint bar displays explanation of the objects the mouse is placed on.
10. Drag the vertical axis and adjust the vertical position of the signal. Press the right mouse button and setup channel 1.



1. Press this button to place a comment label in the display.
2. Press the Window button to select a window function for the FFT.
3. Stop the continuous measurements.
4. Make a hard copy of the current signal.
5. Measure the Total Harmonic Distortion.
6. Double click here to switch channel 2 on.
7. Press the right mouse button to setup the frequency axis.
8. Dragging this slider adjusts the horizontal position of the signal. Pressing the right mouse button allows setup of the frequency axis.
9. Drag this end of the slider (or the other end) to adjust the horizontal magnification. Press the right mouse button to setup the frequency axis.
10. Drag this label of the axis and adjust the vertical magnification. Press the right mouse button to setup channel 1.
11. Drag the axis and adjust the vertical position of channel 1. Press the right mouse button to setup channel 1.
12. This led indicates whether the input signal is clipped (creates higher harmonics) or not.

## Structure of the program

The program is divided into 4 integrated measurement instruments, which are available on the instrument taskbar:



- An oscilloscope
- A true RMS voltmeter
- A data logger
- A spectrum analyzer

If an instrument is not active (instrument button state is up), the instrument can be made active by pressing the left mouse button over the instrument button. If the instrument is active (instrument button state is down) and you press the left mouse button over the instrument button, the instrument will be made inactive and it will hide itself.

If you have 2 or more instruments active, one instrument can be on top of the other. To make the inactive instrument active you can move the currently active window away from the inactive window, so you can select it. However there is a faster way (assuming the taskbar is visible): press the right mouse button over the instrument button.

The other four buttons represent the following:

- The signal generator which is part of the hardware instrument. Not all instruments do have a signal generator. In that case the button is not visible.
- The program setup
- The online help
- The exit button

## Starting the program

The program can easily be started by using the program group created by the setup program. By double clicking the icon with the same name as the hardware instrument you are using, the program is started.

Now an intro screen will appear with the logo, the instrument name, the program version number and the copyright notice.

After a few seconds the intro screen disappears and the instrument taskbar appears. If in the program setup any instruments are set to be active at power up, these instruments are switched on as well.

## Help

For all instruments online help is available by pressing the Help button in the instrument taskbar or by pressing the key <F1> in a window.

## Controlling the instruments

All instruments are controlled by use of the mouse, the keyboard and by menus.

## Keys that can be used

<space>	Force a trigger (oscilloscope)
<1>	Toggle reference 1
<2>	Toggle reference 2
<Ctrl + ←>	Move the left side of the record view scrollbar to the left
<Ctrl + →>	Move the left side of the record view scrollbar to the right
<Alt + ←>	Move the right side of the record view scrollbar to the left
<Alt + →>	Move the right side of the record view scrollbar to the right
<←>	Move the record view scrollbar to the left
<→>	Move the record view scrollbar to the right
<A>	Set the signal coupling of channel 1 to AC
<Shift+A>	Set the signal coupling of channel 2 to AC
<D>	Set the signal coupling of channel 1 to DC
<Shift+D>	Set the signal coupling of channel 2 to DC
<E>	Toggle Envelope mode
<I>	Toggle invert of channel 1 (oscilloscope and data logger)
<Shift+I>	Toggle invert of channel 1 (oscilloscope and data logger)
<L>	Switch the vertical axis of channel 1 between Linear and Log (spectrum analyzer)
<Shift+L>	Switch the vertical axis of channel 2 between Linear and Log (spectrum analyzer)
<M>	Toggle Measuring of Maximum values (spectrum analyzer)
<O>	Perform a single measurement (One shot)
<Ctrl+P>	Make a printout of the current display
<R>	Toggle reference 1
<Shift+R>	Toggle reference 2
<S>	Start or Stop measuring
<T>	View the Total measured record
<X>	Switch to X-Y mode (oscilloscope)
<Y>	Switch to Y-t mode (oscilloscope)
<F1>	Call the online Help
<F3>	Switch the time base one step slower (oscilloscope and spectrum analyzer)
<F4>	Switch the time base one step faster (oscilloscope and spectrum analyzer)
<F5>	Switch channel 1 one step more sensitive

<F6>	Switch channel 1 one step less sensitive
<F7>	Switch channel 2 one step more sensitive
<F8>	Switch channel 2 one step less sensitive
<F11>	Make the record length one step shorter
<F12>	Make the record length one step longer

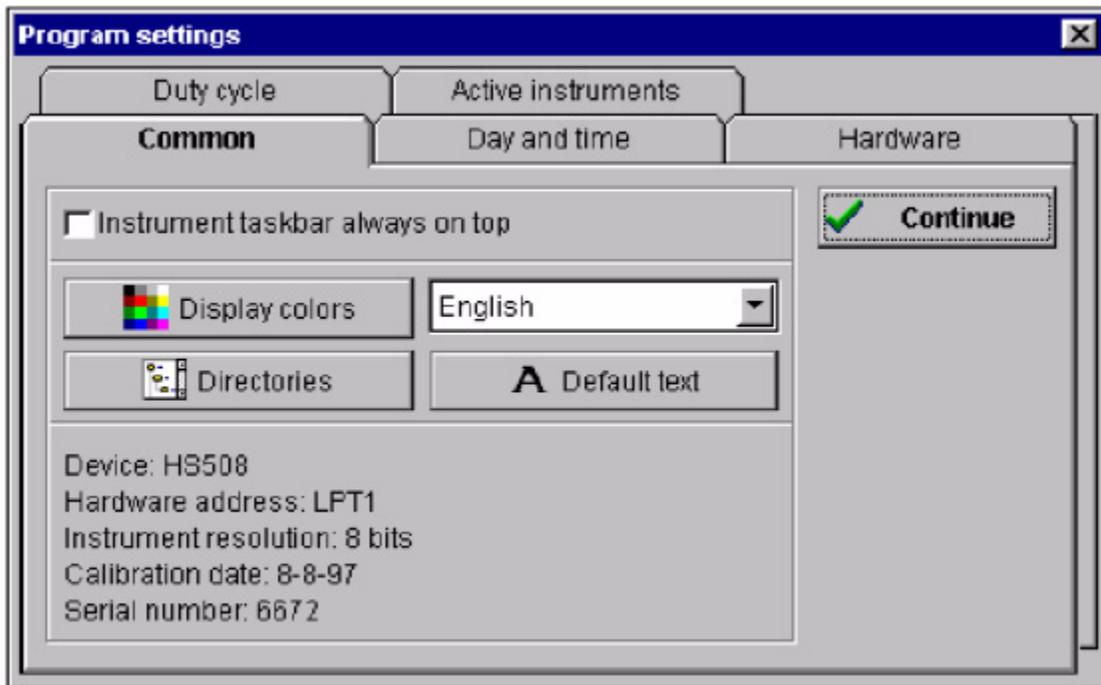
## Controlling with the mouse

All objects in the instrument windows give access to the corresponding settings by clicking the right mouse button on the objects. A popup menu will appear in which related settings can be altered.

When the mouse is placed on an object in an instrument window, information about the object is given with "fly over hints" (small rectangles with a few words of text) and some text in the hint bar, at the bottom of the instrument window.

## Program setup

Several parts of the program can be set up to your own demands. To change those settings, press the Setup button in the instrument taskbar. The following dialog will appear:



The dialog contains 5 pages with related settings, the pages can be selected by clicking the tabs. Settings that are made in these pages are saved when the application is closed and restored when the application is started again. The positions and sizes of the instrument windows are also saved and restored.

## Instrument taskbar always on top

It is possible to let the instrument taskbar appear always on top of the other windows. In that case the instrument taskbar will always be visible. The selection for that is made on the Common page, with the Instrument taskbar always on top check box.

## Setting display colors

The colors of channel 1, channel 2, reference signal 1, reference signal 2, math signal, trigger symbol, timebase, background, grid and cursors can be set to your own preferences. Press the Display colors button on the Common page and a dialog will appear in which the settings can be made.

## Setting the application language

By selecting the language selection pull down on the Common page, you can select an other language to be used in the application. The program will instantly switch to the selected language. Also a help file in the selected language will be opened when the Help function is activated.

## Setting the location of files

By pressing the Directories button on the Common page, it is possible to set the directories for the files the program works with. A popup menu will appear. From this menu you can select a directory which you wish to change. After selecting a directory, an input dialog appears in which you can enter the name for directory.

## Default text on the printer output

It is possible to add three lines of text to the application. These lines of text will be placed on every printout. They are placed on the upper left corner of the printout. Enter here e.g. the company address, employee name, telephone number etc.

To enter the text, click the button Default text on the Common page and an input dialog will appear.

The three lines of text can be entered here. To switch between the three lines, use the <Tab> key or click with the mouse in the requested line.

This text is saved when the application is closed and loaded when the application is started.

## Device info

In the lower part of the Common page some device info is given. Information that is displayed is:

- The hardware device
- The address where the hardware is found
- The resolution of the hardware

Depending on the hardware device some extra info can be displayed:

- The date of the last calibration
- The serial number

## Setting the date and time style

At several places in the program dates and/or times are displayed. The way they are displayed can be set at the date and time page. For both the date and the time several styles are available to select from.

## Search hardware

When the software is started without an instrument, a dialog appears to notify the user that the instrument is not found. It is possible to press Ignore to be able to examine previous saved waveforms. When the instrument is then attached to the computer, it can be searched and activated by pressing the Search hardware button on the Hardware page.

## Manually adjusting of calibration factors

By selecting the Adjust cal. values button at the Hardware page, a dialog appears in which the current calibration values of the instrument can be changed.

The dialog has a grid in which for both channels the gain and offset calibration values can be adjusted. On the right are the following buttons:

Continue	The currently displayed values are set as the calibration values to use. Loading of a calibration file at start-up will be disabled.
Cancel	All changes (including the applied) are ignored and the calibration values which were used prior to pressing the adjust cal. values button are restored.
Load values	A file dialog is presented in which a file can be selected with calibration values, which has to be loaded and displayed in the grid.
Save values	A file dialog is presented in which a filename can be typed or selected to which the values currently displayed in the grid can be saved.
Apply	Sets the current grid values as the calibration values to use. A one shot is done so the result is displayed. The apply is not permanent until the Continue button is pressed.

### Restore factory calibration factors

The DI-770 has a set of calibration factors in EEPROM which are read at power up of the software. If the calibration factors are altered manually, the original values can be restored by pressing the Restore factory cal. button.

If loading a calibration file at startup is enabled, this will be disabled by pressing the Restore factory cal. button.

### Loading a set of calibration values at start up

When a specific set of calibration values is made, it can be stored on disk. To use this file of calibration values at power up, place a check mark in the Use cal. file check box at the Hardware page. If no file name was selected yet, a file open dialog will appear in which a file with calibration values can be selected. If an illegal file is selected or cancel is pressed the check box will be untagged.

Pressing the Restore factory cal. button will override the values from the file.

When loading calibration values from a file is enabled and a different file has to be loaded, this file can be selected by pressing the Change cal. file button at the Hardware page. If cancel is pressed or an illegal file is selected, no changes are made to the current selected calibration file.

### Selecting the type of duty cycle

The software can use two different types of duty cycle calculation:

- The normal duty cycle setting displays the percentage of which the period is high.
- The inverted duty cycle setting displays the percentage of which the period is low.

Selection can be made on the Duty cycle page, by selecting one of the two radio buttons. The pictures next to the radio buttons display the two types in a graphical way.

### Active instruments at start up

At the Active instruments page a selection can be made which instruments have to be switched on at start up of the software.

Place a check in the check boxes of the instruments that have to be switched on at start up of the software.

*Note: The data logger can not be activated at the same time as one of the other instruments.*

When at least one instrument is selected, it is possible to select a settings file that has to be loaded at start up as well. Each single instrument can have its own settings file.

Selection of the file is done by pressing the Change file name button on the Active instruments page. In the appearing dialog the file can be selected. The name of the selected file will be displayed in the Instrument settings file box.

To clear the file name, press the Clear button.

*Note:* When more than one instrument is selected to be switched on at start up, the settings file associated with the instrument with the highest 'priority' will be loaded. The priorities in file loading are: highest: the data logger, then the oscilloscope, then the voltmeter and lowest the spectrum analyzer.

### **Saving instrument settings on disk**

When a certain kind of measurement is done often, it is useful to save the instrument setting on disk and restore it each time it is needed. This will avoid each time setting up the instrument in the same way.

To save the instrument setting to disk, enter the File menu from the oscilloscope, the data logger or the spectrum analyzer and select Save instrument setting.... A save dialog will appear in which a drive and directory can be selected and a name for the setting can be entered.

If an invalid file name or a name of an existing file name is entered, a warning will be displayed.

The settings for all instruments are saved in one file. This file has the extension .SET. It is an ASCII file, in a Windows INI format.

*Note:* This file can be edited in any ASCII editor, but it is recommended not to do so, to avoid invalid (combinations of) settings.

### **Restore instrument settings from disk**

When a certain kind of measurement is done often, it is useful to save the instrument setting on disk and restore it each time it is needed. This will avoid each time setting up the instrument in the same way.

To restore instrument settings from disk, enter the File menu in the oscilloscope, data logger or spectrum analyzer and select the Restore instrument settings... item. A restore dialog will appear in which a drive, a directory and a file name can be selected.

After pressing the OK button the file is loaded and all instruments are set according the settings in the file.

### **Storing measurement data on disk**

To save measured data for later examination, it can be stored on disk. This can be done in two ways: by activating the Write waveform to disk... item from the File menu in the oscilloscope, data logger or spectrum analyzer or by clicking the Write waveform button ().

*Note:* The voltmeter has a completely different way of storing measurement data to disk. This will be discussed in the chapter voltmeter.

Both ways will bring a save dialog in which the drive, the directory and the filename can be entered. When a name is entered and the OK button is pressed, all waveforms will be saved, together with the settings of the instrument and the references. Three files are written:

- a file with the extension .DAT, containing the waveform data of both channels and both references
- a file with the extension .GEG, containing the instrument settings
- a file with the extension .REF, containing the reference channels settings

The files containing the settings are ASCII files, with a windows INI file structure. The .DAT file can be either binary or in ASCII.

When the file already exists, a warning will be given. When the disk is full, the file will not be written and an error will be put on the screen.

### **Setting the data file type**

The waveform data can be stored in a binary and in an ASCII format.

The binary format is built of records of 8 bytes large. The first two bytes are for channel 1, bytes 3 and 4 are for channel 2, bytes 5 and 6 are for reference 1 and bytes 7 and 8 are for reference 2. The two bytes for each channel form a 16 bits value, where 0 represents -sensitivity and 65535 represents +sensitivity.

The ASCII format exists of one line of information for each sample in the record. Each line exists of:

- the sample number, related to the trigger point (sample number 0)
- the sample time, related to the trigger point ( $t=0$ )
- the voltage value for channel 1
- the voltage value for channel 2
- the voltage value for reference 1
- the voltage value for reference 2

The items are separated by commas.

The ASCII files can simply be imported in other applications, like e.g. spread sheet programs.

Files in ASCII format are much larger than files in binary format, up to 7 times.

To set the file type, enter the File menu and select the item Data file type. Choose the requested setting from the menu.

### **Loading saved measurement data from disk**

Previously saved waveforms can be read from disk in the oscilloscope, data logger and the spectrum analyzer. There are two ways to do this: by activating the Read waveform from disk item from the File menu or by clicking the Read waveform button ().

Both ways will bring up a load dialog in which a drive, directory and file can be selected. When a file is selected and the OK button is pressed, the waveform will be read from disk.

Each waveform consists of three files: a .DAT file, a .GEG file and a .REF file. If one of these files is missing or the waveform data is not compatible, an error is generated.

The data file type in which the data is written will be determined by the software and the correct reading functions are used.

Then the data is read according to the Waveform read method that is set.

After reading the data, the instrument is set to Pause (not required in the data logger) to avoid the data being replaced by new measured data.

**Note:** *Measurement data stored in the voltmeter can be read in the oscilloscope and the data logger.*

*The values of display 1 of channel 1 will be placed in channel 1 of the data logger.*

*The values of display 2 of channel 1 will be placed in Reference 1 of the data logger.*

*The values of display 1 of channel 2 will be placed in channel 2 of the data logger.*

*The values of display 2 of channel 2 will be placed in Reference 2 of the data logger.*

### **Setting the Waveform read method**

When a waveform is written to disk, all four channels (Ch1, Ch2, Reference 1 and Reference 2) are written to disk.

There are two methods to read waveforms from disk:

#### **Read all signals in file**

With this method all signals that were active when the waveform was written are read. If e.g. Ch1 and Reference 2 were active when the waveforms were saved, Ch1 and Reference 2 are read. the data in Ch2 and Reference 1 will remain unchanged. Ch1 and Reference 2 will be switched on and Ch2 and Reference 1 will be switched off.

#### **Only read current displayed signals**

With this method only the currently active channels are read from the waveform file. The other channels will remain unchanged and they will remain switched off.

To select the proper waveform read method, enter the File menu, select the Waveform read method item and select the correct method. When closing the application, the selected method is saved in the application settings file. This file is always read when starting the application.

### ***Comparing signals using the reference channels***

To be able to compare live signals to other signals, the oscilloscope, the data logger and the spectrum analyzer have two reference channels, R1 and R2. R1 is a reference channel for channel 1 and R2 is a reference channel for channel 2. These reference channels are filled once and remain unchanged while measuring. The references only change when new data is stored in the references.

### **Fill the references**

To fill the reference channel(s), two ways are available: copying the current displayed signals to the reference channels or read previously saved waveform data from disk.

To copy the current live signals to the reference channels, either press the corresponding button on the speed button tool bar () or select the item Copy live to refs from the References menu. This will copy the waveform data from the active channel(s) to the corresponding reference channel(s). The settings of the channel(s) will also be copied to a special reference setting. If the reference channel(s) was (were) not yet displayed, it is (they are) switched on as well.

To fill a reference channel with data from disk, see the paragraph on Loading saved measurement data from disk.

The data in the reference channels will remain unchanged until new data is stored in the reference channels. The reference channels are displayed with the same record view offset and record view gain as the live channels. Changing the pre trigger value will not affect the reference channels. When the record length of the live channels is changed to a shorter record after filling the reference channels, less data of the reference channels is displayed, but the data is kept in memory. When the record length is changed to a longer record, the part of the reference channel that was not filled, is filled with zeroes.

### **Switching references on or off**

Switching a reference channel on or off can be done by selecting the item Display Refx from the Reference menu, where x stands for 1 or 2. It can also be done by means of a button on the speed button tool bar:  for reference 1 and  for reference 2. When a button appears to be pressed, the corresponding reference is switched on.

### **Scaling references**

When a reference channel is filled, the data remains the same until new data is stored in the reference channel. When the input sensitivity, the software offset or the software gain for the corresponding channel changes, the reference channel will not change.

It can be useful that when a signal is copied to a reference channel to compare live measurements with that reference channel, that the settings of the reference channel will follow the settings of the live channel. So when e.g. the input sensitivity of the live channel changes, the reference channel is scaled in such a way that the live channel and the reference channel can still be compared on a 1:1 base.

To do that, select the item Scale references from the Reference menu. Any change of the input sensitivity, the software offset or the software gain of a channel will also be applied to the reference channel. When scaling references is switched on, a check mark is placed in the menu item.

To switch Scaling references off, select the Scale references item again.

## Update reference comment

When a measurement is taken, three lines of comment text can be added to the measurement. This text is saved to disk together with the waveform data and printed out with the waveform. (See Add comment text to a measurement)

When the reference channels are filled with data from a file and the comment text has to be changed, this can be done using the item Update comments ... from the Reference menu. After activating the item, a dialog appears in which the three lines of text can be entered / changed. When the dialog is closed with the OK button, the new text is automatically saved to disk.

## Documentation of measurements

In the oscilloscope, the data logger and the spectrum analyzer several features are available to document measurement data.

## Add comment text to a measurement

For explanation and documentation, each measurement can be supplied with three lines of text.

This text will be saved together with the waveform data when the waveform is written to disk. The text will also appear on a hardcopy printout.

To enter the text, open the Settings menu and select the item Comment... A dialog appears in which the three lines of text can be typed. To switch between the three lines, use the mouse or the <Tab> key.

*Note: The text remains the same until new text is entered or the lines are cleared. So when text is entered and more than one waveforms is saved or printed, each waveform will have the same comment.*

## Place a comment label

When documenting a measurement, it can be useful to add some comment to a specific part of the signal. This can be done with the special comment labels.

These comment labels are placed within the signal display at a specific position in relation to the waveform. The comment labels contain text and can have an arrow pointing at a specific point of the waveform. The shape, the colors and the arrow can all be customized to your specific wishes. When printing the measurement, the comment labels are placed as well. To place a comment label, press the comment label button () or activate the Add a comment label item from the Settings menu.

The comment label is now placed within the signal display. It can be dragged to any position with the mouse and the arrow can be dragged to any position as well. The comment labels are waveform based, when the record view offset or the record view gain changes, the position of the comment label remains the same in relation to the record.

## Edit a comment label

To change the properties of a comment label, place the mouse pointer on the comment label or the arrow of the comment label and press the right mouse button. A menu will pop up in which all properties can be edited.

### Comment text...

With this item the text in the comment label can be edited. An input dialog with a text input box appears after activating. Enter the text for the comment label in the input box. Multiple lines can be entered and will appear in the comment label as they appear in the text input box.

### Comment shape

With this item the shape of the comment label can be selected. Three choices are available: a rectangular shape, a rectangular shape with rounded corners and an ellipse shape.

### Draw arrow

With this item the arrow can be switched on or off. When the arrow is switched on, a check mark is placed in front of the menu item.

### Arrow point shape

The arrow point can have different shapes: no point at all, a square, a circle, a triangle and an arrow. When no point is selected, only a line from the comment label to the end point of the arrow is drawn. When the arrow point is selected, a triangle is drawn that is always pointing away from the comment label. When Draw arrow is switched off, this item is not available.

### Comment colors

With this choice the foreground color and the background color of the comment label and the arrow can be selected. The foreground color is used for the line around the comment label, for the text, for the arrow and for the line around the arrow point. The background color is used to fill the comment label and the arrow point.

### Deleting a comment label

To delete a comment label, place the mouse cursor on the comment label to delete and press the right mouse button. A menu will pop up. Select the Delete comment item to delete the comment label.

### Make a hard copy print out

To save a measurement on paper, a WYSIWYG hardcopy utility is added.

The printout contains all active signals with the settings at the moment of activating the hardcopy function. All relevant information about the signals (sensitivity, offsets, gains, sampling frequency, record length etc.) are placed around the signal display.

A record view indicator is placed in the upper right corner of the printout, indicating which part of the measured record is displayed, where the trigger point is located and where the cursors are located in the record (if switched on).

If cursors are switched on, the cursors are drawn (when in the display) and a table with the cursor readout is drawn.

If comment labels are placed, they are printed in the same way as they appear on the screen.

The hardcopy can be activated in three ways: using the key combination <Ctrl+P>, using the print button () or by entering the File menu and selecting the Print... item.

A print dialog appears in which the current selected printer is indicated. To change the selected printer or the settings of the selected printer, press the Setup button. A new dialog appears in which another printer can be selected or the settings of the current printer can be changed.

*Note: In theory all paper sizes are supported, but envelope paper sizes will give unusable results. Selecting landscape or portrait has no use, since the hardcopy function forces landscape.*

When the correct printer is selected and all settings are made, press the OK button to leave the dialog. Press the Cancel button to leave the dialog without changes.

The check box Color print out determines whether a color printout is made or a black and white printout is made. When this box is checked and the printer supports colors, a color printout is made. If the printer does not support colors, different shades of gray are created (by the windows printer driver). If the box is not checked, only black is used.

To start printing now, press the Print button. A dialog appears, indicating that the hardcopy is generated and sent to the printer.

### Ending the program

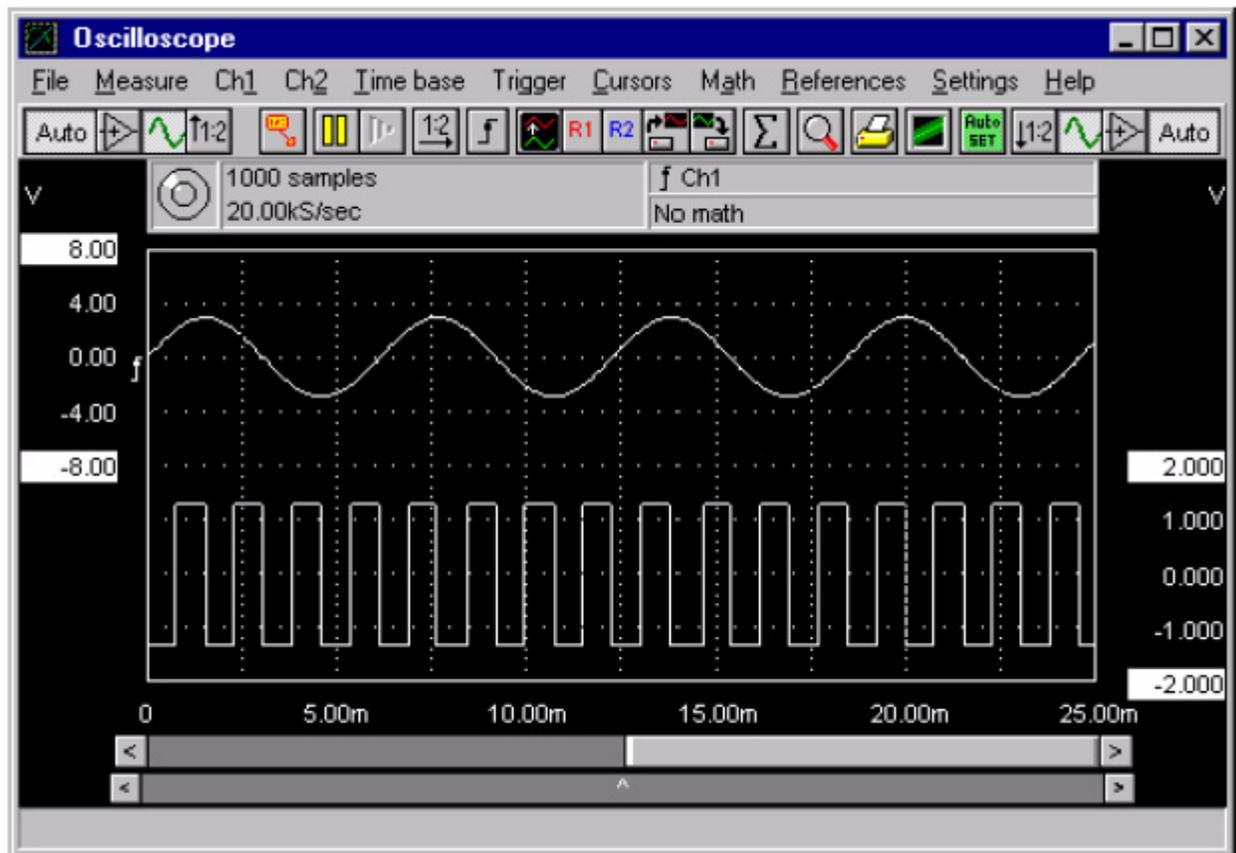
With the right most button in the instrument taskbar the application can be terminated. The application settings and instrument window positions are saved on disk.

# 6. The Oscilloscope

The oscilloscope is an instrument with which electrical voltages varying in time can be displayed. With the oscilloscope time dependent electrical signals can be examined easily.

The oscilloscope has separated voltage inputs, channels, which can be configured individually. It is a digital sampling oscilloscope. That means that the oscilloscope takes samples at fixed times. From each sample the value is determined and the size is displayed at the screen. The screen is filled with all samples. Between two adjacent samples on the screen a line is drawn. The speed at which the samples are taken, is adjustable.

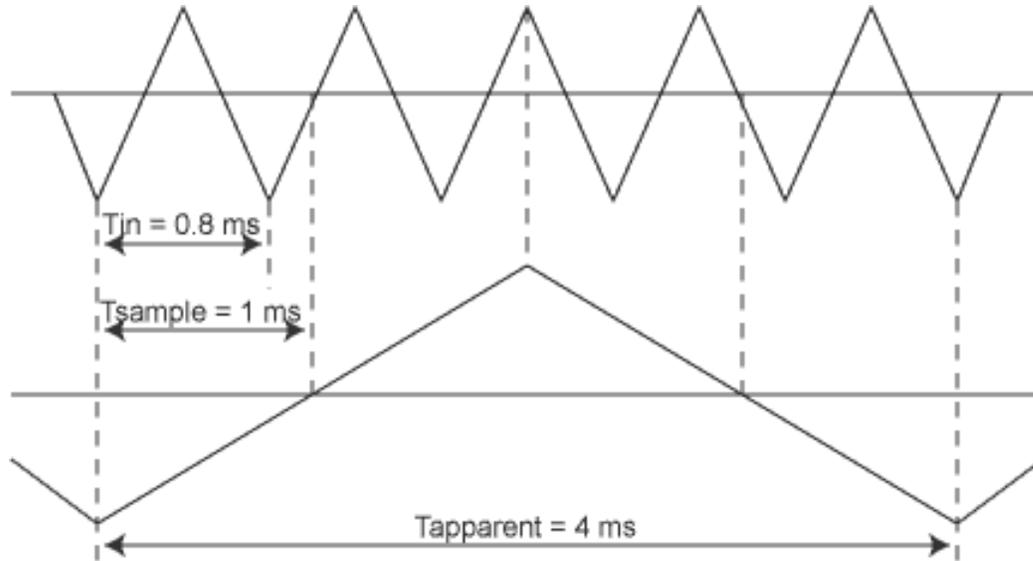
Cursors are available to perform voltage, time or frequency measurements on the displayed signal.



### Aliasing

A disadvantage of digital sampling oscilloscopes is the fact that aliasing can occur.

Aliasing occurs when a too low and therefore wrong sampling frequency is used. The next illustration shows how aliasing occurs.



The input signal is a triangular signal with a frequency of 1.25 kHz (upper most in the illustration). The signal is sampled at a frequency of 1 kHz. The dotted signal is the result of the reconstruction. From that triangular signal the periodical time is 4 ms, which corresponds with an apparent frequency (alias) of 250 Hz (1.25 kHz - 1 kHz).

To avoid aliasing, the sample frequency must be greater than 2 times the maximum frequency of the input signal.

If you have any doubts about the displayed signal, you can set the timebase of the oscilloscope one step faster or slower and check whether the signal at the display changes accordingly. If that does not give any clearance, you can determine the frequency of the input signal with the spectrum analyzer. Therefore you must set the frequency range of the spectrum analyzer to the maximum, to avoid aliasing with the spectrum analyzer. For more information, see the chapter Spectrum Analyzer.

### Displaying channels

The oscilloscope has several modes to display the two channels. To select the measure mode, enter the Measure menu and select:

Ch1	measure and display only channel 1
Ch2	measure and display only channel 2
Ch1 and Ch2	measure channel 1 and channel 2 simultaneously and display the values of both channels

**Note:** When the measure mode Ch1 and Ch2 is selected, displaying math signals becomes available.

When only one channel is measured, the controls of the other channel are disabled or removed from the display.

### Envelope mode

When a signal is monitored and a glitch is expected, envelope mode can be switched on to be sure to see the glitch. In envelope mode, for each sample in the record a highest value and a lowest value is stored. Each measurement is checked whether the new value lies between the previous highest and lowest or not. If not, the highest or lowest value

is changed to the new value. Then a vertical line is drawn between the lowest and highest value. So a glitch is always captured.

Envelope mode is reset each time when the setting of the instrument changes (sensitivity, offset, record length etc.), when the window size changes and when the pre defined reset number is reached. To set the reset number, enter the Envelope menu, which can be found in the Measure menu or by clicking the right mouse button on the Envelope mode button (). Select the item Reset envelope after and a menu with the possible option pops up. When the

choice 32 measurements is selected, envelope mode will be reset after 32 measurements. When the choice infinite measurements is selected, envelope mode will not be reset automatically, only by changes in the instrument settings or by a window size change.

Envelope mode can be switched on by clicking the envelope button, by pressing the <E> key or by selecting the item Envelope mode from the Measure menu. When envelope mode is switched on, the Envelope button stays down.

Envelope mode is switched off by pressing the button again, pressing the <E> key again or by selecting the Envelope mode menu item again.

*Note: Envelope mode is only a display function. The envelope signal is not stored to disk or printed in a hardcopy operation.*

## Averaging

When a noisy signal is measured, the influence of the noise can be eliminated by averaging several measurements. The noise component(s) will disappear and the actual signal remains.

Averaging can be switched on by selecting the item Perform averaging of from the Measure menu. In the next menu a number of measurements can be selected that have to be averaged. The item 1 measurement will switch off averaging. Averaging can also be switched on or off by pressing the Averaging button (.

When averaging is set to e.g. 20 measurements and the oscilloscope is set to Pause, pressing the One shot button will result in taking 20 measurements which are averaged

Averaging is reset each time when the setting of the instrument hardware changes (sensitivity, coupling, sampling frequency, record length etc.) and when the pre defined reset number is reached.

*Note: When a number of pre samples is set, averaging will have a large effect on the shape of the signal in the pre samples because this part of the measured record changes much.*

## Math mode

The oscilloscope is equipped with a special Math channel. This channel can display the following signals:

Ch1 + Ch2  
Ch1 - Ch2  
Ch2 - Ch1

These are display added or subtracted signals. This means that the on screen signal sizes are added or subtracted. The position and size of the math signal can be controlled by the software offset and software gain of both channels.

To get a math channel, both channels have to be on. Selecting a math channel is done by selecting a menu item from the Math menu. The Math menu can be found in the main menu or can be called by clicking the math button (.

*Hint: When the oscilloscope operates in split screen mode and ch1 uses the upper half of the screen and ch2 uses the lower half of the screen, the math channel comes in the center of the screen.*

The math channel can be switched off again by selecting the item No math from the Math menu.

## Setting up a channel

The two channels of the oscilloscope are completely configurable. Both channels are identical, each has its own menu item in the main menu. Since the possibilities for both channels are equal, channel 1 is explained here. Where channel 1 is described, the same applies to channel 2.

With the menu item Ch1 (or Ch2) from the main menu a popup menu is called with all settings for a channel.

## Setting the sensitivity of a channel

The input sensitivity of a channel determines how large a certain signal will be displayed, the lower the full scale value, the larger the signal will be displayed. It is also possible to enable an autoranging function.

The sensitivity can be set through the Sensitivity menu. The Sensitivity menu is called by enabling the Ch1 menu and then selecting the Sensitivity menu. In this menu the input sensitivities are available. Also an Autoranging setting is available. The Sensitivity menu can also be entered by clicking the right mouse button over one of the Ch1 or Ch2 controls and readouts.

To switch on Autoranging, a special button is available. When auto ranging is switched on, the button caption is Auto, otherwise it holds the current set input sensitivity. Pressing the right mouse button on this button will bring up a menu with all sensitivities.

**Note:** When autoranging is switched on, several measurements might be needed to set the input sensitivity, depending on the input signal.

Another way to set the sensitivity is by using the keyboard. A few keys have been reserved for setting the input sensitivity of the channels. These keys are:

<F5>	Increase Ch1 sensitivity
<F6>	Decrease Ch1 sensitivity
<F7>	Increase Ch2 sensitivity
<F8>	Decrease Ch2 sensitivity

## Setting the signal coupling of a channel

The input signal can be coupled directly to the ADC (DC) or through a capacitor to the ADC (AC). In the hardware, a capacitor is switched into (AC) or out of (DC) the signal path.

In COUPLING DC the complete signal (AC+DC) at the input is passed through. In COUPLING AC only the AC component of the input signal is passed through. It is then possible to examine e.g. a  $\pm 20$  mV ripple on a 30 V DC voltage, since only the ripple is passed through and the DC voltage not. AC voltages with a frequency lower than  $\pm 4$  Hz are not coupled correctly.

There are three different ways to set the coupling. Two ways are: by clicking the coupling button or by pressing keys. The third way is through the Signal coupling menu.

The Signal coupling menu is called by enabling the Ch1 menu and then selecting the Signal coupling menu. In this menu the settings Coupling AC and Coupling DC can be found. The menu can also be entered by clicking the right mouse button over one of the Ch1 or Ch2 controls and readouts.

In the tool bar for each channel a coupling button is found, the coupling button for Ch1 at the left hand side of the screen, for Ch2 at the right hand side of the screen. The picture at the button indicates the state of the coupling setting. The left button of the two buttons below is indicating AC coupling, the right button is indicating DC coupling.



The current coupling state is displayed in the coupling button.

The keys which can be used to change the signal coupling are:

<A>	sets the coupling of channel 1 to AC
<D>	sets the coupling of channel 1 to DC
<Shift+A>	sets the coupling of channel 2 to AC
<Shift+D>	sets the coupling of channel 2 to DC

## Setting the vertical position of a channel

The vertical position of a channel on the screen is software adjustable. When setting the vertical position of a channel the signal is adjusted graphically after all the samples have been taken. This only affects the displaying of the signal. If the input signal is clipped, because it is larger than the selected input sensitivity range, and the signal is adjusted, the signal stays clipped. The clipping will show very clearly on the screen. The signal is cut off at the top or bottom. This positioning method is mainly used to move signals which overlap each other on the screen.

The software positioning can be changed by using a menu or by dragging the vertical axes. To adjust the offset by means of a menu, enter the Ch1 menu and select the choice Software offset ... A dialog box appears in which the offset can be entered. The value is in Volts.

The easiest way of adjusting the offset is by dragging the axis. This is done by pressing the left mouse button within the vertical axis, but not over the inverted area (on the top and bottom of the axes) and moving the mouse up or down while keeping the mouse button pressed.

## Magnifying or reducing the vertical axis

The measured values can be magnified or reduced by the software, by the Software gain. By doing this, the vertical axes can be made larger or smaller.

This can be used to display two signals of different sizes on the screen, to be compared more easily. For example: Ch1 is measuring a 5 Volt signal, Ch2 is measuring a 4 volt signal. Set the gain of Ch1 to 0.8 and the gain of Ch2 to 1. Both signals are displayed with the same size.

Another application is to display two signals in a 'split screen mode'. By setting the gain of the channels to 0.5 and the offset of Ch1 2 divs up and the offset of Ch2 2 divs down, the channels are displayed separately.

The software gain can be changed by using a menu or by dragging the vertical axes. To adjust the software gain through the menu, enter the Ch1 menu and select the choice Software gain... A dialog box appears in which the gain can be entered. The value of the gain has to be between 0.25 and 4, where values smaller than 1 make the signal appear smaller and values larger than 1 make the signal appear larger. To change the software gain by dragging the vertical axis, place the mouse pointer over either the upper most or the lower most label of the vertical axis. These labels are drawn inverted. When the mouse pointer is over one of these labels, press the left mouse button and drag the label upwards or downwards, thereby enlarging or shrinking the axis and the signal.

## Inverting a channel

With the software it is possible to invert the signal of a channel. This software invert multiplies all measured values by -1 and adapts the values along the vertical axis accordingly.

There are three ways to invert a signal: by means of a menu, by means the keyboard and by pressing the invert button.

The Invert menu is found in the Ch1 menu. In this menu two choices are available to switch the invert on or off.

A few keys have been reserved to set the invert:

<i>	toggles the invert of channel 1
<Shift+i>	toggles the invert of channel 2

In the tool bar for each channel an invert button is found, the invert button for Ch1 at the left hand side of the screen, for Ch2 at the right hand side of the screen. The picture at the button indicates the state of the invert setting. The left button of the two buttons below is indicating Invert On, the right button is indicating Invert Off.



## The channel view button

For controlling the software offset and the software gain some pre defined settings are available by pressing the channel view button.

These settings are:

- offset = 0 and gain = 1 (using full screen)
- offset = +0.5 input sensitivity and gain = 0.5 (upper half of the screen)
- offset = -0.5 input sensitivity and gain = 0.5 (lower half of the screen)
- offset and gain have the last value set by the user

These settings are available by pressing the channel view button. Each setting has its own picture on the button. The image below displays the above mentioned settings from left to right.



The image at the button indicates the current settings. Each click on the button will select the next state. If the offset or gain are changed by other means and one of the predefined states is obtained, the button will go to that state as well, otherwise user is displayed.

The channel view button for channel 1 is found at the left hand side of the display, for channel 2 at the right hand side of the display.

## Changing the units of the vertical axis

Along the vertical axis, normally the voltage of the input signal is projected. The unit of measure is then Volt and the numbers along the axis correspond with the size of the input signal.

The oscilloscope allows you to create another definition of the vertical axis.

By means of the choice Units of measure from the Ch1 menu the units of measure of the vertical axis can be changed.

After activation, a popup menu with 13 choices appears. The unit that is selected, is displayed at the top of the vertical axis. It is also placed on the paper when printed out. When activating the choice User defined ... a dialog box appears, in which a custom unit can be entered, with a maximum length of 5 characters.

The Units of measure popup menu can also be called by clicking the right mouse button over the vertical axes.

If you are measuring e.g. with a temperature probe, you can choose DEGREE C for the units.

By activating the choice Units gain... from the Ch1 menu, a multiplication factor can be entered for the units of the vertical axis. A dialog box appears in which the factor can be entered. The numbers along the vertical axis are multiplied by the factor and replaced by the result.

If you are using a 10x probe, you can by means of units gain enter the value 10, to get correct numbers along the vertical axis.

Another application is when measuring with a temperature probe. Suppose the temperature probe gives a voltage change of 1 volt at a temperature change of 5 degrees. By entering the factor 5, the temperature change is displayed directly in the number of degrees.

It is also possible to add an offset to the numbers of the vertical axis. In the dialog box called by activating Units off-set... from the Ch1 menu, an offset can be entered. This offset is added to the numbers along the vertical axis. Nothing is done to the signal.

An application for this is the temperature measurement. Suppose the output voltage of the temperature probe at 0 C is +10 Volt. By entering a units offset of -10, the zero level of the axis is changed in a way that the absolute temperature is directly readable.

### Mouse sensitivity for the vertical axis

For dragging the vertical axis, the mouse sensitivity can be adjusted from course to fine, from 1 step per division to 25 steps per division.

When a mouse sensitivity of 1 step per division is selected, each mouse movement will make the software offset become one division larger or smaller. When the software gain is adjusted, it will result in a two division smaller or larger signal size (one division at the top and one division at the bottom of the signal).

When the mouse sensitivity is set to 25 steps per division each mouse movement will result in an offset change of 1/25th division or a signal size change of 2/25th division.

The mouse sensitivity can be set by selecting the item Mouse sensitivity vertical axis from the Ch1 menu.

### Setting up the time axis

Along the horizontal axis of an oscilloscope the measuring time is projected. The measuring time is determined by the number of samples that is measured and the sample frequency

### Setting the sample frequency

The sample frequency determines the speed with which the instrument will take samples of the input signal(s).

The sample frequency can be set in two ways: using the menu or by means of special function keys.

To change the sample frequency using the menu, enter the Time base menu and select the Sample frequency item. A new menu will appear with all possible sample frequency settings. Together with the sample frequency the maximum total measure time with the current record length and a pre trigger value of 0% is given. The Time base menu can also be entered by pressing the right mouse button over one of the time base controls or read outs.

The last item in the menu is a user defined setting. Activating this item will bring up a dialog box in which a self defined sample frequency (within the instrument hardware limitations) can be entered. Due to the way the sample frequency is generated in the hardware, not all frequencies can be generated. If an invalid sample frequency is entered, the closest valid sample frequency will be generated.

The following keys can be used to change the sample frequency:

<F3>	Set the sample frequency one step slower
<F4>	Set the sample frequency one step faster

*Note: Because the record length, the number of post samples and the record viewing gain are free settable, it is very difficult to set the time base in a certain number of sec/div. Therefor the sampling frequency is set.*

### Setting the record length

The oscilloscope takes a number of samples to display the input signal. This number of samples is called the Record length. To change the record length, two ways are available: by means of a menu or by means of function keys.

To change the record length using the menu, enter the Time base menu and select the Record length item. Another menu will pop up, containing the available record lengths and an item for a user defined record length. Selecting this item will bring up a dialog box in which a self defined record length (within the instrument hardware limits) can be

entered. The Time base menu can also be entered by pressing the right mouse button over one of the time base controls or read outs.

For changing the record length the following function keys are available:

<F11>	Select a shorter record length
<F12>	Select a longer record length

Selecting another record length will change the total measure time, but the pre trigger value will keep the same value.

### Setting the pre trigger value

The oscilloscope is equipped with a trigger system. This trigger system monitors the input signal(s) and determines, based on the settings of the trigger system and the input signal, when measuring and displaying the signal(s) has to start.

The moment of trigger, the trigger point, can be placed at any position in the record with measured data. This will result in a certain number of samples before the trigger point (pre samples) and a number of samples after the trigger point (post samples). Pre and post samples together form the record length. The position of the trigger point is referred to as the pre trigger value, indicating how many pre samples are recorded in the total record. Usually (and default) this number is given in a percentage of the record length, but it can be given in a number of samples or in a time value as well.

To change the pre trigger value, two ways are possible: by a menu or with the special trigger position scroll bar, which is located below the signal display.



This special scroll bar represents the complete record and the slider (^) represents the trigger point. In the above picture, the trigger point is set to 25%. To change the trigger point, place your mouse pointer over the arrow (the mouse pointer will change shape), press the left mouse button and drag the slider to the position you want it to have. When one of the small buttons at the left hand side or the right hand side of the scroll bar are clicked, the pre trigger value will change by 1%.

When the mouse is clicked within the scroll bar, but not on the slider, the pre trigger value will increase or decrease by 5%, depending on whether the mouse is clicked to the right or to the left of the slider. When the mouse button remains pressed, an auto repeat function will be activated.

To change the pre trigger value by using the menu, enter the Time base menu and select the Pre trigger value item. this will bring up a menu with 5 pre trigger value options and a user defined option.

The pre trigger value options are default given in a percentage, but can be given in a number of samples or in time. To do that, first the user defined option has to be selected. The Time base menu can also be entered by pressing the right mouse button over one of the time base controls or read outs.

When the user defined option is selected, a dialog box appears. In this dialog box can be selected in what unit the pre trigger value has to be entered (percentage, samples or time) and what the pre trigger value has to be. When the dialog box is closed using the OK button, the entered value is accepted and the selected unit will be used the next time the pre trigger value menu is called.

### Mouse sensitivity pre trigger position adjustment

For adjusting the pre trigger value by dragging the slider of the pre trigger position scroll bar, the mouse sensitivity can be adjusted from course to fine, from 4 steps per screen to 100 steps per screen.

When a mouse sensitivity of 4 steps per screen is selected, the pre trigger position scroll bar is divided in 4 positions. Each mouse movement will adjust the pre trigger value to the next position in the direction the mouse is moved. So the available pre trigger positions are 0%, 25%, 50%, 75% and 100%.

When a mouse sensitivity of 10 steps per screen is selected, the available pre trigger positions are 0%, 10%, 20%, 30%, ..., 90% and 100%.

When the mouse sensitivity is set to 100 steps per screen each mouse movement will result in a pre trigger value change of 1%.

**Note:** The mouse sensitivity does not affect adjusting the pre trigger value by clicking the buttons of the scroll bar or by clicking in the scrollbar outside the slider.

To set the Mouse sensitivity, enter the Time base menu and select Mouse sensitivity pre trigger position adjustment. In the menu that pops up, the choices are available.

## Magnifying the horizontal axis

The software supports a wide range of record lengths which can all be viewed completely. A special displaying algorithm guarantees that when-ever the record length is larger than the display size, no amplitude information will be lost.

To take a closer look at a smaller part of the record, that part can be magnified to fill the screen. This is done with the record view gain, which is completely variable between 1.0 (full record view) and a value depending on the instrument window width (about 40 - about 80).

Adjusting the record view gain can be done through a menu and by means of the special record view scroll bar.

To change the record view gain by means of a menu, enter the Time base menu and select the Record view gain item. This will bring up a dialog box in which the requested record view gain can be entered. The Time base menu can also be entered by pressing the right mouse button over one of the time base controls or read outs.

## Setting the horizontal position of the signal(s)

When a record view gain > 1 is selected, only a part of the record is visible. To view a certain part of the record, a record view offset can be applied.

Applying a record view offset can be done by means of the special record view scroll bar and by means of a menu.

To use the menu, enter the Time base menu and select the Record view offset item. A dialog box will appear in which the record view offset can be entered, in seconds. When the trigger position is not equal to 0, a negative value will make pre samples visible, a positive value will make post samples visible. The Time base menu can also be entered by pressing the right mouse button over one of the time base controls or read outs.

When the record view gain is equal to 1.0, the Record view offset item is not available in the menu.

## The record view scroll bar

For an easy way of adjusting the record view gain and the record view offset, the oscilloscope is equipped with a special record view scroll bar, located under the signal display.



The dark gray part of the scroll bar represents the measured record, the slider (light gray) represents the visible part of the record. The red lines represent the position of the cursors. If the cursors are not switched on, no red lines are visible.

To change the record view gain, place the mouse pointer on the left hand side or the right hand side of the slider (the mouse pointer will change shape), click the left mouse button and drag the side of the slider to the position you want. Simultaneously the record view gain will be adjusted and the signal in the display changes accordingly.

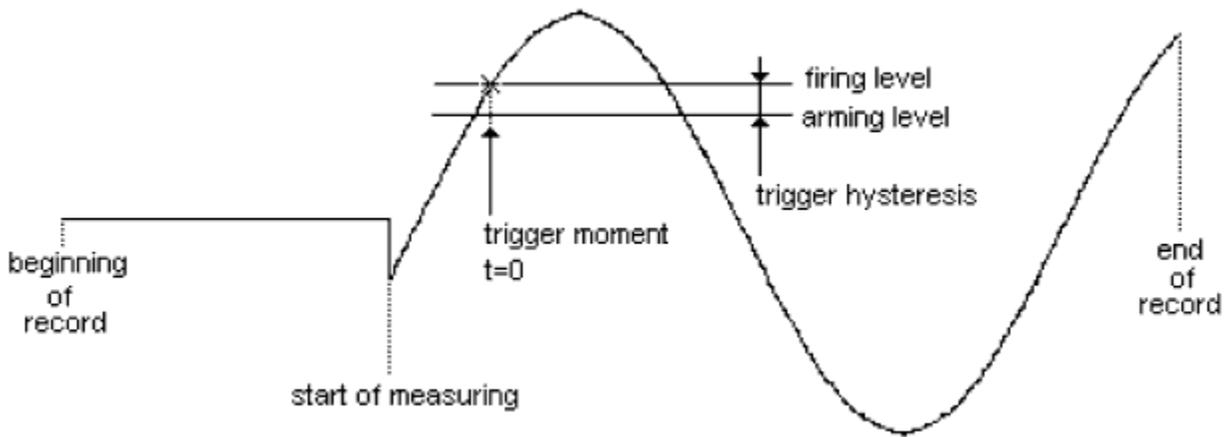
**Note:** The minimum slider width is 10 display pixels and the scroll bar width is dependent of the window size, so the maximum gain is limited to (scroll bar width in pixels / 10). With a large window, the record view gain can be set to a larger value than with a small window size.

To change the record view offset, place the mouse pointer anywhere on the slider, except on the left hand side or the right hand side. The mouse pointer will not change shape. Press the left mouse button and drag the slider to the position you want. The record view offset will be adjusted and the signal in the display changes accordingly.

When the small buttons are clicked, the record view offset is increased or decreased by 1 sample. When the mouse is clicked within the scroll bar but outside the slider, the record view offset is increased or decreased by 5 samples, depending on which side of the scroll bar is clicked. When the mouse button is kept pressed, an auto repeat function is activated.

### Triggering

To be able to examine a signal, the moment of displaying the signal has to be adjustable. Therefore an oscilloscope is equipped with a triggering system. This functions as follows:



The input signal is compared with two levels in the trigger system: the arming level and the firing level. When the input signal passes the arming level, the trigger system is armed. If the input signal passes the firing level, the trigger system becomes active and 'fires' a pulse. This pulse is used to start the display of the signal.

The arming level and the firing level are coupled to each other by the trigger hysteresis and their level is determined by the trigger level. The firing level corresponds to the trigger level. The trigger hysteresis defines at which signal size change can be triggered, the change has to be that large that both levels are passed. With a small trigger hysteresis it is possible to trigger on small signals. If a signal contains a lot of noise, a small trigger hysteresis causes triggering on the noise, instead of the original signal, which gives an unstable display. A trigger hysteresis larger than the noise level is then necessary.

In the illustration a signal and the two levels are displayed. In this case it is triggered on the rising slope of the signal. The signal passes the two levels from low to high. When triggering on the falling slope of a signal, the two levels are swapped. Then the signal has to pass the two levels from high to low, to generate a trigger.

### Selecting the trigger source

The trigger system can have different sources to determine when measuring has to start. These sources are:

Ch1	Trigger on channel 1
Ch2	Trigger on channel 2
External	Trigger on the external trigger input
Ch1 AND Ch2	Trigger on channel 1 and channel 2; the signals on both channels have to meet their trigger settings
Ch1 OR Ch2	Trigger on channel 1 or channel 2; only one of signals has to meet its trigger setting.

To change the trigger source, enter the Trigger menu and select the Source item. In this menu a source can be selected. The trigger menu can be found in the main menu and can be called by clicking the trigger button () or by pressing the right mouse button when the mouse is over one of the trigger controls or readouts.

## Selecting the trigger mode

The trigger system has several trigger modes:

Rising slope	trigger on the rising slope of a signal
Falling slope	trigger on the falling slope of a signal

To change the trigger mode, enter the Trigger menu and select the Trigger mode item. A menu will popup and a trigger mode selection can be made. Another way to change the trigger mode is by using the trigger symbol. Double clicking the left mouse button on the trigger symbol will change the trigger mode from:

Falling slope to Rising slope  
Rising slope to Falling slope

## Setting the trigger level

Adjusting a trigger level can be done in several ways: by means of a menu or by using the special trigger symbol.

To change a trigger level by means of a menu, enter the Trigger menu and select the Level Ch1 ... or Level Ch2 ... item. An input dialog appears in which the trigger level can be entered, in volts.

In this dialog, also a check box for selecting auto level triggering is found. When auto level triggering is selected, the trigger level will be set to a value that lies in the middle between the highest and the lowest value of the previous measurement. This can be very useful with continuously changing signals. When auto level triggering is selected, the trigger symbol will show that with a letter A in the symbol. The level can no longer be adjusted with the symbol.

*Note:* When a trigger source External is selected the trigger level adjustment is disabled.

## Setting the trigger hysteresis

Adjusting a trigger hysteresis can be done in several ways: by means of a menu or by using the special trigger symbol.

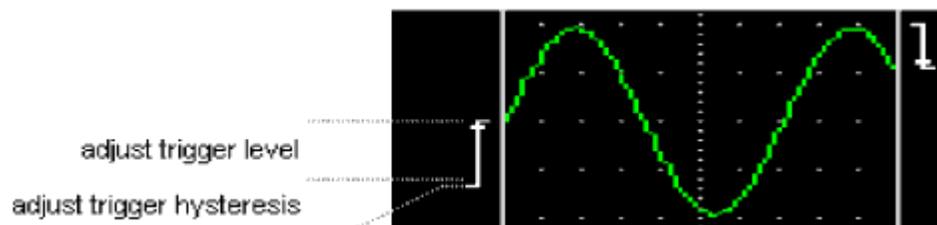
To change a trigger hysteresis by means of a menu, enter the Trigger menu and select the Hysteresis Ch1 ... or Hysteresis Ch2 ... item. An input dialog appears in which the trigger hysteresis can be entered, in volts.

When auto level triggering is selected, the trigger symbol will show that with a letter A in the symbol. The hysteresis can still be adjusted with the symbol.

*Note:* When a trigger source External is selected the trigger hysteresis adjustment is disabled.

## The trigger symbol

The oscilloscope is equipped with a special trigger symbol, that is used to adjust the trigger level, the trigger hysteresis and the trigger mode. The WYSIWYG trigger symbol is found at the left hand side or the right hand side of the signal display, depending on the trigger source.



In the above picture, the trigger mode of the left hand symbol is set to Rising slope and the trigger mode of the right hand symbol is set to Falling slope.

To adjust the trigger level using the trigger symbol, place the mouse pointer on the trigger symbol, in the area that is marked adjust trigger level in the above picture. The mouse pointer will change to a  symbol. Press the left mouse button and drag the trigger symbol to the correct position, while keeping the left mouse button pressed. While dragging, the trigger level is adjusted according the position of the trigger symbol on the screen. When the mouse button is released, trigger level adjustment stops and the normal mouse pointer shape is returned.

To adjust the trigger hysteresis using the trigger symbol, place the mouse pointer on the trigger symbol, in the area that is marked adjust trigger hysteresis in the above picture. The mouse pointer will change to a  symbol. Press the left mouse button and drag the hysteresis level of the trigger symbol to the correct position, while keeping the left mouse button pressed. While dragging, the trigger hysteresis is adjusted according the size of the trigger symbol. When the mouse button is released, trigger hysteresis adjustment stops and the normal mouse pointer shape is returned.

To adjust the trigger mode using the trigger symbol, place the mouse pointer on the trigger symbol and double click the left mouse button. The trigger mode will change from:

Falling slope to Rising slope  
Rising slope to Falling slope

## The trigger time out

When the input signal does not meet the trigger conditions, no new measurement data will be displayed and the last displayed signal will remain on the screen. This can be very inconvenient with unknown signals, since it is not known how the trigger system has to be set. To provide in that case for a representation of the input signal on the screen, a trigger time out setting is available.

With the time out setting it is determined how long the delay will be for a trigger pulse. If that time expires, a trigger pulse is forced by the software and the available data will be displayed. This will not result in a stable display, but it will give an impression how the trigger has to be set.

To set the trigger time out, enter the Trigger menu and select the Time out ... item. An input dialog will come up in which the time out time can be entered, in seconds.

When a time of zero is entered, there will be no trigger time out, but no triggering as well. As soon as a signal is displayed, the instrument will be set up for a new measurement and immediately a trigger pulse is generated, without waiting for a trigger. This can be useful when e.g. a slowly changing DC level has to be monitored, on which can't be triggered.

In the dialog also a check box for an infinite time out is available. When this box is checked, the software will keep waiting for the input signal to meet the trigger settings, without ever forcing a trigger pulse. This can be very useful in one shot measurements and with auto disk measuring.

## Performing measurements

When the oscilloscope is completely set up, measurements can be performed.

By default the oscilloscope is always measuring and displaying. Changes that are made in the settings can be made while measuring and will be executed while measuring, so the effects are immediately visible.

## Stopping the measurements

To avoid a new screen of measurement data replacing an old screen, measuring and displaying of data can be stopped.

This can be done in several ways: by pressing the Pause button () , by pressing the <S> key or by activating the item Stop continuous measuring from the Measure menu.

When measuring is stopped, the latest measured data will remain on the screen. The Pause button changes to a Play button () and the One shot button () becomes available.

The measured signal can now be examined.

To perform a single measurement, press the One shot button, press the <O> key or activate the Perform a single measurement item from the Measure menu.

Continuous measuring can be started again by pressing the Play button, the <S> key or the item Start continuous measuring from the Measure menu. The One shot button will be disabled again.

## Perform Auto disk measurements

The oscilloscope can store measurement data automatically on disk. This is very useful when e.g. an interference glitch has to be measured. Suppose that at unpredictable times a glitch occurs. By using the auto disk measurements, the oscilloscope will wait for the glitch to occur. When the glitch occurs, it is measured and the data is stored on disk. The time and date of the measurement are stored as well.

To activate the auto disk function, first set up the oscilloscope to the appropriate settings (input sensitivity, sample frequency, record length etc.) Then, enter the Measure menu and select the Auto disk item. A special auto disk dialog will appear. In this dialog, a target disk, a target directory and a name for the files can be selected. For the file name, only 5 characters can be entered, three characters will be added by the software for a file serial number. At the bottom of the dialog a bar graph is displaying the available disk space on the selected drive.

The trigger system of the oscilloscope has a time out function. When the input signal does not meet the trigger conditions within a certain time, a trigger is forced. This is used to see an unknown signal. This feature can be unwanted in the auto disk function, therefore a Time out infinite check mark is available to set the time out to infinite. Then the oscilloscope will keep waiting for the input signal to meet the trigger conditions.

When all is set, press the Start button to start the auto disk measurements. The Cancel button will change to a Stop button and measuring takes place.

The auto disk measurements stop when the Stop button is pressed or when the disk is full.

**Note:** In the root of a disk only 128 files can be stored. A RAM drive has a user definable number of entries (default 64) and can store no more files than there are entries available. The auto disk function will stop with a disk full error when this number of files are written. However, DOS or Windows might still report a certain amount of free space on the disk. To be able to store more files, create a directory on the disk and store the files in that directory. A directory can contain many more files.

**Note:** The auto disk function can store only 999 files, due to the three characters that are used for creating the file serial number.

## Auto setup

When an unknown signal is connected to the oscilloscope, the easiest way to setup the oscilloscope is by using the Auto setup function.

The Auto setup function is activated by selecting the item Auto setup from the Measure menu or by pressing the Auto setup button ()

This function sets the sensitivity for both channels to auto ranging, so the optimal signal size is obtained. Also the trigger level of the channel that is the trigger source is set to auto level, to insure there is always a trigger.

Then the time base is set in a way that the signal at the channel that is triggered on, is displayed with 2 to 6 periods.

The auto setup function does nothing to the signal coupling, the measure mode (ch1, ch2 or ch1 and ch2) or the trigger source.

## Zooming

When a certain part of the screen has to be enlarged to full screen, the zoom function is the easiest way to do that.

The zoom function is activated by pressing the zoom button (). This button stays down until the zoom function is finished.

When the zoom function is activated, a rectangular area in the signal display can be selected with the mouse. Place the mouse at one of the corners of the rectangle, press the left mouse button, keep it pressed and drag the mouse to the opposite corner of the rectangle. While dragging, a dashed rectangle is drawn. When the rectangle has the correct size, release the mouse button. The zooming function will finish and the software offset and gain of both channels and the record view offset and gain are adjusted to the values that result in viewing the selected part of the screen.

## Performing cursor measurements

To perform measurements on the measured data, two pairs of cursors are available. Each cursor pair exists of a horizontal and a vertical cursor that can be placed anywhere inside or outside the signal display. Where the two cursors cross each other, a link is made. For the left cursor this link is pink, for the right cursor this link is black.

## Switching on cursors

To switch on cursors, enter the Cursors menu and select the Large cursors item or the Small cursors item. Large cursors are drawn from the left to the right hand side of the signal display or from the top to the bottom of the signal display. Small cursors only form a small cross. The Cursors menu can be found in the main menu or can be called by clicking the right mouse button in the signal display. When the cursors are switched on, a special cursor readout window appears as well.

The cursors are waveform based. This means that the cursor position is related to the waveform. If the record view offset or record view gain is changed, the cursor position will remain at the same position in the waveform. If the input sensitivity, the software offset or software gain is changed, the cursor will stay at the same voltage level. Because of this, cursors can get off screen. In the record view scroll bar two red lines are drawn to indicate the position of the cursors in the record.

To position a cursor, bring the mouse pointer on the cursor to position. The mouse pointer will change shape:

-  position a horizontal cursor
-  position a vertical cursor
-  position a cursor pair

*Hint:* To position an off screen cursor, bring the mouse pointer to the edge of the signal display at which the cursor exceeds. The mouse pointer will change shape.

Now press the left mouse button and drag the cursor to the correct position. When that is reached, release the left mouse button. The original mouse pointer shape is restored.

## Setting up the cursor measurements

With the cursors it is possible to measure the input signal(s) in several ways. To set up the cursor measurements, press the settings button at the cursor readout window (the right most button at the left top of the window caption) A menu appears, containing the following options. The menu can also be called by pressing the right mouse button over the table.

## Voltmeter measurements

All measurements which can be selected in the voltmeter measurements submenu are also available in the voltmeter. The only difference between the measurements is that the voltmeter uses the previous two measurement values and

---

the current measured values which are added together and meaned before the value is displayed. For a complete description of the measurements, see the chapter on the voltmeter.

**Time**

**Rise time left:** the rise time of the signal at the position of the left cursor is determined. The rise time is determined by finding the closest zero crossing  $((\text{max} + \text{min}) / 2)$  relative to the cursor position. Next the 10% and 90% values are calculated. Now, the 10% and 90% points are searched left and right of the zero crossing point. By using the number of samples between the 10% and 90% points, the rise time is calculated.

**Rise time right:** the rise time at the position of the right cursor is determined.

**Sample time left:** using the left cursor position, the time on which the selected sample is taken will be displayed.

**Sample time right:** using the right cursor position, the time on which the selected sample is taken will be displayed.

**Sample time difference:** the difference between the right and the left cursor sample time is determined.

**Cursor frequency:** using the number of samples between the left and the right cursor, the frequency is calculated. The calculation is done using the following formula:  $(\text{number of samples between cursors} / \text{total number of samples}) * \text{sample frequency}$ . To determine a correct frequency always select one period or multiply the frequency by the number of selected periods for greater precision.

**Voltage**

**Voltage left:** the voltage of the left cursor will be displayed.

**Voltage right:** the voltage of the right cursor will be displayed.

**Voltage difference:** the difference between the voltage of the left and the voltage of the right cursor is calculated. Selecting the min and max of the signal, the top-top value is calculated.

**Slew rate**

**Slew rate left:** the slew rate of the signal at the position of the left cursor is determined. The slew rate is the voltage level which a signal drops or gains per millisecond.

**Slew rate right:** the slew rate of the signal at the position of the right cursor is determined.

**Phase difference**

These measurements are only available when both channels are switched on. They measure the phase angle difference between the signals on channel 1 and channel 2. When the software can not detect a complete period of the signal on both channels or when the frequency of the two signals are different, no phase difference is determined.

**Degrees:** The angle is presented in degrees

**Radials:** The angle is presented in radials

**cos():** The cosine value of the angle is presented

**Set impedance value**

The dBm and power measurements require a reference resistance. The value of the resistance can be set using this menu item. The value has to be entered in Ohm and is default 600.

**Automatic correct points**

By selecting this option, the automatic correction of the cursors to the nearest zero crossing is enabled or disabled. The right cursor is positioned one or more periods from the left cursor. To determine a frequency or duty cycle, position the right cursor at the end of the second period or further for correct results.

**Automatic tracking cursors**

If this option is enabled, the voltage levels of the cursors are automatically set to the voltage levels of the sample numbers which correspond to the left and right cursor.

**Calculate values**

If this option is disabled, no new calculations will be done until the option is enabled.

**Show measurement name**

### **Show measured value**

#### **Show measurement unit**

To reduce the space used by the window on the screen, the measurement name, value and/or unit can be hidden. This results in the disappearance of the corresponding column which results in a smaller window.

#### **Display font**

By selecting this option, the font which is used to display the strings and values can be changed. The font settings which can be changed are: font name, font style, font size and the font color.

#### **Background color**

By selecting this option, the background color can be altered. Some font colors (e.g. light green) are not showing up very good on a white background. Change the background color to e.g. black for a better result.

### **Selecting the active channel**

The cursors can only calculate values for one channel.

The buttons in the left top of the window caption bar can be used to select the channel from which data is used to calculate the selected measurements.

There are two ways to change between channels, pressing the buttons with the mouse or by using the corresponding number on the keyboard.

### **Showing or hiding the cursor readout**

The left button of the two in the right top corner of the window caption bar can be used to hide () or show () the grid with measured data. When hiding the data, the height of the window equals the height of the window caption bar.

### **Switching off cursors**

To switch off cursors, enter the Cursors menu and select the No cursors item. The Cursors menu can be found in the main menu or can be called by clicking the right mouse button in the signal display.

When the cursors are switched off, the cursor readout window disappear as well.

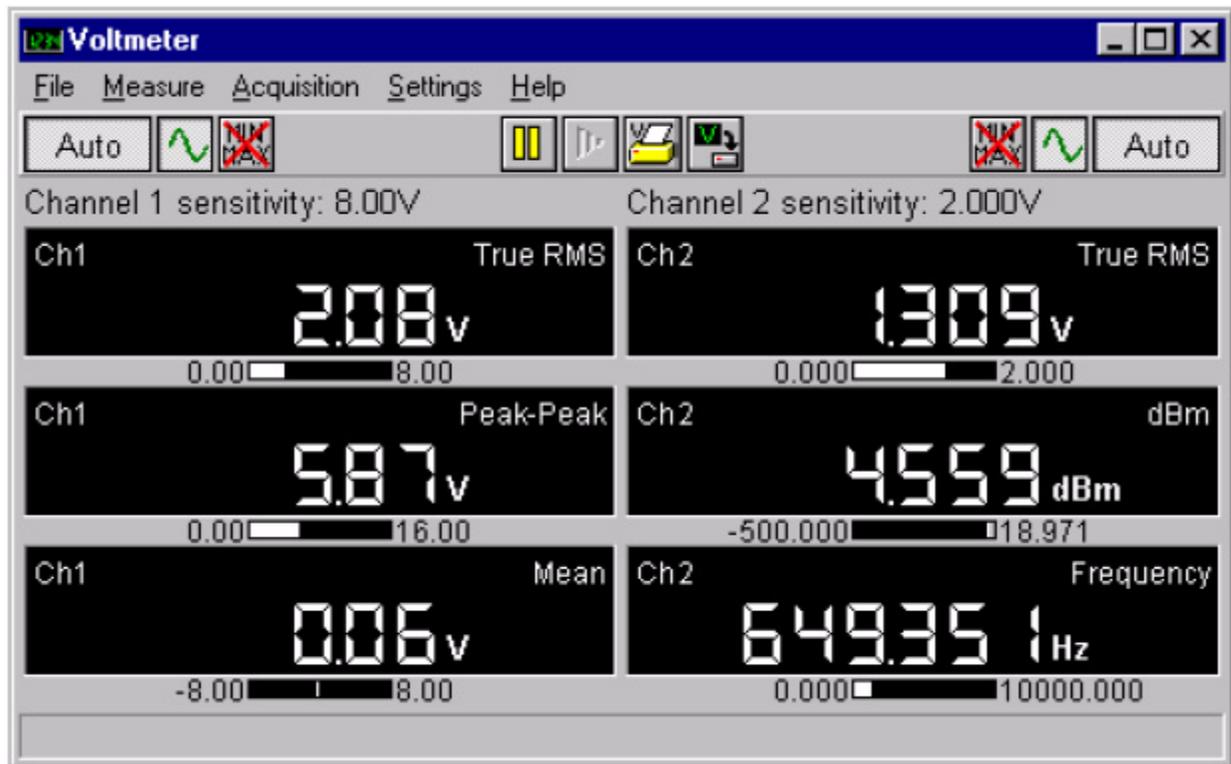
The cursors can also be switched off by clicking the close button () in the cursor readout window.

# 7. The Voltmeter

If from the input signals only the size is important and not the time information or frequency components, a voltmeter is a suitable instrument to measure with. The software is equipped with a two channel digital voltmeter. The voltmeter functions as follows:

- A measurement is performed (minimal 200 samples).
- The measured data is processed, e.g. for calculating the RMS value or the mean value. Eleven different operations are available.
- The calculated values are displayed, e.g. add CH1 and CH2 and display on channel one. Sixteen different display methods are available.

For each channel the voltmeter has up to three displays to present the measured and calculated values. The value displayed in a display is fully configurable. Also for each display a bar graph is available, to give a quick overview of the signal size in relation to the input range.

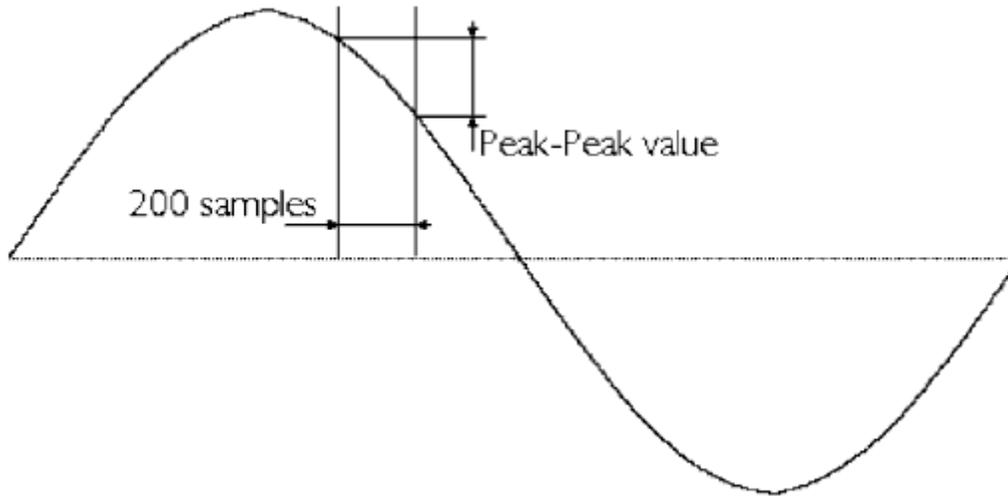


Of all four instruments, the voltmeter has the lowest settings priority. This means that if for example the oscilloscope is active, the record length, pre samples and post samples of the oscilloscope are used in the voltmeter. The voltmeter only uses post samples to perform the calculations. If less than 200 post samples are available, no calculations are done and a message is displayed. The only setting that can be changed by the voltmeter when the oscilloscope or spectrum analyzer is active, is the sample frequency.

## ***The measure system of the voltmeter***

The voltmeter takes 200 samples of the input signal with a certain sample frequency. These 200 samples are used to perform several calculations.

The voltmeter has a limited frequency range in which the measurements are performed correctly. If the periodical time of the input signal is much larger than the measuring time, a wrong value is measured. See the next illustration.



If the time in which the 200 samples are taken is too long in relation to the periodical time of the input signal, aliasing can occur (see also the chapter about the oscilloscope).

### Setting the frequency range

To avoid errors like the ones mentioned above, the frequency range of the voltmeter has to be set properly. The sampling frequency is then changed. By activating the choice Frequency range from the Settings menu, a popup menu appears, in which several frequencies are available. The frequencies indicate the center frequency of a frequency range in which the voltmeter measures properly. The size of the frequency range is determined from  $0.2 \times f_{\text{center}}$  to  $5 \times f_{\text{center}}$ .

When Auto freq. is chosen, the voltmeter determines the best frequency setting by itself. For each measurement the frequency has to be determined, resulting in a longer time before updating the displays.

*Note: The Auto frequency setting overrides the sample frequency setting in the oscilloscope and the spectrum analyzer. This will result in a less usable display in those instruments.*

To avoid this, the choice Scope freq. is available in the menu. The voltmeter will use the same sample frequency as the oscilloscope or spectrum analyzer.

### Displaying channels

The voltmeter has several modes to display the two channels. To select the measure mode, enter the Settings menu and select Channels. Use the Measure channel option to enable or disable measuring from a channel. Measuring takes place from at least one channel.

When only one channel is measured, the controls of the other channel are disabled or removed from the display.

### Setting up a channel

The channels of the voltmeter can be setup in several ways. Both channels have the same settings, so only channel 1 is discussed here.

### Setting the input range

In the voltmeter, the input range or sensitivity can be set through the channels settings dialog.

The channels setting dialog is called by selecting the Channels option from the Settings menu. In this dialog, the input sensitivities are available, as well as an autoranging setting.

To switch on Autoranging for the Voltmeter, a special button is available. When auto ranging is switched on, the button caption is Auto, otherwise it holds the current set input sensitivity. Pressing the right mouse button on this button will bring up a menu with all sensitivities.

When the input signal is larger than the selected input range, '-----' is displayed.

**Note:** When Autoranging is switched on, several measurements might be needed to set the input sensitivity properly, depending on the input signal.

Another way to set the sensitivity is by using the keyboard. A few keys have been reserved for setting the input sensitivity of the channels. These keys are:

<F5>	Increase Ch1 sensitivity
<F6>	Decrease Ch1 sensitivity
<F7>	Increase Ch2 sensitivity
<F8>	Decrease Ch2 sensitivity

### Setting the input coupling

The input signal can be coupled directly to the ADC (DC) or through a capacitor to the ADC (AC). In the hardware, a capacitor is switched into (AC) or out of (DC) the signal path.

In COUPLING DC the complete signal (AC+DC) at the input is passed through. In COUPLING AC only the AC component of the input signal is passed through. It is then possible to examine e.g. a  $\pm 20$  mV ripple on a 30 V DC voltage, since only the ripple is passed through and the DC voltage not. AC voltages with a frequency lower than  $\pm 4$  Hz are not coupled correctly.

There are three different ways to set the coupling. Two ways are: by clicking the coupling button or by pressing keys. The third way is through the channel settings dialog. The channels setting dialog is called by selecting the Channels option from the Settings menu. In this dialog, the signal coupling setting is available.

In the tool bar for each channel a coupling button is found, the coupling button for Ch1 at the left hand side of the screen, for Ch2 at the right hand side of the screen. The picture at the button indicates the state of the coupling setting. The left button of the two buttons below is indicating AC coupling, the right button is indicating DC coupling.



The current coupling state is displayed in the coupling button.

The keys which can be used to change the signal coupling are:

<A>	sets the coupling of channel 1 to AC
<D>	sets the coupling of channel 1 to DC
<Shift+A>	sets the coupling of channel 2 to AC
<Shift+D>	sets the coupling of channel 2 to DC

### Setting a DC hardware offset

The signal offset is adjustable in two ways: a hardware method and a (per display) software method.

Using the hardware method, an adjustable DC voltage is added before sampling the input signal. The hardware method can be realized by selecting the Channels option from the Settings menu. Type the offset in the DC hardware offset edit box. This voltage will be added to the input signal. If a voltage has to be subtracted, enter a negative value.

## Configuring the displays

The voltmeter can display the measured values in several different ways, in up to six displays. Below the display, a bargraph is showing the value in the display in a graphic format.

By activating the choice Displays from the Settings menu, a dialog appears in which the settings of all the displays can be changed. This method works always, even if the display and bargraph for which the settings are changed are invisible.

The screenshot shows the 'Display settings' dialog box with the following configuration:

- Measurement: True RMS
- Display measured value: Ch1
- Units of measure: Volt
- Numbers precision: Auto
- Units per measurement unit: 1
- Set low value: -20
- Set high value: 20
- Set impedance value: 600
- Set relative value: 0
- Relative measuring:
- Show display:
- Show bar graph:

At the bottom, the tabs are: Display 1,1 / Display 1,2 / Display 2,1 / Display 2,2 / Display 3,1 / Display 3,2.

These settings can also be done by placing the mouse cursor over a display and pressing the right mouse button. A popup menu appears with all settings for that display. This method only works if the display or bargraph is visible.

All settings applied to a display are also applied to the corresponding bargraph.

The following explanations expect that the Display settings dialog is displayed and active.

### Switching displays on and off

A display can be made visible by placing a checkmark in front of the Show display string. If a display is turned on, while the matching channel is not measured, the display will not be showed.

If all six displays are visible, the layout is as following:

- The left hand side displays belong to channel 1.
- The right hand side displays belong to channel 2.

### Switching bargraphs on and off

Each display has a bargraph that can be switched on or off independently from the displays. The bargraphs can be switched on or off by tagging or untagging the Show bargraph option.

## Processing the measured value

The software can process the measured value in 11 different ways. By activating the Measurement selector, the following calculations can be set:

TRUE RMS:	The true RMS value of the input signal is calculated. The true RMS value (Root Mean Square) is the value of a random voltage, corresponding to a DC voltage, dissipating the same power in a resistance in the same amount of time. So an AC voltage of 1 volt RMS corresponds to a DC voltage of 1 volt. (Most -cheap- voltmeters only display, when measuring AC voltage, the correct value when the input signal is a true sine wave and usually in a limited frequency range. This instrument always displays the correct value.)
Peak-peak:	The peak-peak value of the input signal is determined. The peak-peak value of a DC voltage is zero. The peak-peak value of a true sine wave with a RMS value of 1 volt is 2.828 volt.
Mean:	The mean value of the input signal is determined. The mean value of a varying voltage corresponds with the value of a DC voltage across the same load. The mean value of a 1 volt DC voltage is 1 volt. The mean value of a true sine wave AC voltage is 0.
Max:	The maximum of the input signal is determined. The maximum value of a 1 volt DC voltage is 1 volt. The maximum value of a 1 volt AC sine wave voltage is 1.414 volt. This is determined from the 200 samples.
Min:	The minimum of the input signal is determined. The minimum value of a 1 volt DC voltage is 1 volt. The minimum value of a 1 volt AC sine wave is -1.414 volt. Determined from the 200 samples.
dBm:	The value of the input signal is converted to dB. This is done using the formula: $dBm = 10 \times \log \frac{1000 \times value^2}{reference\ resistance}$ 'value' is the RMS value of the input signal. The reference resistance can be set using the edit box set impedance value.
Power:	The value of the input signal is converted to a power, using the formula: $POWER = \frac{value^2}{reference\ resistance}$ 'value' is the RMS value of the input signal. The reference resistance can be set using the edit box Set impedance value.
Crest:	The value of the input signal is converted to a crest factor, using the formula: $CREST = \frac{peak\ value}{RMS - Value}$ 'peak value' is the highest voltage of the input signal with respect to the zero level.
Frequency:	The frequency of the input signal is determined. Frequencies lower than 0.5 Hz, are displayed as 0 Hz. The frequency can be determined for one channel at a time. If the frequency is determined for channel 1, the frequency of channel 2 can't be determined.
Duty cycle:	The duty cycle of the input signal is determined. The duty cycle indicates how much of a signal is low/high (depending on the duty cycle setting) in relation to the periodic time of the signal. The number is given in percent.

Moment. value:	Displays the voltage level of the first post sample.
----------------	--

*Note: If a frequency or duty cycle measurement is set, the voltmeter is slower on all displays, because for determination of the frequency or duty cycle more measurements have to be done.*

### Displaying the calculated value

For displaying the measured and calculated values, 16 methods are available. Activate the Display measured value selector to set the following possibilities:

CH1:	The measured value of Ch1 is displayed.
CH2:	The measured value of Ch2 is displayed.
CH1*CH2:	The values of Ch1 and Ch2 are multiplied and the result is displayed.
CH1/CH2:	The value of Ch1 is divided by the value of Ch2. The result is displayed.
CH2/CH1:	The value of Ch2 is divided by the value of Ch1. The result is displayed.
CH1-CH2:	The value of Ch2 is subtracted from the value of Ch1. The result is displayed.
CH2-CH1:	The value of Ch1 is subtracted from the value of Ch2. The result is displayed.
CH1+CH2:	The values of Ch1 and Ch2 are added and the sum is displayed.
> than HI:	The value is compared to the high value. If the value is higher than the high value, then PASS is displayed, else LO is displayed.
< than LO:	The value is compared to the low value. If the value is lower than the low value, then PASS is displayed, else HI is displayed.
>> COMP:	The value is compared to the high value and the low value. If the value is higher than the high value, HI is displayed. If the value is lower than the low value, LO is displayed. If the value is lower than or equal to the high value and higher than or equal to the low value, PASS is displayed.
<> HI LO:	The value is compared to the high value and the low value. If the measured value is higher than the high value, PASS is displayed. If the value is lower than the low value, PASS is displayed. If the value is lower than or equal to the high value and higher than or equal to the low value, FAIL is displayed.

		DISPLAY			
		> than HI	< than LO	>> COMP	<> HI LO
measured value	HI	PASS	HI	HI	PASS
	LO	LO	HI	PASS	FAIL
	LO	LO	PASS	LO	PASS
MAX:	The maximum measured value ever is displayed.				
MIN:	The minimum measured value ever is displayed.				
LOG(1/2):	The result of the calculation $20 \times \log \frac{\text{measured valueCH1}}{\text{measured valueCH2}}$ is displayed.				
LOG(2/1):	The result of the calculation $20 \times \log \frac{\text{measured valueCH2}}{\text{measured valueCH1}}$ is displayed.				

### Change the units of measure

Normally, every type of measurement has its own unit. The voltmeter allows you to change this unit for each display separately. There are twelve predefined values and a user defined value.

The selection can be made with the Units of measure pulldown. The unit which is selected, is displayed in the display. To select a user defined unit of measure, just type a string with a maximum length of five characters.

### Change the units per measurement unit

Using the edit box Units per measurement unit, the number of units per selected unit can be set. In the edit box, the multiplication factor can be entered. The measured value is multiplied by the factor and displayed.

This can be used if e.g. a 1:10 probe is used. To correct the reading, enter a value 10 in the edit box.

### Relative measurements

The voltmeter can take absolute measurements and relative measurements. When measuring absolute, the measurements are related to the 0 level (ground). When measuring relative, the measurements are related to a settable value. This value is subtracted from the measured value by the software. This means that before displaying the value in the display, the relative value is subtracted from the calculated value.

The value can be set by entering a value in the Set relative value edit box. The value has to be entered in volts.

To enable relative measurements, a checkmark has to be placed after Relative measuring. Relative measuring is then enabled.

Relative measurements can be used with e.g. a temperature measurement. Suppose the output voltage of a temperature probe at 0 degrees Celsius equals +10 volt. By entering a relative value of 10 volt, the absolute temperature is directly readable.

## Comparison measurements

For displaying the measured and calculated values, 16 methods are available (See: displaying the measured value). Four of those methods are comparison measurements:

> than HI  
< than LO  
>< COMP  
<> LO HI

These measurements require a high and a low value. By entering a value in the Set high value or Set low value edit boxes, the high and low levels can be set.

*Note:* For easy entering of low and high values, the software does not check whether the high value is larger than the low value. So it is possible to enter a low value which is larger than the high value. Measurements using these values will give useless results.

## Setting the reference impedance

The dBm and power measurements require a reference resistance. The value of the resistance can be set using the Set impedance value edit box. The value has to be entered in Ohm. The default value is 600.

## Setting the number of digits after the decimal separator

With every calculation done on a measurement, a default number of digits after the separator is set (auto setting). This number corresponds with the significant number of figures determined by the hardware. The number of digits after the separator can be changed using the Number precision selector.

*Note:* The displays have a maximum number of 6 digits. When the value in the display is that large that digits before the decimal separator can't be displayed, an arrow pointing to the left is displayed left from the first digit. The displayed digits are correct though.

## Sound settings

With comparison measurements, for each value a sound signal can be set. When this is done, the set limits are indicated audibly.

By activating the choice Sounds from the Settings menu, a dialog appears from which for each display value a sound signal can be set. For each display value can be chosen from: No sound, 100 Hz, 200 Hz, 500 Hz, 1 kHz, 2 kHz, 5 kHz and 10 kHz.

With the choice No sound, no sound signal is generated with the selected display value.

If you are using the PC speaker or a WAV device, it is only possible to make one sound at a time. This means that the sound which is played first, is repeated until the condition is invalid. As soon as the condition is invalid, the sound for the next condition can be played.

*Note:* If your computer is equipped with a digital playback soundcard (a soundcard which can play WAV files), you can also play the sounds through your soundcard. Select the sound device pull down after which you can select wav device from the list. If this option is missing and you have a soundcard, you should install drivers for the soundcard or you don't have a digital playback soundcard.

**Note:** If the computer is equipped with a soundcard, it is also possible to select the MIDI device option. The MIDI device uses the instrument settings as defined in the General MIDI standard. When using MIDI, it is possible to play all four sounds (LO, HI, FAIL and PASS) at one time.

### Resetting the minimum and maximum values

When displaying the measured values, the maximum value ever and/or the minimum value ever can be displayed. These values are kept until the software is closed.

To reset the minimum and maximum values, for each channel a reset button is available in the speedbar () and through the settings, channel menu. By pressing this button, the maximum value is set to the lowest possible measured value. The minimum value is set to the maximum measured value possible.

### Performing measurements

**Hint:** Use the frequency meter to determine the frequency of the input signal.

When the device voltmeter is switched on, and the measurement process is not in one shot mode, measurements take place. The measured values are displayed continuously. If the oscilloscope is active, the trigger conditions also apply to the voltmeter.

### Start and stop measuring

The measuring can be halted by pressing the pause button () or <S> key. The oneshot button () next to the pause button becomes available (<O> key), and the pause button changes to a play button ()

The measuring can also be halted by selecting the Stop continuous measuring option from the Measure menu. The option Perform a single measurement becomes available and the Stop continuous measuring option changes to Start continuous measuring.

### Averaging

Normally the voltmeter adds the two previous measured values with the current measured value and divides it by three. In this way, very small changes do not show up which results in a more stable values.

To switch averaging on or off, enter the Settings menu and select the Mean values option. The check mark in front of the menu item indicates whether averaging is on or off.

### Fast measuring

By selecting the option Fast measuring from the Settings menu, fast measuring is enabled or disabled. Before the fast measuring option can be enabled, you have to set a frequency range other than scope or auto.

When enabling the fast measuring option, the trigger timeout is set to zero, the record length and post samples are set to 200 and the averaging is disabled.

### One shot measurements

A single measurement can be performed by pressing the oneshot button or by selecting the Perform single measurement option from the measure menu.

**Note:** One measurement consists of three successive measurements (except if fast measuring is selected or mean values is disabled, these take one measurement), from which the measured values are averaged. Between the measurements the display is updated. If the voltmeter has to perform an autoranging action or has to change the sample frequency during an oneshot, measurements are taken until all changes to the range and the frequency have been done. Then three measurements are taken with the final settings.

Continuous measuring can be switched on by activating the play button or pressing the <S> key.

### **Storing measurement values on disk or paper**

The voltmeter can store measurements on disk or print them. This is done in a completely different way than the other instruments.

The voltmeter can be set to measure (acquire) at fixed times. The measured value will then be stored on disk or sent to the printer.

The time at which measurements are taken is not 100% guaranteed. The reason for this is the Windows environment which is not a real-time multitasking system. Because of this there is no guarantee that the time between two measurements is the set interval. If this occurs, close all windows programs except the voltmeter and then try again.

### **Setting the time between two measurements**

By activating the choice Time from the Acquisition menu, the time between two measurements can be set. A dialog box appears in which the time in seconds can be entered.

*Note: When a time shorter than 0.5 seconds is entered, the acquisition speed is determined by the changes of the displays.*

### **Setting the hysteresis**

With an acquisition time smaller than 0.5 seconds, the acquisition speed is determined by the changes in the displays. It can occur that the input signal has a value just between two displayable values. In that case the display will constantly change between the two values. To avoid a constant storage to disk or printer in that case, a hysteresis can be set. Only when the input value change exceeds the hysteresis value, the input value will be stored.

By activating the item Hysteresis from the Acquisition menu, a dialog appears in which the hysteresis can be entered. The hysteresis is entered as a percentage of the last measured value. With each new measurement, the size of the hysteresis in volts is recalculated.

The hysteresis can only be set with an acquisition time smaller than 0.5 seconds.

### **Start an acquisition**

By activating the choice Write data from the Acquire menu, a menu is shown with which a selection can be made to write to disk or printer.

The choice Disk will cause the measurement data to be stored on disk. After activating the choice, a save dialog appears in which the drive, directory and name of the file can be typed. If the typed file already exists, an overwrite prompt will appear. The measuring will start after this. At fixed times a measurement is taken and stored on disk. Measuring can be aborted by pressing the cancel button.

The choice Printer will cause the measurement data to be written to the printer. After activating the choice, a print dialog appears from which a printer can be selected. A dialog will appear in which the title of the job can be typed. Now measuring starts. At fixed times a measurement is taken and sent to the printer. Measuring can be aborted by pressing the Cancel button.

The printing and writing can also be activated by pressing the following buttons on the speedbar:



In both modes, the measurement values are preceded by a block of information. In that block is indicated how the channels are set and how the high, low and relative levels are set. Each measurement is preceded by the time and date of the measurement.

If a measured value is larger than the selected input range, '-----' is displayed. This will also be stored on disk or sent to the printer. The date/time format can be set through the program settings dialog.

# 8. The Data Logger

For measuring slowly changing signals (e.g. the temperature change in a room) the data logger is the most suitable instrument.

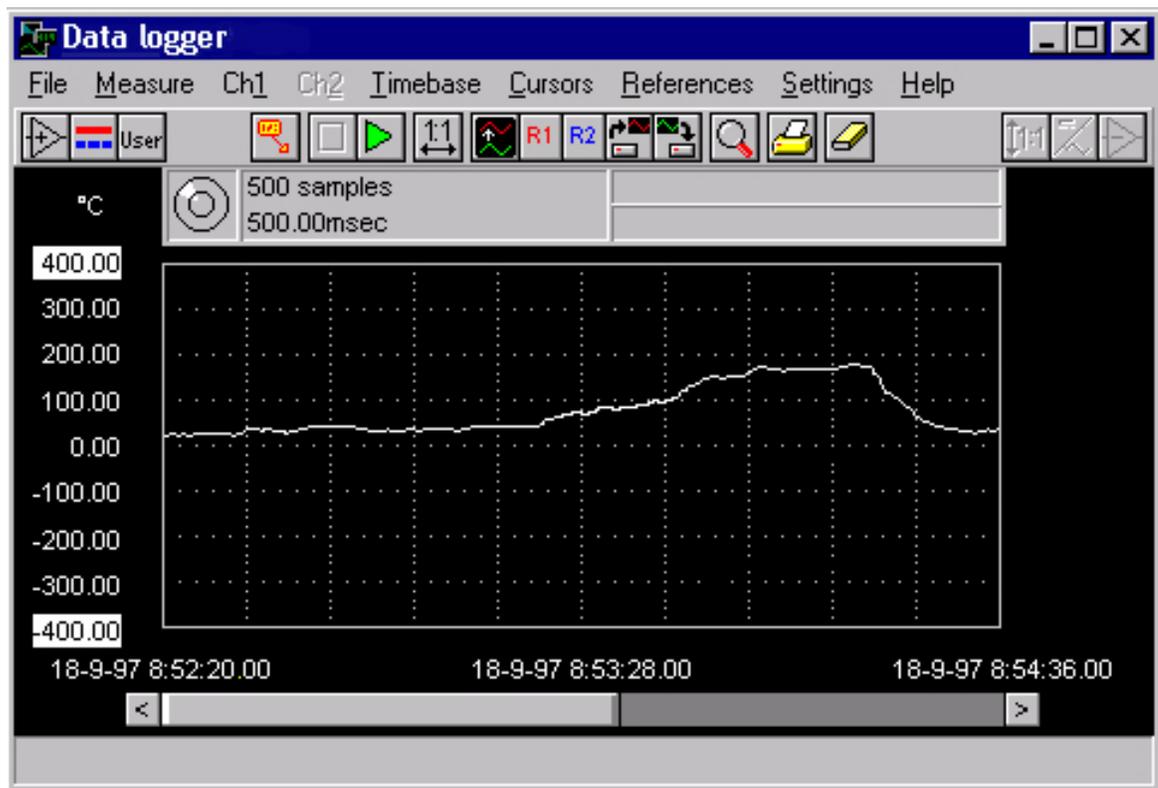
The data logger is a two channel, direct registering measuring instrument, displaying the changes of the input signal graphically on the screen or on paper.

The data logger measures at settable, fixed times and processes the measured value if necessary. The number of measurements to be taken is also settable.

*Note: The data logger displays each measured sample immediately, the oscilloscope first takes all samples and then displays the complete record.*

The data logger measures the input signals at adjustable times. The time between two measurements is adjustable from 0.01 second to 500 seconds. The number of samples is also adjustable from 1 to 32760. The maximum measuring time is  $500 \text{ sec} \times 32760 \text{ samples} = 16380000 \text{ seconds}$  (189.6 days).

*Note: Due to the different way of measuring the input signals, the data logger can not be used together with the oscilloscope, voltmeter or spectrum analyzer.*



## ***The data logger measuring system***

Before measuring with the data logger, the channels and the time base have to be set up completely.

When all settings (input sensitivity, coupling, number of samples, sample speed etc.) are made, the data logger can be started. All settings will be applied to the hardware and the hardware begins to measure. The hardware will continue measuring until the predefined number of samples is measured or when the user interrupts the measurement. Each measured sample will be stored in the board memory of the hardware.

When the measurement is running, the software will constantly monitor whether a new sample is measured. If a new sample is measured, it is obtained from the hardware acquisition memory and displayed on the display. This will continue until the measurement is ready.

**Note:** At certain sample speeds and/or relatively slow computers, it might occur that the software can't keep up with the hardware, in other words, the software misses samples the hardware has measured and stored. This is only a minor problem. The software detects that it has missed samples and will warn the user by lighting up a red LED in the display and displaying a warning message in the hint bar.

**Note:** The missed sample in the record is filled with the value of the previous sample.

**Note:** Trying to reload the missed sample(s) would take so much time, that again samples would be missed.

When the total measurement is ready, the software will reload the complete measured record from the hardware acquisition memory and display it.

**Hint:** There are a few things to make the software keep up with the hardware longer:

- make the data logger window as small as possible
- Select Scan mode instead of Roll mode
- Do not show references while measuring
- close all other applications (especially applications in DOS windows)
- Set the Windows video driver to display up to 256 colors instead of True color or High color

## Displaying channels

The two channels can be measured in several different ways:

Ch1	only channel 1 is measured and displayed
Ch2	only channel 2 is measured and displayed
Ch1 and Ch2	channel 1 and channel 2 are measured and displayed simultaneously

Selecting which way to measure the channels can be done using the Measure menu.

When only one channel is measured, the controls of the other channel are disabled or removed from the display.

## Setting up a channel

Setting up a channel in the data logger is done in the same way as in the oscilloscope. The only difference is that in the data logger no auto ranging for the input sensitivity is available.

For information on setting up a channel, see chapter 2, the oscilloscope.

## Setting up the time axis

The time base in the data logger is almost the same as in the oscilloscope. However, the data logger has no pre trigger setting.

Therefore only the items that differ from the oscilloscope time base settings are discussed here.

## Setting up the sample speed

The data logger measures a number of samples with a certain time between two samples. This results in a total measure time of (number of samples x time between two samples).

To setup the timing characteristics of the data logger measurement, open the data logger measurement speed dialog. The dialog can be opened by activating the choice Sampling speed... from the Time base menu.

In the upper part of the dialog a selection has to be made how the timing characteristics of the data logger measurement have to be set. There are three possible ways:

- Enter the number of samples and the time between to measurements. The total measuring time is determined by the dialog.
- Enter the number of samples and the total measuring time. The time between two samples is determined by the dialog.
- Enter the time between two samples and the total measuring time. The number of samples is determined by the dialog.

When a selection is made, the selected items can be entered in the input boxes in the lower part of the dialog. Under the input boxes, the determined value of the third parameter is displayed. When an invalid combination is made, the text in the input box that causes the invalid combination turns into red. When this is ignored, the value is clipped to the nearest valid value.

When the parameters are set properly the OK button can be clicked to apply the values and return to the data logger.

*Note: The timing in the hardware is generated by a crystal oscillator and two programmable integer dividers. This system can't generate all frequencies, but comes close. When the OK button is clicked, the software will determine the division values for both dividers, in such a way that the generated frequency approaches the requested frequency as close as possible. A lot of divider combinations have to be checked for that. While determining the correct division factors, an hour glass is displayed*

## Setting the record length

The data logger takes a number of samples to display the input signal. This number of samples is called the Record length. To change the record length, three ways are available: using the record length menu, using the sample speed dialog or by means of function keys.

To change the record length using the menu, enter the Time base menu and select the Record length item. Another menu will pop up, containing the available record lengths and an item for a user defined record length. Selecting this item will bring up a dialog box in which a self defined record length (within the instrument hardware limits) can be entered. The Time base menu can also be entered by pressing the right mouse button over one of the time base controls or read outs.

For changing the record length the following function keys are available:

<F11>	Select a shorter record length
<F12>	Select a longer record length

For the third way to change the record length, see setting up the sampling speed.

## Setting the horizontal axis type

The data logger can measure for a very long time. In that case it can be useful to have real time and date values along the time axis instead of the number of seconds since the starting moment.

This can be selected by opening the Settings menu and select the Horizontal axis type item. Now you can select between Date / Time and Sample times.

## Perform measurements

Before measuring with the data logger, the channels and the time base have to be set up completely.

The channels do not have auto ranging, so the sensitivity has to be set manually. Since the data logger usually measures slow signals, the input coupling should be set to DC, otherwise the input capacitor will block the signal.

## Start measuring

When all settings are made, the data logger can be started. This can be done in three ways:

- Clicking the Play button ()
- Pressing the <S> key
- Enter the Measure menu and select the Start measuring item

When the measurement is started, all controls except the Stop controls are disabled.

### Stop measuring

When the predefined number of samples is reached, the data logger will stop measuring. It is also possible to stop the data logger manually, e.g. when the event to measure is measured, or the instrument settings are wrong (coupling, sensitivity etc.). This can be done in three ways:

- Clicking the Stop button ()
- Pressing the <S> key
- Enter the Measure menu and select the Stop measuring item

When the measurement is stopped, all controls are enabled again.

### Scroll mode

While measuring, the data logger displays the measured signal on a 1:1 base, meaning that for each sample 1 display pixel is used. When the number of measured samples is larger than the available number of pixels in the display, the record view offset has to be changed in such a way that the latest samples will be visible.

There are two ways to do this:

Scan mode: When the display is full, the record view offset will change a full screen, so the new samples will start filling the display from the left hand side of the display again.

Roll mode: When the display is full, the record view offset will change with one sample, making space for one new sample. Each new sample causes a change in the record view offset.

Roll mode looks nicer than Scan mode, but is also slower, so the data logger will sooner miss samples in Roll mode than in Scan mode.

To set the scroll mode, enter the Measure menu, select the item Scroll mode and choose between the two possibilities.

### ***Comparing signals using the reference channels***

To make the data logger keep up with the measuring hardware longer, references can be switched off while measuring. To do this, open the References menu and select the Show references while measuring item.

When the item Show references while measuring is checked, switched on references are shown while the data logger measures. If the item is not checked, the references are switched off when the measurement is started and switched on again when the data logger is ready.

### ***Performing cursor measurements***

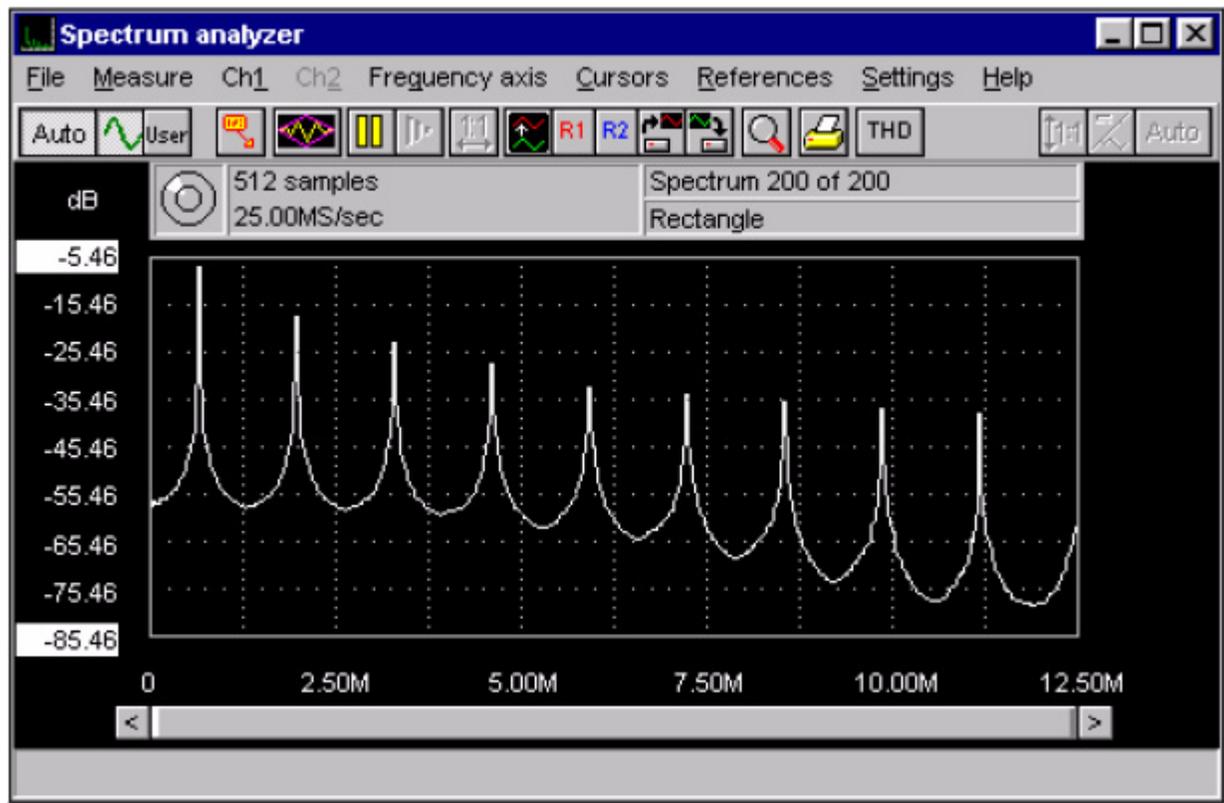
Like in the oscilloscope, the data logger has a set of cursors to perform measurements on the displayed signals.

The cursors in the data logger work exactly like the oscilloscope, so refer to chapter 2, the oscilloscope, for an explanation on how to work with the cursors.

# 9. The Spectrum Analyzer

The common way to examine electrical signals is in the time domain, using an oscilloscope. The time domain is used to determine amplitude, time and phase information, which is necessary to describe the behavior of an electrical system.

Not all electrical systems can be characterized in the time domain. Circuits like filters, amplifiers, oscillators, mixers, modulators and detectors can be characterized best by their frequency behavior. That frequency behavior is best obtained by observing the electrical signals in the frequency domain. To display the frequency domain, an instrument is needed that can distinguish different frequencies from each other and measure the signal size at the different frequencies. An instrument that can display the frequency domain is the spectrum analyzer. It graphically displays voltage as a function of frequency.



In the time domain all frequency components of a signal are seen summed together. In the frequency domain, complex signals (signals composed of more than one frequency) are separated into their frequency components, where the voltage of each component is displayed.

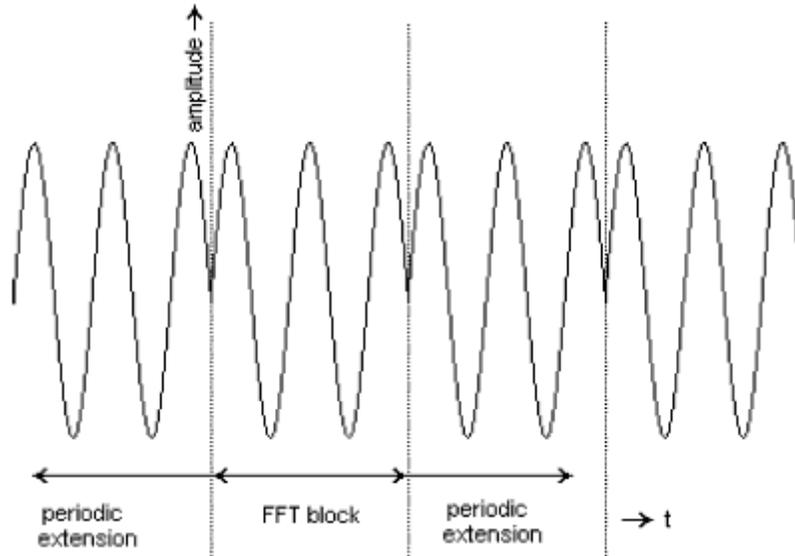
The frequency domain contains information which is not available in the time domain. Therefore the spectrum analyzer has certain advantages compared with an oscilloscope.

- With a spectrum analyzer small harmonic distortions on a signal can be displayed better than on an oscilloscope. A sine wave may look good in the time domain, however in the frequency domain the harmonic distortion is visible.
- A noise signal may look fully random in the time domain, in the frequency domain it can appear that one frequency is dominantly present.

- In the frequency domain it is very simple to determine carrier frequency, modulation frequency, modulation level and modulation distortion from a modulated signal (AM or FM).

**Description of FFT**

The spectrum analyzer is using the Fast Fourier Transform (FFT) principle. For this transform, a number of samples are taken with a previously set sample speed. The number of samples has to be a power of 2, e.g. 512, 1024 or 2048. With these samples, here called FFT-block, the spectrum is calculated. Therefore the FFT places an infinite number of FFT-blocks behind each other, in the positive and negative direction. Each FFT-block forms 1 period of the created periodic signal.

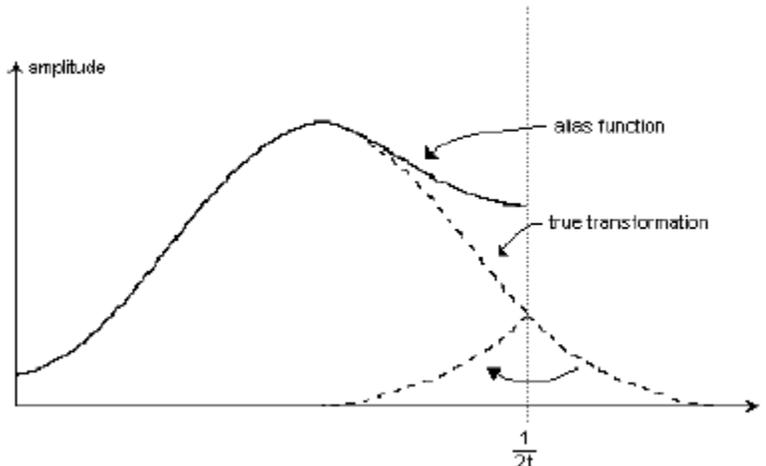


From the signal obtained like this, the spectrum is calculated. The FFT calculation results in a number of spectral components which is half the number of samples in the FFT block (e.g. 256, 512, 1024). These spectral components together form the frequency spectrum of the input signal.

The spectrum analyzer displays an amplitude spectrum, which means that a 2 volt peak-peak sine wave is displayed with a 1 volt spectral component.

**Aliasing**

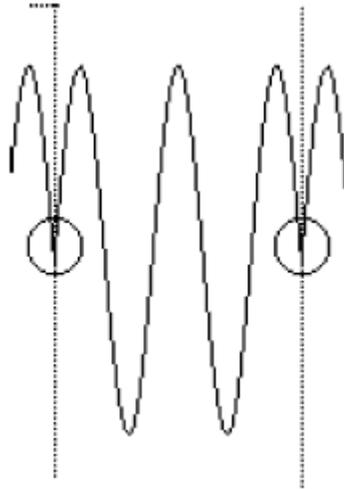
Like the oscilloscope, the spectrum analyzer needs a high enough sampling frequency to avoid aliasing. The result of aliasing when using FFT is that the frequency curve is folded back on itself.



The original signal is sampled at a sample time  $t$ . The critical frequency (Nyquist frequency) is  $(1/2t)$  Hz.

## Windowing

FFT treats the FFT-block like it is one period of a periodic signal. If the sampled signal is not periodic, harmonic distortion can arise, since the periodic waveform, 'created by the FFT', can have sharp discontinuities. See also the following illustration. These discontinuities are also called calculation errors.



This means that because of the calculation errors extra frequency components are generated around the true frequency. Because of this 'smearing' the amplitude of the true frequency decreases, since the area under the curve remains the same.

The smearing, caused by the calculation errors, can be decreased by placing a window on the FFT-block samples, in such a way that the ends of the FFT-block are smoothly tapered to zero and discontinuities do not occur when the FFT treats the windowed block as one period of a periodic signal. Each sample of the FFT-block is therefore multiplied by a factor, whose size is dependent on the position of the sample in the FFT-block.

The software supports six different windows:

- rectangular (all data is not changed)
- Hanning (sinusoidal)
- Hamming (sinusoidal)
- Bartlett (sinusoidal)
- Blackman (triangular)
- Parzen (triangular)

## Displaying channels

The spectrum analyzer has several modes to display the two channels. To select the measure mode, enter the Measure menu and select:

Ch1	measure and display only channel 1
Ch2	measure and display only channel 2
Ch1 and Ch2	measure channel 1 and channel 2 simultaneously and display the values of both channels

When only one channel is measured, the controls of the other channel are disabled or removed from the display.

## Setting up a channel

The two channels of the spectrum analyzer are completely configurable. Both channels are identical, each has its own menu item in the main menu. The possibilities of the vertical axes in the spectrum analyzer are the same as in the oscilloscope, so only the differences are discussed here. Since the possibilities for both channels are equal, channel 1 is explained here.

### Setting the input sensitivity

The input sensitivity of a channel determines how large a certain signal will be displayed, the lower the full scale value, the larger the signal will be displayed. It is possible to enable an autoranging function.

*Note: The vertical axis in the spectrum analyzer has a range that is twice the input sensitivity. This is done because the frequency components of which a signal is built, can have a larger amplitude than the input sensitivity. So if an input sensitivity of 8 Volt full scale is selected, the axis will have values from 0 to 16 Volt full scale.*

The sensitivity can be set through the Sensitivity menu. The Sensitivity menu is called by enabling the Ch1 menu and then selecting the Sensitivity menu. In this menu the input sensitivities are available. In the spectrum analyzer also an Autoranging setting is available. The Sensitivity menu can also be entered by clicking the right mouse button over one of the Ch1 or Ch2 controls and readouts.

To switch on Autoranging, a special button is available. When autoranging is switched on, the button caption is Auto, otherwise it holds the current set input sensitivity. Pressing the right mouse button on this button will bring up a menu with all sensitivities.

*Note: When Autoranging is switched on, several measurements might be needed to set the input sensitivity properly, depending on the input signal.*

Another way to set the sensitivity is by using the keyboard. A few keys have been reserved for setting the input sensitivity of the channels. These keys are:

<F5>	Increase Ch1 sensitivity
<F6>	Decrease Ch1 sensitivity
<F7>	Increase Ch2 sensitivity
<F8>	Decrease Ch2 sensitivity

### Logarithmic or linear vertical axis

The vertical axis of the spectrum analyzer can be set to linear or to logarithmic.

When set to linear, the normal axis is used, running from 0 to 2 x Sensitivity. When set to logarithmic, the position on the linear axis that is marked Sensitivity will get the value

$$\text{Value} = 20 \times 10^{\log \text{Sensitivity}}$$

The rest of the labels will get a value that is related to this value and differ from it with 10 dB per division (when Software gain is set to 1). The measured data will be transformed to logarithmic data as well.

Selecting the axis type can be done by entering the Ch1 menu, choosing the Axis type menu and selecting the axis type.

The axis type can also be set by the keyboard using the following keys:

<L>	toggle the Ch1 axis between Linear and Logarithmic
<Shift+L>	toggle the Ch2 axis between Linear and Logarithmic

**Note:** Due to 'smearing' errors, caused by discontinuities (See Windowing) and (within specifications) distortion in the Analog to Digital Converter non-existing frequency components will show up when a logarithmic axis is selected. These components are all outside the dynamic range of the measurement system.

To avoid confusion with these non-existing frequency components, the logarithmic graph can be clipped on the theoretical dynamic range of the measurement system. This is done by selecting the Vertical log axis range item from the Settings menu and choosing either Not clipped or Clipped conform hardware specs.

### Setting up the frequency range

Along the horizontal axis of the screen, the frequency of the measured spectrum is projected.

Magnifying the frequency axis and horizontally positioning the signal is done in the same way as in the oscilloscope and is therefor not discussed here.

### Setting up the frequency axis type

The frequency axis can be set up in four different modes:

- Linear            the frequency axis runs from 0 to the maximum frequency and has a linear division.
- Logarithmic    the frequency axis runs from the minimum frequency to the maximum frequency and has a logarithmic division.
- Octaves         the frequency axis runs from 22.1 Hz to 22.6 kHz and is divided in 10 octaves. The center frequencies, the lower limits and the higher limits of the octaves are:

$f_{\text{center}}$	$f_{\text{lo}}$	$f_{\text{hi}}$
31.25 Hz	22.1 Hz	44.2 Hz
62.5 Hz	44.2 Hz	88.4 Hz
125 Hz	88.4 Hz	176.8 Hz
250 Hz	176.8 Hz	353.5 Hz
500 Hz	353.5 Hz	707.1 Hz
1 kHz	707.1 Hz	1.414 kHz
2 kHz	1.414 kHz	2.828 kHz
4 kHz	2.828 kHz	5.656 kHz
8 kHz	5.656 kHz	11.314 kHz
16 kHz	11.314 kHz	22.627 kHz

Each octave is displayed with the same width on the display.

This mode requires that the sampling frequency is about 44 kHz and the record length is at least 1024 spectral components. When this display mode is switched on, the sampling frequency can no longer be set and the record length can only be set to a few lengths.

- Third octaves    the frequency axis runs from 17.54 Hz to 22.6 kHz and is divided in 31 thirds of octaves. The center frequencies, the lower limits and the higher limits of the octaves are:

$f_{\text{center}}$	$f_{\text{lo}}$	$f_{\text{hi}}$
19.68 Hz	17.54 Hz	22.09 Hz

---

24.80 Hz	22.09 Hz	17.84 Hz
31.25 Hz	27.84 Hz	35.1 Hz
39.4 Hz	35.1 Hz	44.2 Hz
49.6 Hz	44.2 Hz	55.7 Hz
62.5 Hz	55.7 Hz	70.2 Hz
78.7 Hz	70.2 Hz	88.4 Hz
99.2 Hz	88.4 Hz	111 Hz
125 Hz	111 Hz	140 Hz
157 Hz	140 Hz	177 Hz
198 Hz	177 Hz	223 Hz
250 Hz	223 Hz	280 Hz
315 Hz	280 Hz	354 Hz
397 Hz	354 Hz	445 Hz
500 Hz	445 Hz	561 Hz
630 Hz	561 Hz	707 Hz
794 Hz	707 Hz	891 Hz
1.00 kHz	891 Hz	1.12 kHz
1.26 kHz	1.12 kHz	1.41 kHz
1.59 kHz	1.41 kHz	1.78 kHz
2.00 kHz	1.78 kHz	2.24 kHz
2.52 kHz	2.24 kHz	2.82 kHz
3.17 kHz	2.82 kHz	3.56 kHz
4.00 kHz	3.56 kHz	4.49 kHz
5.04 kHz	4.49 kHz	5.66 kHz
6.35 kHz	5.66 kHz	7.13 kHz
8.00 kHz	7.13 kHz	8.98 kHz
10.1 kHz	8.98 kHz	11.3 kHz
12.7 kHz	11.3 kHz	14.3 kHz
16.0 kHz	14.3 kHz	17.9 kHz
20.2 kHz	17.9 kHz	22.6 kHz

Each third octave is displayed with the same width on the display.

This mode requires that the sampling frequency is about 44 kHz and the record length is at least 4096 spectral components. When this display mode is switched on, the sampling frequency and the record length can no longer be set.

*Note: When the spectrum analyzer is set to Hold and the frequency axis is set to octaves or third octaves, the signal will not match with the axis and the grid until a new measurement is taken.*

## Setting the frequency range

The frequency range can be set in two ways: using the menu or by means of special function keys.

The frequency range is determined fully on the sample frequency: the frequency range is half of the sample frequency.

To change the Frequency range using the menu, enter the Frequency axis menu and select the Frequency range item. A new menu will appear with all possible frequency range settings. The Frequency axis menu can also be entered by pressing the right mouse button over one of the frequency axis controls or read outs.

The last item in the menu is a user defined setting. Activating this item will bring up a dialog box in which a self defined frequency range (within the instrument hardware limitations) can be entered. Due to the way the sample frequency is generated in the hardware, not all frequencies and thus not all frequency ranges can be generated. If an invalid frequency range is entered, the closest valid frequency range will be generated.

The following keys can be used to change the frequency range:

<F3>	Set the frequency range one step smaller
<F4>	Set the frequency range one step larger

## Setting the spectrum record length

The spectrum analyzer takes a number of samples to determine the frequency spectrum of the input signal. This number has to be a power of 2. After the FFT calculation, a spectrum with half of that number spectral components is obtained. This number of spectral components is called the Record length.

To change the record length, two ways are available: by means of a menu or by means of function keys.

To change the record length using the menu, enter the Frequency axis menu and select the Record length item. Another menu will pop up, containing the available record lengths. The Frequency axis menu can also be entered by pressing the right mouse button over one of the frequency axis controls or read outs.

For changing the record length the following function keys are available:

<F11>	Select a shorter record length
<F12>	Select a longer record length

*Note: Calculating a spectrum is time consuming. The time need for calculating a spectrum is strongly dependent on the calculating power of the computer and of the selected record length.*

## Setting up the FFT

### FFT window function

To reduce 'smearing' errors (See Windowing), a windowing function can be selected.

To select a windowing function, press the Window button () or select the item FFT window function... from the Settings menu. A dialog will appear in which the requested window can be selected. A graphic display will show effect of the windowing function on the sampled signal.

The available window types are:

- rectangular (all data is not changed)
- Hanning (sinusoidal)
- Hamming (sinusoidal)
- Bartlett (sinusoidal)
- Blackman (triangular)
- Parzen (triangular)

## **Performing measurements**

When all settings are made, measurements can be made.

### **Start and stop measuring**

To avoid a new screen of measurement data replacing an old screen, measuring and displaying of data can be stopped.

This can be done in several ways: by pressing the Pause button () , by pressing the <S> key or by activating the item Stop continuous measuring from the Measure menu.

When measuring is stopped, the latest measured data will remain on the screen. The Pause button changes to a Play button () and the One shot button () becomes available.

The measured signal can now be examined.

### **Single measurements**

To perform a single measurement, press the One shot button, press the <O> key or activate the Perform a single measurement item from the Measure menu. When in the Spectrum analyzer averaging is selected, there will be performed as many measurements as there are set for the averaging and then measuring will stop.

Continuous measuring can be started again by pressing the Play button, the <S> key or the item Start continuous measuring from the Measure menu. The One shot button will be disabled again.

### **Averaging**

When a noisy signal is measured, the influence of the noise can be eliminated by averaging several spectra. The noise component(s) will disappear and the actual signal remains.

Averaging can be switched on by selecting the item Perform averaging of from the Measure menu. In the next menu a number of measurements can be selected that have to be averaged. The item 1 measurement will switch off averaging.

When averaging is set to e.g. 20 measurements and the spectrum analyzer is set to Pause, pressing the One shot button will result in taking 20 measurements which are averaged

### **Measuring maximum values**

The spectrum analyzer can be set to measure and display only the maximum values of the spectral components. After a number of measurements the highest value of each spectral component is displayed then.

To enable measuring maximum values, enter the Measure menu and select the choice Measure max values. A check mark in front of the menu item indicates whether measuring maximum values is switched on or off.

**Note:** The display of maximum values is reset when the instrument setting changes

**Hint:** This mode can be useful when the frequency response characteristic of a device needs to be determined. Apply a frequency swept signal and switch on Measure max values and the characteristic will be created.

## Measure the Total Harmonic Distortion of a signal

With the spectrum analyzer it is possible to measure the Total Harmonic Distortion of a signal.

The THD is defined as:

$$20 \times 10 \log \frac{\sqrt{V_2^2 + V_3^2 + V_4^2 + V_5^2 + V_6^2 + V_7^2 + V_8^2 + V_9^2 + \dots + V_n^2}}{V_1}$$

where  $V_1$  is the RMS amplitude of the base frequency of the input signal,  $V_2 \dots V_n$  are the RMS amplitudes of each higher harmonic.

To measure the THD, enter the Measure menu and select the Total Harmonic Distortion item or by pressing the THD button (  ). The software will determine the base frequency of the signal by assuming that the spectral

component with the largest amplitude is the base frequency. Then the amplitudes of all higher harmonics (up to 100) in the measured record are determined and the THD is determined.

In the THD output window the base frequency of the signal is given, the harmonic distortion in dB is given and a table with the amplitudes of all used harmonics is given. This is done for all active channels.

The number of harmonics that is used for the calculation depends on the base frequency. If the base frequency is low, more harmonics are found than when the base frequency is high. The calculation routine will use up to 100 harmonics or up to a user defined number. This user defined number can be set by entering the Measure menu and selecting the item Max. number of harmonics to use. A new menu will pop up with the possible numbers.

## Performing cursor measurements

Like in the oscilloscope, the spectrum analyzer has a set of cursors to perform measurements on the displayed signals.

The cursors in the spectrum analyzer work almost like in the oscilloscope, so only the differences are discussed here. Refer to chapter 2, the oscilloscope, for more explanation on how to work with the cursors.

## Setting up the cursor measurements

With the cursors it is possible to measure the input signal(s) in several ways. To set up the cursor measurements, press the settings button at the cursor readout window (the right most button at the left top of the window caption) A menu appears, containing the following options. The menu can also be called by pressing the right mouse button over the table.

### Time

Frequency left: the frequency at the position of the left cursor is determined.

Frequency right: the frequency at the position of the right cursor is determined.

Frequency difference: the frequency difference between the left and right cursor is determined.

### Voltage

Voltage left: the voltage of the left cursor will be displayed.

Voltage right: the voltage of the right cursor will be displayed.

Voltage difference: the difference between the voltage of the left and the voltage of the right cursor is calculated. Selecting the min and max of the signal, the top-top value is calculated.

**Distortion**

The Total Harmonic Distortion of the input signal is determined. Up to 100 harmonics are used for the calculation

**Automatic tracking cursors**

If this option is enabled, the voltage levels of the cursors are automatically set to the voltage levels of the sample numbers which correspond to the left and right cursor.

**Calculate values**

If this option is disabled, no new calculations will be done until the option is enabled.

**Show measurement name**

**Show measured value**

**Show measurement unit**

To reduce the space used by the window on the screen, the measurement name, value and/or unit can be hidden. This results in the disappearance of the corresponding column which results in a smaller window.

**Display font**

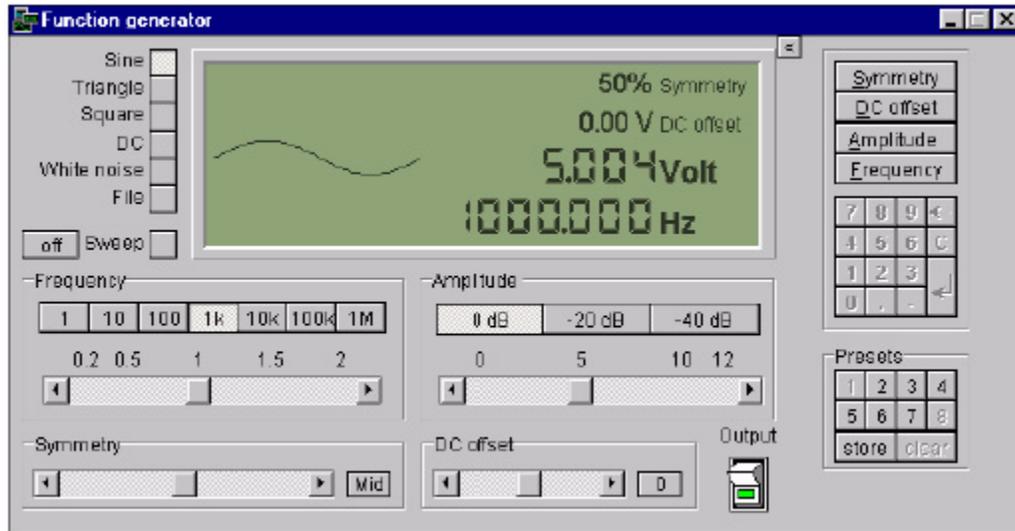
By selecting this option, the font which is used to display the strings and values can be changed. The font settings which can be changed are: font name, font style, font size and the font color.

**Background color**

By selecting this option, the background color can be changed. Some font colors like e.g. light green are not showing up very good on a white background. Change the background color to, e.g. black for a better result.

# 10. The Function Generator

The DI-770 is also equipped with a function generator.



With the function generator, signals of various shapes, with adjustable frequency, symmetry, amplitude and offset.

Next to that, the function generator contains a sweep generator, which can in combination with the spectrum analyzer, be used for creating frequency response characteristics of components or circuits.

And the function generator provides for a number of user definable preset positions.

## ***Principle of the function generator***

The function generator uses the DDS technique. DDS stands for Direct Digital Synthesis, a technique that uses digital data processing blocks to generate a frequency-tunable with a fixed-frequency precision clock. DDS allows to generate low distortion signals with very accurate frequency adjustment, in a fully digital way.

## ***Structure of the function generator***

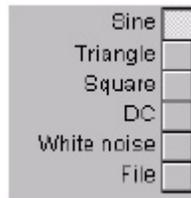
The function generator is built out of a number of functional blocks.

At the top a display is located, which shows the signal shape and the other settings of the function generator. Left of the display there are the signal shape selector buttons. Below the display are the controls for the frequency, symmetry, amplitude and offset of the signal. At the right of the display are a key pad for numerical input and a key pad for the presets.

This part of the screen can be hidden using the small button at the right upper corner of the display of the function generator. The function generator will become smaller and taking up less space on the computer display. Pressing the button again will expand the window again. The same can be obtained by pressing keys <Shift-<> and <Shift->>.

## Controlling the function generator

### Signal shape

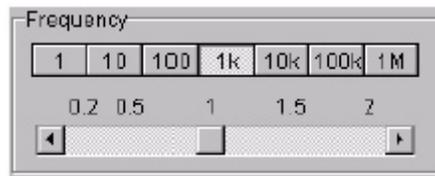


With the signal shape selector buttons the signal type can be set. Possible signals are:

Sine	a sine wave is generated
Triangle	a triangle wave is generated
Square	a square wave is generated
DC	a constant DC level is generated
White noise	a white noise is generated
File	a signal from a file is generated

The signal type File is discussed later.

### Frequency



In the Frequency box all controls for adjusting the signal frequency are located. Seven decade buttons set a frequency range in which the frequency can be set using the slider. Each range runs from 0.2 to 2 times the value on the button.

When the signal shape is set to DC, the frequency adjustment is disabled.

### Symmetry

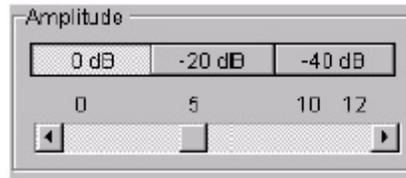


The symmetry of a signal indicates the relation between the length of the positive part of the period to the total length of the period. A symmetry of 50% means that the positive part is exactly half of the total period.

Using the slider in the Symmetry box, the symmetry can be adjusted between 1% and 99%. The Mid button sets the symmetry to 50%, the centre position.

When the signal shape is set to DC or White noise, the symmetry adjustment is disabled.

## Amplitude



The amplitude of the output signal can be adjusted in three ranges: not attenuated, -20 dB attenuated or -40 dB attenuated. With the slider a voltage between 0 Volt and 12 Volt (or 0 Volt and 10 Volt, depending on the instrument) can be set.

## DC offset



It is possible to add a DC offset to the output signal. In the DC offset box there is a slider for the amplitude of the offset. The button will set the DC offset back to zero.

When the signal shape is set to DC, the slider is used to toggle the polarity of the DC signal

## Numerical input



All described functions can also be controlled using the keyboard. When the window is expanded, a small key pad is located which enables entering all settings.

This key pad can be controlled by mouse but also with the keyboard of the computer. In that case, the window doesn't need to be expanded.

The top four keys indicate which setting will be adjusted. The corresponding reading in the display will start blinking. Also the numerical keys will be enabled and the required value can be entered. The input can be ended by pressing the <Enter> key.

The following keys are available:



Adjust the symmetry of the signal



Adjust the DC offset of the signal

-  Adjust the amplitude of the signal
-  Adjust the frequency of the signal
-  Add this figure to the entered number
-  Add a decimal point to the entered number
-  Reverse the polarity of the entered number
-  Terminate the input
-  Remove the last entered character
-  End the input and apply it
-  Terminate the input

### **Presets**

The function generator has 8 preset positions to store signal settings.



With a single press on a button, a signal with a specified shape, amplitude, offset, frequency and symmetry can be generated.

The presets are stored together with the application settings and are restored each time the application is started.

### **Storing**

When a signal is set that is not yet stored in a preset position, and there is a free preset position available, the store button is enabled. By pressing the button, the signal settings are stored in the free preset position with the lowest number.

## Selecting

When presets are stored, the corresponding preset buttons are enabled. By hovering the mouse pointer over one of the buttons, the settings of that preset position are displayed.

By pressing a preset button, the function generator is set according to the stored settings. The button of the preset will be lowered, indicating that preset is active.

When during use of the function generator a setting is changed, and the new setting corresponds with a stored preset, the corresponding button of that preset will be lowered. When a preset is selected and the setting of the function generator is changed to a setting that does not correspond with the preset, the corresponding button will be raised.

## Clearing

To clear a stored preset, it first needs to be selected by pressing the corresponding button.

The clear button will become active. Pressing this button will clear the preset position and make the location available for a new preset. The corresponding button will be disabled.

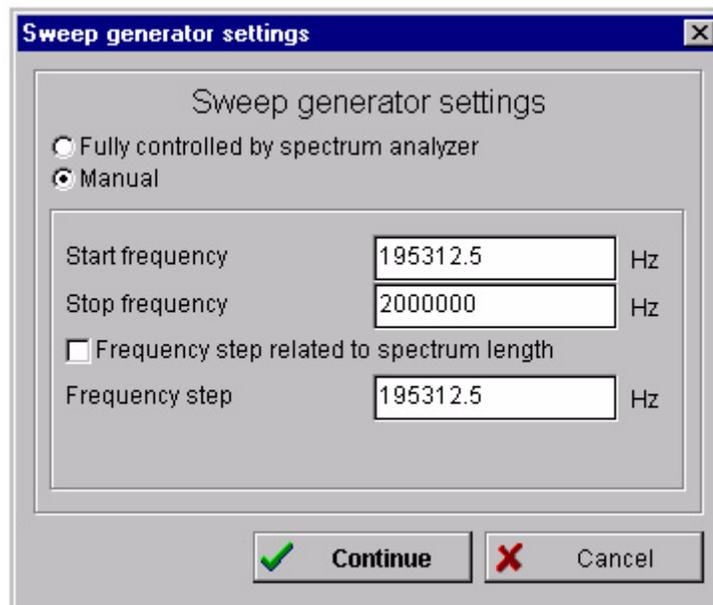
## Performing a sweep

The function generator offers, in combination with the spectrum analyzer, the possibility to generate a signal with a frequency sweep.

For this purpose the spectrum analyzer needs to be active as well. After each measurement the spectrum analyzer performs, the frequency of the generator is increased with a certain step size, until the highest value is reached. In that case the frequency will be set to the lowest value.



Below the signal shape buttons is a sweep button located, for setting the sweep parameters. Next to the button there is an on/off button, for switching the sweep function on and off. When the sweep button is pressed, the sweep generator settings dialog is displayed.



In this dialog all settings of the sweep can be made.

The easiest setting is Fully controlled by spectrum analyzer. In this setting, the setting of the frequency axis of the spectrum analyzer determines how the start and stop frequency and the frequency step size are set. The lowest visible

frequency in the spectrum analyzer sets the start frequency of the sweep. The highest visible frequency in the spectrum analyzer sets the stop frequency of the sweep. This is limited to the maximum range of the function generator. The frequency step size is determined by the frequency range of the spectrum analyzer, divided by the number of bins in the spectrum. When a sweep is active and a setting in the spectrum analyzer is changed, the sweep parameters will change accordingly.

At each step, the frequency of the generator will be set to a value that falls exactly in one single bin, therefore no calculation errors occur and no window is required.

The setting Manual requires that all settings are set by the user. The start and stop frequency can be set freely. The frequency axis of the spectrum analyzer will be set accordingly, so only the range that is swept will be visible.

The frequency step size can be determined from the spectrum length. With each step a frequency will be set that generates no calculation errors. It's also possible to set the step size manually, but that will probably do generate calculation errors in the FFT.

**Hint:** For easy creating frequency response characteristics, set the measure mode of the spectrum analyzer to Measure maximum values and switch off autoranging for the channels

After setting the sweep parameters, the sweep function can be switched on using the on/off button. During the sweep, the frequency readout in the display will follow the output frequency.

**Note:** When the sweep function is used to determine frequency response characteristics, be certain to set the signal shape to Sine.

### **Generating a previous measured signal from a file**

It is possible to generate previously measured signals with the function generator, provided of course that the amplitude and frequency of the signal are within the specs of the generator.

### **Measuring and storing a signal**

The function generator has two possibilities to generate previous measured signals. It can either use the DDS mode or it can use a linear mode.

In the DDS mode, a waveform of 1024 points is used to generate the signal. Due to the DDS technique, very accurate output signal frequencies can be obtained.

In linear mode, a waveform of 64K points is used. The sample clock of the generator can be set to a number of settings, in order to obtain different output frequencies. The number of settings is limited. When a stored waveform is used that is smaller than 64K points, the remaining points are filled with a 0 volt value.

A signal that is measured, stored and later generated, has to meet certain points, in order to be usable.

- The signal has to be measured in channel 1 of the scope.
- For DDS mode, the record length of the scope has to be set to 1024 points, for linear mode to any required length, but 64K is preferred.
- If possible, the sampling frequency of the scope has to be set in such a way that exactly 1 period of the signal is captured in the 1024 points or 64K points.

After the signal is measured, it can be saved to disk using Save waveform... from the File-menu.

### **Setting the function generator**

At the left hand side of the display in the generator, a number of signal shape buttons are located, one of them is named File. With this button a signal from a file can be selected and loaded into the generator.

Pressing this button will pop up a file selection dialog in which the proper waveform file can be selected.

After reading the signal, the amplitude and frequency of the signal can be adjusted using the proper controls.





DATAQ Instruments, Inc.  
241 Springside Drive  
Akron, Ohio 44333  
Telephone: 330-668-1444  
Fax: 330-666-5434  
E-mail: [support@dataq.com](mailto:support@dataq.com)

### Direct Product Links

(click on text to jump to page)

[Data Acquisition](#) | [Data Logger](#) | [Chart Recorder](#) | [Thermocouple](#) | [Oscilloscope](#)