



DSO USERMANUAL PCSU1000

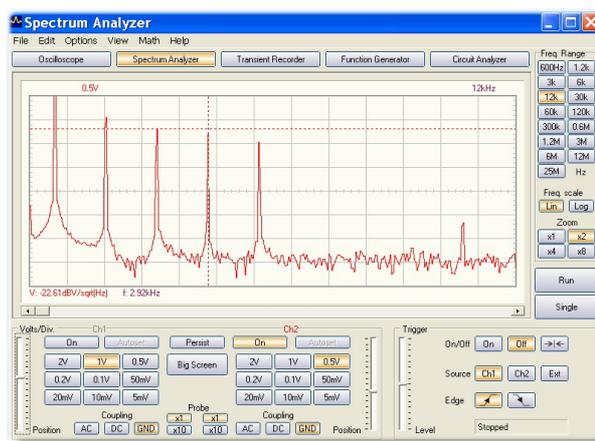
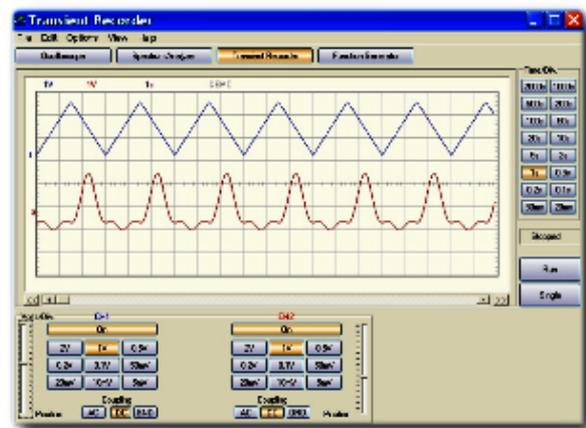
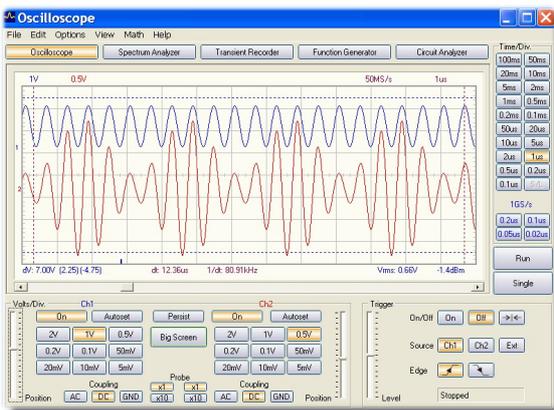


Table of Contents

Foreword	0
Part I Contents	2
1 Safety Instructions	2
2 1GS/s oversampling mode	3
3 Controls	3
4 Connection	5
5 Troubleshooting	6
6 Waveform Parameters Display	6
7 Add comment text in the signal screen	9
Part II Menu Options	9
1 File Menu	9
2 Edit Menu	10
3 Options Menu	10
4 View Menu	12
5 Spectral Density	14
6 Math Menu	16
7 Help Menu	16
Part III Data Transfer	16
1 Data acquisition to other applications	16
2 Data acquisition to Microsoft Excel	20
Index	0

1 Contents

Operation Instructions for Velleman PC Oscilloscope PCSU1000

- **DIGITAL STORAGE PC OSCILLOSCOPE**
- **FFT SPECTRUM ANALYZER**

[Safety Instructions](#)

[1GS/s sampling mode](#)

[Controls](#)

[Connection](#)

[Waveform Parameters Display](#)

[Troubleshooting](#)

[Adding comment text in the signal screen](#)

[Menu Options](#)

[Spectral Density marker](#)

[Data acquisition to other applications](#)

[Data acquisition to Microsoft Excel](#)

1.1 Safety Instructions

SAFETY and WARNINGS



Important safety information, see user manual.

	WARNINGS Instrument grounds are interconnected and connected to PC ground. Max. Voltage between probe tip and ground : 30V AC+DC at all times! Before performing any measurement, make sure that the circuit is galvanically separated from the AC power (mains)! If necessary, use a safety isolation transformer.
---	--

Before making measurements and for safety reasons, it is important to know some information about the measured unit.

Safe devices are:

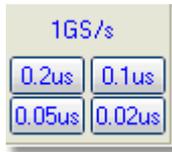
- Battery operated equipment
- Equipment supplied via a transformer or adapter.

Do not measure:

- Equipment directly connected to mains.
- Equipment that contains components that are directly connected to mains (dimers...).
- If necessary to measure above mentioned equipment, use an isolation transformer.

Please remember that the grounds of both channels are interconnected and connected to the PC ground !

1.2 1GS/s oversampling mode



- The 1GS/s sampling rate is in use on blue 0.2us/div, 0.1us/div, 0.05us/div and 0.02us/div ranges.
- Triggering must be ON to get stable waveform images.
- This mode is useable only for repetitive signals.
- This operation mode is called "Random Interleaved Sampling" (RIS) method, sometimes also called "Equivalent Time Sampling" (ET) mode or "Random Repetitive Sampling". In this sampling mode the oscilloscope uses successive triggering occurrences to gather the data to construct a picture of a repetitive signal.

1.3 Controls

Spectrum analyzer mode

FREQ. RANGE

Sets the frequency range of the display. It is necessary to slide the screen using X-POSITION in order to see the full range.

LOG/LIN

To display the frequency on a linear or logarithmic scale.

ZOOM x1, x2, x4, x8

In order to expand the screen X1, X2, X4 or X8

Oscilloscope mode

VOLTS/DIV

Selected value indicates the peak-to-peak voltage required to produce a peak-to-peak deflection of one major division on the screen

Big Screen

Displays large waveform screen with separate button bar. Use **Normal Screen** button to return normal mode.

Note: Big screen is available only in Oscilloscope and Spectrum Analyzer modes.

Coupling

AC: the input signal is capacitive coupled to the input amplifier/attenuator. Only the AC components are measured.

GND: the input signal is broken and the input amplifier/attenuator is connected to earth. Use this position for selecting a reference point on the display.

DC: the input signal is directly connected to the input amplifier/attenuator. Both AC and DC voltage are measured.

Probe x1/x10

Use these buttons to adapt the readouts according to the x1/x10-probe setting.

CH1 On, CH2 On

Buttons turn the display of the trace ON or OFF. To get the cursor measurements of CH2 voltage values switch CH1 off.

Autoset

Automatic setup for the Volts/div, Time/div, and Trigger level to produce a stable waveform of usable size. The trigger will be set on if the wave amplitude on the screen is more than 0.5 divisions.

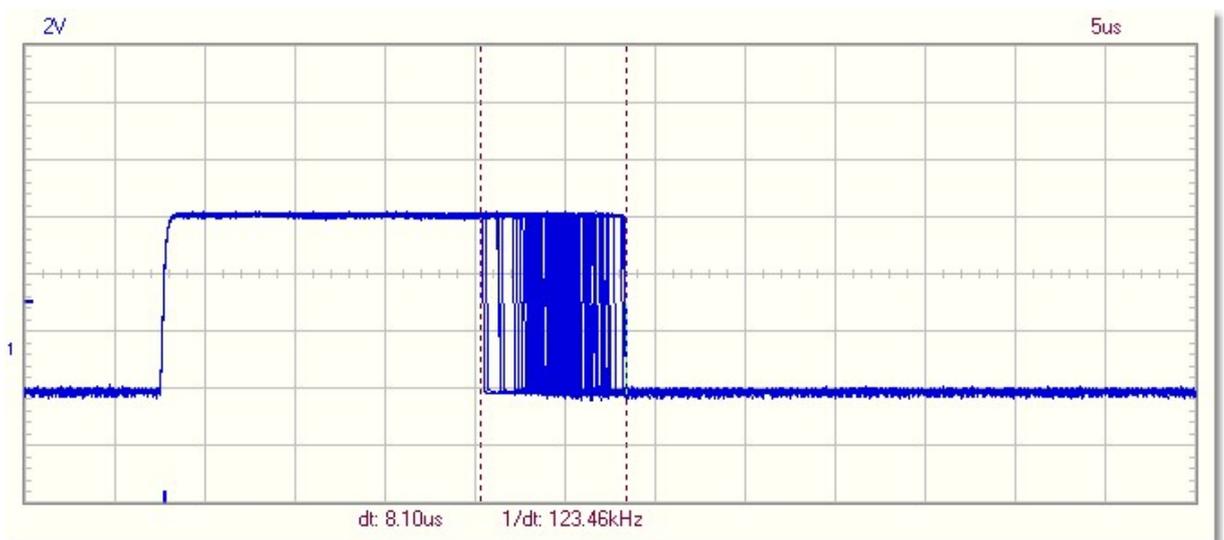
The signal should be repetitive for proper autoset function: Amplitude 5mV to 100V; frequency above 50Hz; duty cycle greater than 10%.

Persist

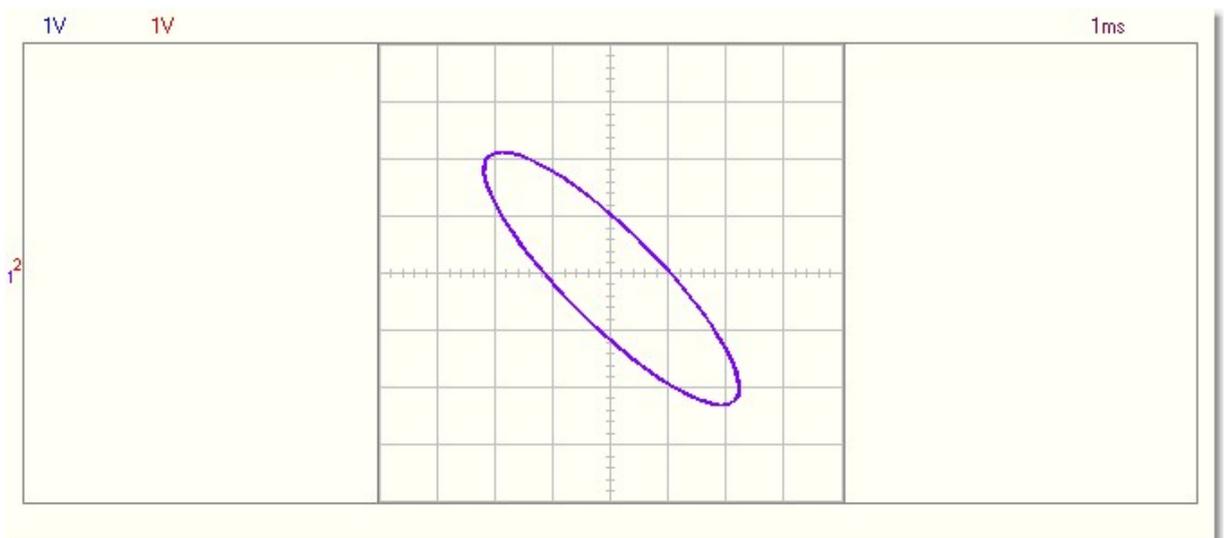
When this button is down the scope captures many acquisitions of a signal to the screen. Record points accumulate until you release the button.

When this button is down the scope captures many acquisitions of a signal to the screen. Record points accumulate until you release the button.

Using Persistence option you can easily analyze worst-case signal variations, such as jitter or noise. The Persist option can also be used to locate errors in digital signals. Using this option you can capture erroneous events even if they only occur once. Persist option makes it easy to compare known and unknown circuits. Click "Single" button to capture multiple waveforms on the same screen.



Persist option lets you see the range over which a signal varies



Use the Persist option to get solid XY patterns in the XY Plot mode

TIME/DIV

Selects the time setting for the beam to sweep one major division on the screen

By selecting different TIME/DIV settings it is possible to **zoom** the frozen waveform on the screen.

TRIGGER On/Off

Selects free run mode or triggered mode.

TRIGGER Level

Selects the signal level at which the sweep is triggered.

Triggering reference mark is displayed with the horizontal line on the left side of the screen.

TRIGGER Channel

Selects the trigger source signal (Ch1, Ch2 or Ext)

TRIGGER Edge

Selects the triggering slope:

Arrow up: Trigger occurs when triggering signal crosses trigger level in a positive going direction.

Arrow down: Trigger occurs when triggering signal crosses trigger level in a negative going direction.



Resets the triggering x-position reference point. Triggering reference mark is displayed with the vertical line on the bottom of the screen.

RUN

Selects recurrent display update mode (RUN). Pressing the button again freezes the display.

SINGLE

When button is depressed and the trigger level is reached, refreshment of the display takes place only once.

X-POSITION SCROLLBAR (Below the waveform display)

Positions the trace horizontally on the screen. Triggering reference point is displayed with the vertical line on the bottom of the screen.

S/L

Button selects linear (L) or sine(x)/x (S) interpolation. Linear interpolation connects the data points with straight lines. Sine(x)/x interpolation uses curves to connect the data points. This smoothed interpolation gives better waveform display at highest sine frequencies. Linear interpolation is better for step signals. The S/L selection effects only at TIME/DIV settings 0.2 and 0.1us.

Note: Sine interpolation can result in incorrect top-top signal displaying, at frequencies above 5MHz.

1GS/s

This 1GS/s sampling rate is in use on 0.2us/div, 0.1us/div, 0.05us/div and 0.02us/div ranges.

CH1 + CH2**CH1 - CH2****XY Plot****INV. CH2**

This button appears only in math mode. Toggles between math mode and normal mode.

Tip: Use Wheel Mouse's scroll wheel to fine adjust the triggering level and traces' y-position.

1.4 Connection

Connect the oscilloscope unit to the **USB** port.

For Circuit Analyzer (Bode Plotter) option connect function generator PCG10 or K8016 to **LPT1**, **LPT2** or **LPT3**.

To select the LPT port address for the function generator click **Hardware Setup** on **Options** menu or select it from **Pc-Lab2000** startup screen.

1.5 Troubleshooting

Spectrum analyzer doesn't work.

- No arithmetic coprocessor in the computer.

No signal on the oscilloscope display

- No communication with the computer (check that the cable is connected to the USB port)
- If USB cable is connected, close the program. Disconnect and reconnect the USB cable and run the Pc-Lab2000 program again.
 - RUN button is not ON.
 - The channel concerned is OFF.
 - TIME/DIV switch is in the wrong setting.
 - TRIGGER is ON, set TRIGGER OFF
 - The unit input selection is at GND.
 - Y position is wrongly adjusted.
 - Input amplitude is too large, adjust VOLTS/DIV setting.

If the above tips have no effect, then test on a different computer or different USB port.

Note. Close the program before disconnecting the USB cable.

1.6 Waveform Parameters Display

Waveform Parameters Display

When the menu option "Waveform Parameters" of the View menu is selected the software automatically calculates various voltage and time parameters of a signal, such as DC mean, amplitude, rise time etc.

These parameters are displayed in a separate window. Use the check boxes in the window to select the parameters you which to be displayed.

You may select the parameters to display for frozen waveform as well.

You may even re-open saved waveform data file to perform these measurements.

Please Note: Do not change the scope settings when the re-opened waveform parameters are to be read.

The green labeled parameters (High, Low, Amplitude, Rise time and Fall time) are mainly intended for pulse shaped waveform measurements only.

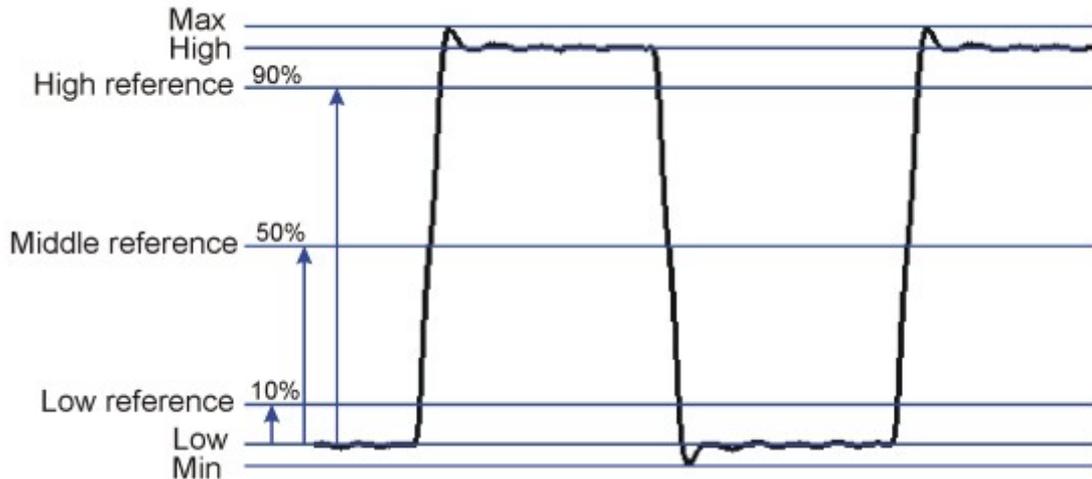
For proper waveform measurement the signal levels must be appropriate. Signal levels too small will result in noisy and inaccurate measurements. Signal levels too large will result in clipping and yield incorrect results.

Error indication:

?? Indicates that clipping has occurred.

??? Indicates that there are too few or too many wave cycles in the waveform data or the signal amplitude is too low. Also too noisy signal and variable frequency signal causes this indication.

Voltage parameters



DC Mean

The arithmetic mean of the entire waveform data.

Max

The signal's positive peak voltage.
(Difference between zero and highest value.)

Min

The signal's negative peak voltage.
(Difference between zero and lowest value.)

Peak-to-Peak

The signal's peak-to-peak voltage.
(Difference between highest and lowest value.)

High

The statistical maximum level recorded for all the cycles in the signal.

Low

The statistical minimum level recorded for all the cycles in the signal.

Amplitude

The voltage difference between the High and Low of the signal.

AC RMS

The true RMS value of the AC component of the signal is calculated and converted to voltage.

AC dBV

The measured signal (AC only) is converted to dBV (0dB= 1V).

AC dBm

The measured signal (AC only) is converted to dBm (0dB= 0.775V).

AC+DC RMS

The true RMS value of the wave (AC+DC) is calculated and converted to voltage.

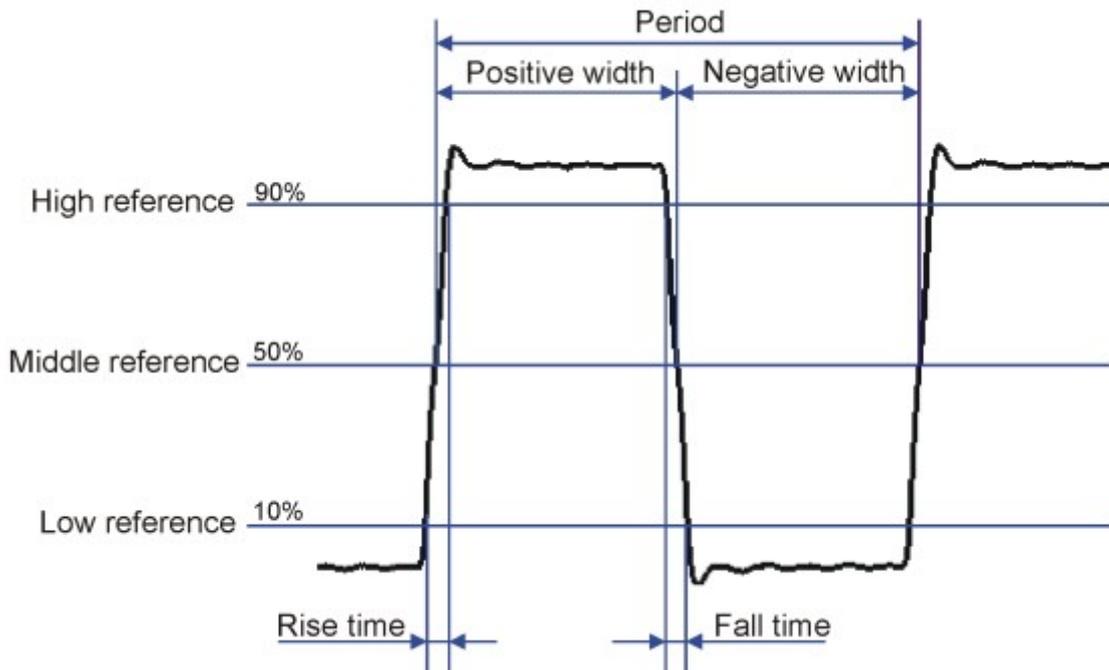
AC+DC dBV

The measured signal (AC+DC) is converted to dBV (0dB= 1V).

AC+DC dBm

The measured signal (AC+DC) is converted to dBm (0dB= 0.775V).

Time parameters



Duty Cycle

The ratio (expressed as a percentage) of the average positive pulse width to the average period of the signal. The time intervals are determined at middle reference level.

Duty Cycle = (Positive Pulse Width)/Period x 100%

Positive Width

Average positive pulse width in the waveform.

The time intervals are determined at middle reference level.

The middle reference is the middle point between the high and low levels.

Negative Width

Average negative pulse width in the waveform.

The time intervals are determined at middle reference level.

Rise Time

Time for a signal's rising edge to go from the low reference level to the high reference level.

The low reference level is 10% and high reference level is 90% of the pulse amplitude.

Fall Time

Time for a signal's falling edge to go from the high reference level to the low reference level.

The low reference level is 10% and high reference level is 90% of the pulse amplitude.

Period

The time interval between two consecutive crossings on the same slope of the signal at the middle reference level.

Frequency

The inverse of the Period of the signal.

Phase

Phase angle in degrees between CH1 and CH2.

For phase measurement the frequency of CH1 must be equal to the frequency of CH2.

Phase measurement is time consuming process. In slow PCs this reduces the display update rate.

1.7 Add comment text in the signal screen

For explanation and documentation, each measurement can be supplied with a comment text.

This text will be saved together with the waveform data to the disk file.

To enter the text:

1. Right mouse click into the screen.
2. Text box will open, to write your comment.
3. Click **Add Text on Screen** or **Remove** to remove previously inserted text.
4. Right click on the screen to position your text.
5. Click **Close**.

To make the text transparent with the background, check **Transparent text**. The text will have the same color as the vertical time/frequency markers.

2 Menu Options

[File Menu](#)

[Edit Menu](#)

[Options Menu](#)

[View Menu](#)

[Math Menu](#)

[Help Menu](#)

2.1 File Menu

Note: Default subdirectory (folder) **\DATA** for image and data files is created when the program is run the first time.

Open Image

Opens an image file and displays it on the screen.

Open DSO Data

Opens and displays the waveform data saved in text format using the **Save DSO Data** option.

Save Image

Saves the image to a file in Windows Bitmap (*.BMP) format.

Save DSO Data

Saves the waveform data in text format. All captured data (4096 points/channel) is saved.

Save FFT Data

Saves the FFT data in text format. Only the portion of the data displayed on the screen is saved (250 points).

Save Settings

Saves the Oscilloscope, Spectrum Analyzer and Transient Recorder settings to a file. Also Function Generator settings (frequency, amplitude, offset and duty cycle) are stored to the file.

Recall Settings

Loads a previously-stored settings file to the oscilloscope.

Print

Prints the image in color.
You can edit the image caption.

Print Setup

Selects a printer and sets printer options before printing. The available options depend on the printer you select.

Exit

Terminates the program.

Calibrate & Exit

Makes the oscilloscope calibration, saves the calibration values to the WinDSO.INI file and terminates the program. This option should be used when the new oscilloscope has been on for about 1h.

This option performs following operations:

1. The fine adjustment of the trace Y-position (offset) on different Volt/Div and Time/Div scales.
2. Sets the trace labels (on the left side of the screen) to correspond the trace GND level.
3. Sets the triggering level mark to correspond the triggering level.

2.2 Edit Menu

Copy

Copies the image to the Windows' clipboard.

Paste

Pastes the image residing in Windows' clipboard to the screen.

2.3 Options Menu

FFT Window

Set the window function used to taper the original signal before calculating the FFT..

Spectrum analyzer supports six different FFT windows

1. Rectangular
2. Bartlett
3. Hamming
4. Hanning
5. Blackman
6. Flat top

The Hamming window is set as default at the startup

Background information

It is common practice to taper the original signal before calculating the FFT (Fast Fourier Transformation). This reduces any discontinuities at the edges of the signal. This is done by multiplying the signal with a suitable window function. By choosing a tapered window it is possible to achieve a good compromise between main lobe width and sidelobe levels of a spectral line. The undesirable spectral leakage can be reduced using a tapered window, at the expense of some broadening around individual spectral lines. Many different windows have been designed for this purpose.

The choice of a suitable one depends on the nature of the signal or data, and on the type of information to be extracted from its spectrum. In general, a good FFT window has a narrow main spectral lobe to prevent local spreading of the spectrum, and low sidelobe levels to reduce 'distant' spectral leakage. In some cases it may be best to leave the data alone -- in effect, to use a rectangular window. For example, if a signal has close spaced components of roughly the same amplitude, the rectangular window will probably offer the best chance of resolving them. Conversely,

if the amplitudes are very different, a window with low sidelobes will reduce leakage around the large component, and should make the small one easier to detect.

Comparison of the window functions

Window type	Window characteristics	Applications
Rectangular	Narrow mainlobe Slow rolloff rate	Broadband random (white noise) Closely spaced sine-wave signals
Bartlett	Narrow mainlobe Fast rolloff rate	
Hamming	Good spectral resolution Narrow mainlobe	Closely spaced sine-wave signals
Hanning	High maximum sidelobe level Good frequency resolution Reduced leakage Fast rolloff rate	Narrowband random signals Sine wave or combination of sine-wave signals
Blackman	Wide mainlobe Fast rolloff rate	
Flat top	Good amplitude accuracy Wide mainlobe Poor frequency resolution More spectral leakage	Sine wave with need for amplitude accuracy

FFT Options

Maximum

Maximum value of each frequency is displayed in Run mode.

This option can be used for recording signal levels as a function of frequency (Bode plot). You can use spreadsheet to display the frequency response curve including the frequency labels. On **File** menu, click **Save FFT Data** to export the data to the spreadsheet.

RMS Average

Use this averaging mode to reduce signal fluctuations.

RMS averaging provides an excellent estimate of the true signal and noise levels of the input signal.

Vector Average

Use this averaging mode to reduce random or uncorrelated noise in the synchronous signal you want to display.

Vector averaging requires a trigger - set **Trigger ON**.

The signal of interest must be both periodic and phase synchronous with the trigger.

Vector averaging reduces the noise floor for random signals since they are not phase coherent from time record to time record.

If not triggered, the signal will not add in phase and instead will cancel randomly.

Hardware setup

Select the LPT port where the hardware is connected.

Operation mode

1. Oscilloscope connected to **USB** port.
2. Demo mode. (No hardware needed).

Select the LPT port address where the Function Generator PCG10 or K8016 is connected 378, 278 or 3BC

You'll find the address from Windows' Device Manager:

1. Click "System" icon in Control Panel and then the Device Manager tab.
2. Click the plus sign next to the "Ports".
3. Double-click "Printer Port (LPTx)".
4. Click the Resources tab to see the Input/Output address.

Select the LPT port communication speed for the Function Generator

Normal

This can be used in most cases.

Slow

Select this option if the waveform of the function generator is corrupte

Colors

Select the display colors.

Select the color for various items on the waveform display.

To change the color of an item, click the corresponding button. This will open a dialog in which you can select the new color.

Full color selection is possible only if True Color (24 bit) palette is used. There are restrictions in the color combinations with other palettes.

Click **Bright Screen** or **Black Screen** button to resets all colors to the **Default settings**.

Trigger Options

Noise Reject

Select this option to get stable triggering on noisy signals.

This option works only in the Run mode and only in the Real-time Sampling mode.

2.4 View Menu

RMS value

Displays AC RMS value of the signal

When this option is selected the true RMS AC value of the signal is displayed on the screen.

- If CH1 is **on** the RMS value of CH1 is shown
- If CH1 is **off** the RMS value of CH2 is shown

dBm Value

Displays AC dBm value of the signal

Sample Rate

Displays the sampling rate on the top of the screen.

When this option is selected the dBm value of the signal's AC component is displayed on the screen.

- If CH1 is **on** the dBm value of CH1 is shown
- If CH1 is **off** the dBm value of CH2 is shown

The dBm value displayed on the screen:

0 dBm = 1 milliwatt at 600 ohms (0.775 Vrms)

Waveform Parameters

Software automatically calculates various voltage and time parameters of a signal, such as DC mean, amplitude, rise time etc.

These parameters are displayed in a separate window. Use the check boxes in the window to select the parameters you which to be displayed.

Markers

Displays Markers on the screen.

Bright Grid

Brightens the signal grid on the screen.

Dot Join

On: The dots of the acquired waveform data are connected by lines.

Off: Only the dots of the acquired waveform data are displayed.

Markers in oscilloscope mode

- Two horizontal markers for measuring voltage. The voltage difference and the absolute voltage values (in parentheses) are displayed.
- Two vertical markers for measuring time and frequency

Note: The voltage markers give preference to channel CH1 if both channels are being used.

Markers in spectrum analyzer mode

- Marker function for absolute and relative voltage measurement is provided.
- The absolute voltage level in dBV or the voltage difference in decibels (dB) can be measured.
- Noise level can be measured using the [Spectral Density](#) ^[14] marker.
- One vertical marker is provided for the frequency measurement.

Moving the markers

1. Place the mouse pointer over a dashed marker line.
2. Press and hold the left mouse button. The markerline turns solid.
3. Drag the marker to the appropriate position

dB

The term **dB** or **decibel** is a relative unit of measurement used to describe power or voltage difference.

Equation to calculate a dB value based on the ratio of two voltages V2 and V1 is:

$$dB = 20 \log_{10} \frac{V2}{V1}$$

dBV

dBV = The dB value is obtained with respect to 1Volt. The dBV is an absolute unit of voltage. It expresses voltages as a ratio relative to 1 volt.

Equation to calculate a dBV value of a voltage V is:

$$dBV = 20 \log_{10} V$$

Equation to calculate voltage V from dBV value is:

$$V = 10^{\frac{dBV}{20}}$$

dBm

A unit of measurement of signal level in an electrical circuit, expressed in decibels referenced to 1 milliwatt.

In a circuit with an impedance of 600 ohms, 0 dBm gives an equivalent voltage level of 0.775 Vrms.

The dBm value displayed on the screen:

0 dBm = 1 milliwatt at 600 ohms (0.775 Vrms)

2.5 Spectral Density

The Spectral Density marker may be used when measuring the density of random or noise signals since it properly takes into account the frequency bin width and the FFT window function used by the spectrum analyzer when measuring noise-like signals.

The Spectral Density marker readout is automatically normalized to 1 Hz.

The displayed unit is: $\text{dBV}/\sqrt{\text{Hz}}$

Note: The Spectral Density marker should not be used to measure discrete frequency components as it will provide misleading level readings.

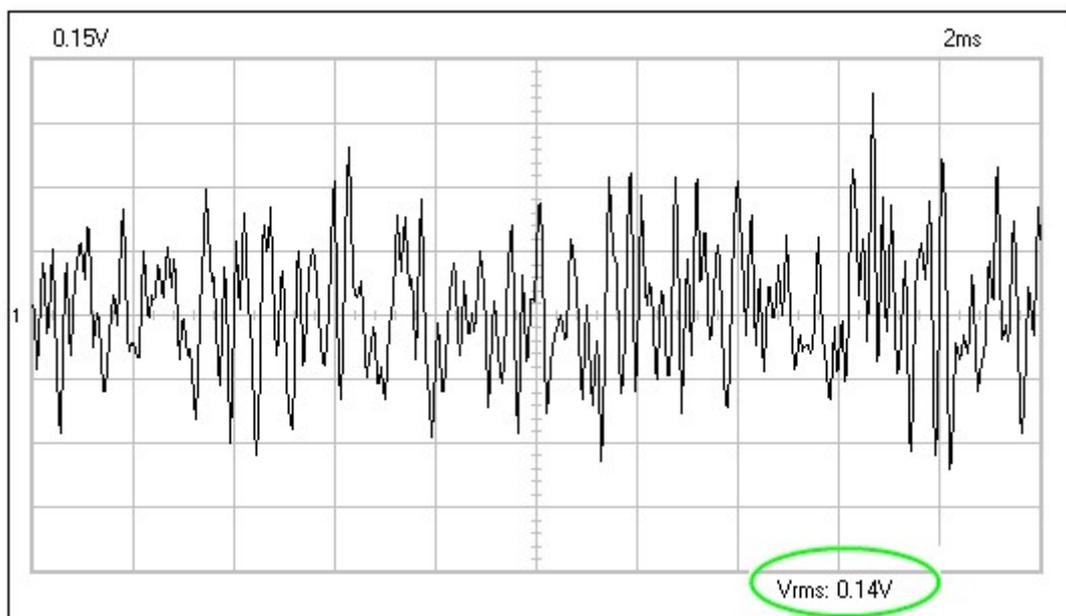
The Spectral Density is simply the magnitude of the spectrum normalized to a 1 Hz bandwidth. This measurement approximates what the spectrum would look like if each frequency component were really a 1 Hz wide piece of the spectrum at each frequency bin.

When measuring broadband signals such as noise with a spectrum analyzer, the amplitude of the spectrum changes with the frequency span. This is because the FFT bin width changes and the frequency bins have a different noise bandwidth.

The Spectral Density marker normalizes all measurements to a 1 Hz bandwidth and the noise spectrum becomes independent of the frequency span. This allows measurements with different spans to be compared.

If the noise is Gaussian in nature, then the amount of noise amplitude in other bandwidths may be approximated by scaling the Spectral Density measurement by the square root of the noise bandwidth.

Example:



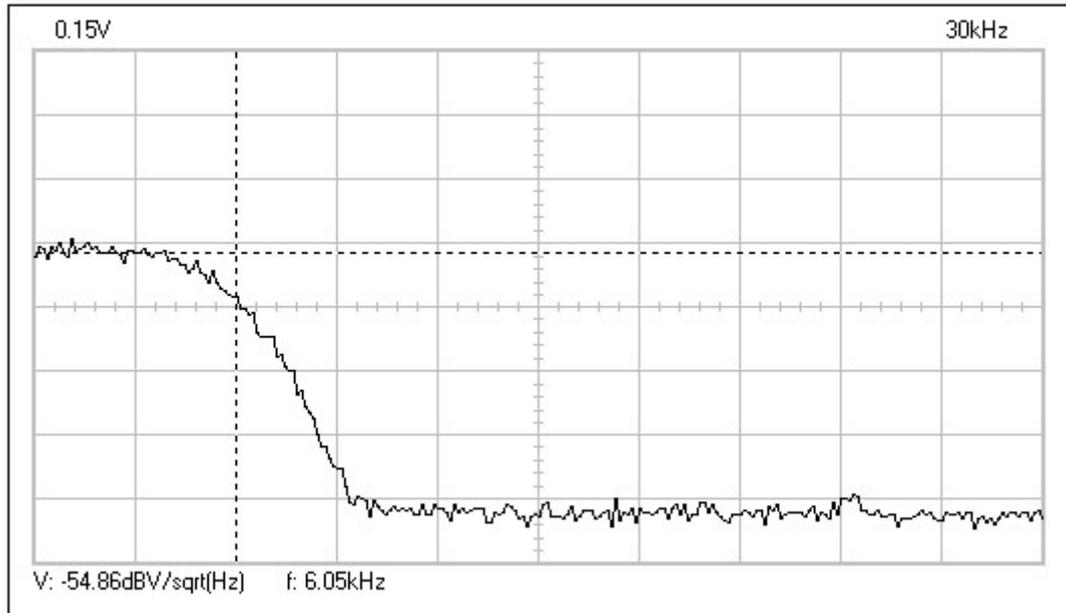
This image shows a band limited noise signal on an oscilloscope screen.

Spectrum analyzer can be used to measure the spectral density of this noise signal.

In the spectrum analyzer select the following menu options:

- Options / FFT Options / RMS Average

- View / Markers (FFT) f & Spectral Density dBV/sqrt(Hz)



This image shows the spectrum of the band limited noise signal.

The analysis of noise in the frequency domain shows the distribution of the noise amplitude as a function of frequency.

Using the **Spectral Density** marker and the Frequency marker, the voltage spectral density (V_{SdBV}) and the noise bandwidth (B_N) can be read from the spectrum analyzer display.

$$V_{SdBV} = -54.86 \text{ dBV}/\sqrt{\text{Hz}}$$

$$B_N = 6 \text{ kHz}$$

First convert the voltage spectral density to $V/\sqrt{\text{Hz}}$.

This can be accomplished using the following calculation:

$$V_S = 10^{-54.86/20} = 0.0018 \text{ V}/\sqrt{\text{Hz}}$$

This is the magnitude of the spectrum normalized to a 1 Hz bandwidth.

You may calculate the noise voltage over any bandwidth by multiplying this value by the square root of the bandwidth.

Assuming a 6 kHz bandwidth, the total output noise voltage is:

$$V_{rms} = V_S \sqrt{B_N}$$

$$V_{rms} = 0.0018 \text{ V}/\sqrt{\text{Hz}} * \sqrt{6000 \text{ Hz}} = \underline{0.139 \text{ V}}$$

(See the V_{rms} value of the oscilloscope waveform image of this noise signal.)

2.6 Math Menu

The result of mathematical operation of channel 1 and 2 is displayed. One of the following functions can be selected:

- Ch1 + Ch2
- Ch1 - Ch2
- XY Plot
- Invert Ch2

XY Plot:

Ch1 data is displayed on Y axis

Ch2 data is displayed on X axis

A button is provided to toggle between Math mode and Normal mode.

2.7 Help Menu

Contents

Displays this help file

Installing Windows NT4 driver

Gives instructions for the Windows NT4, Windows 2000 and Windows XP users

About

Displays information of the program version

3 Data Transfer

[Data acquisition to other applications](#)

[Data acquisition to Microsoft Excel](#)

3.1 Data acquisition to other applications

Pc-Lab2000 software package includes a DLL (Dynamic Link Library) *DSOLink.DLL*, installed to Windows' *SYSTEM32* folder.

This DLL allows you to write custom applications in Excel, Visual Basic, Delphi or any other 32-bit Windows application development tool that supports calls to a DLL.

The DLL gives you direct access to real-time data and settings information from the oscilloscope.

The complete example programs are located on the VELSOFT CD. Those may be used as a starting point how to construct your customized application programs.

Note: Before running the following example programs: The oscilloscope software must be running and **Run** or **Single** button pressed and trace displayed on the oscilloscope screen.

Description of the procedures of the DSOLink.DLL

ReadCh1
ReadCh2

Syntax

```
PROCEDURE ReadCh1(Buffer: Pointer);
PROCEDURE ReadCh2(Buffer: Pointer);
```

Parameter

Buffer: A pointer to the data array of 5000 long integers where the data will be read.

Description

Read all the data and the settings of channel 1 or channel 2 of the PCSU1000.

As a return the following data is put to the buffer:

- [0] : Sample rate in Hz
- [1] : Full scale voltage in mV
- [2] : Ground level in A/D converter counts. The value may be beyond the 0...255 range if GND level is adjusted beyond the waveform display area.
- [3...4098] : The acquired data in A/D converter counts (0...255), from PCSU1000.

The triggering point of the PCSU1000 is at the data location [1027].

Running the DSOLink in Delphi

Check the **IPC-Lab2000 tools\PCSU1000 - PCS500 - PCS100 - K8031\Data transfer DSOLink_DLL\DSOLink_Demo_VB** folder on the Velleman CD to locate the demo files. This folder contains a ready to run *DSOLink_Demo.EXE* program and its source code. You may copy the files to any folder and use Delphi to examine, edit and compile the files.

Example (in Delphi)

```
var
  data: array[0..5000] of longint;

procedure ReadCh1(Buffer: Pointer); stdcall; external 'DSOLink.dll';

procedure TForm1.Button1Click(Sender: TObject);
var i: longint;
    p:pointer;
begin
  p:= @data[0];
  ReadCh1(p);
  mem1.clear;
  mem1.lines.add('Sample rate [Hz]'+chr(9)+inttostr(data[0]));
  mem1.lines.add('Full scale [mV]'+chr(9)+inttostr(data[1]));
  mem1.lines.add('GND level [counts]'+chr(9)+inttostr(data[2]));
  mem1.lines.add('');
  begin
    for i:=0 to 20 do
      mem1.lines.add('Data ('+inttostr(i)+')'+chr(9)+chr(9)+inttostr(data[i+3]));
  end;
end;
```

Running the DSOLink in Visual Basic

Make sure that the file *DSOLink.DLL* is copied to Windows' *SYSTEM32* folder. Check the **IPC-Lab2000 tools\PCSU1000 - PCS500 - PCS100 - K8031\Data transfer DSOLink_DLL\DSOLink_Demo_VB** folder on the VELSOFT CD to locate the demo files. This folder contains a ready to run *DSOLink_Demo.EXE* program and its source code. You may copy the files to any folder and use Visual Basic to examine, edit and compile the files.

Example (in Visual Basic)

Option Explicit

```
Dim DataBuffer(0 To 5000) As Long
Private Declare Sub ReadCh1 Lib "DSOLink.dll" (Buffer As Long)

'This reads the settingsd and 4096 bytes of data from CH1 to the data buffer.
'The first 21 values are displayed.

Private Sub Read_CH1_Click(Index As Integer)
    Dim i As Long
    List1.Clear
    ReadCh1 DataBuffer(0)
    List1.AddItem "Sample rate [Hz]" + Chr(9) + Str(DataBuffer(0))
    List1.AddItem "Full scale [mV]" + Chr(9) + Str(DataBuffer(1))
    List1.AddItem "GND level [counts]" + Chr(9) + Str(DataBuffer(2))
    List1.AddItem ""
    For i = 0 To 20
        List1.AddItem "Data(" + Str(i) + ")" + Chr(9) + Chr(9) + Str(DataBuffer(i + 3))
    Next
End Sub
```

Running the DSOLink in Borland C++ Builder

The following files are available in the **IPC-Lab2000 tools\PCSU1000 - PCS500 - PCS100 - K8031\Data transfer DSOLink_DLL\DSOLink_Demo_BCB** folder on the VELSOF CD for development with Borland C++Builder:

DSOLink.dll the dynamically linked library
DSOLink.h the C/C++ header file for function prototypes
DSOLink.lib the import library
DSOLink_demo.cpp demo source

1. Create a new project in Borland C++ Builder.
2. Add the import library to your project using **Project | Add to Project** menu option.
3. Add a `#include` statement in the main unit that includes *DSOLink.H*.
4. Finally, add code that calls the DLL functions.

DSOLink.h

```
//-----
// DSOLink.h
#ifdef __cplusplus
extern "C" {          /* Assume C declarations for C++ */

#endif

#define FUNCTION __declspec(dllimport)

FUNCTION __stdcall ReadCh1(int* ptr);
FUNCTION __stdcall ReadCh2(int* ptr);

#ifdef __cplusplus
}
#endif
//-----
```

Example (in Borland C++Builder)

```
//-----
// DSOLink_demo.cpp

#include <vcl.h>
#pragma hdrstop
#include "DSOLink.h"
#include "DSOLink_demo.h"
//-----
#pragma package(smart_init)
#pragma resource "*.dfm"
TForm1 *Form1;
//-----
__fastcall TForm1::TForm1(TComponent* Owner)
    : TForm(Owner)
{
}
//-----
void __fastcall TForm1::Button1Click(TObject *Sender)
```

```

{
    int data[5000];
    ReadCh1(data);

    Mem01->Clear();
    Mem01->Lines->Add("Sample rate [Hz]: "+IntToStr(data[0]));
    Mem01->Lines->Add("Full scale [mV]: "+IntToStr(data[1]));
    Mem01->Lines->Add("GND level [counts]: "+IntToStr(data[2]));
    Mem01->Lines->Add("");

    for (int i = 0; i < 20; i++)
    {
        Mem01->Lines->Add("Data "+IntToStr(i)+char(9)+IntToStr(data[i+3]));
    }
}
//-----

```

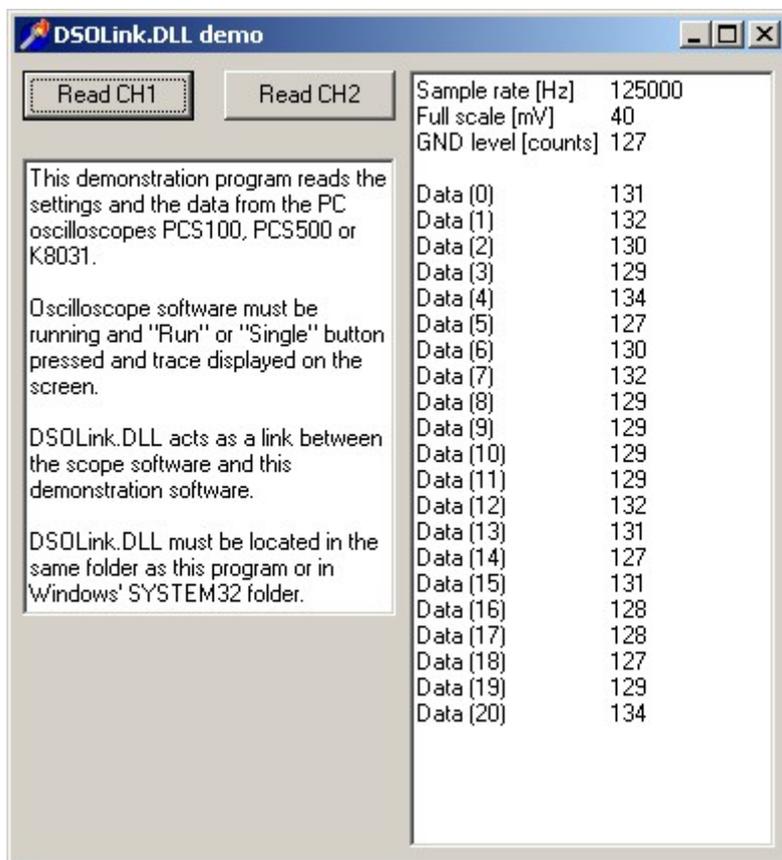
Note: If the import library is not compatible to your Borland C++ version, you can create an import library by running IMPLIB on the DLL.

IMPLIB works like this:

```
IMPLIB (destination lib name) (source dll)
```

For example,

```
IMPLIB DSOLink.lib DSOLink.dll
```



3.2 Data acquisition to Microsoft Excel

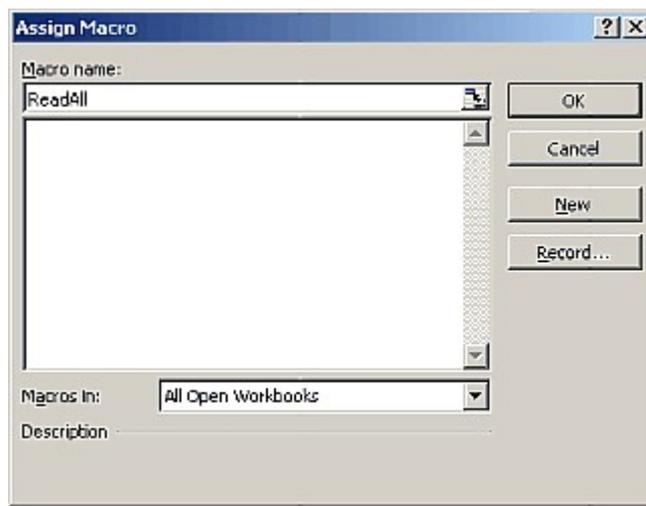
Pc-Lab2000 software package includes a DLL (Dynamic Link Library) *DSOLink.DLL*, installed to Windows' *SYSTEM32* folder.

This DLL allows you to write custom applications in Excel, Visual Basic, Delphi or any other 32-bit Windows application development tool that supports calls to a DLL.

Transferring Waveform Data to an Excel Spreadsheet

The example Excel macro shows you how to collect data directly into the spreadsheet from the Velleman PC oscilloscopes without the need for other software.

1. Open **Microsoft Excel** and start a new workbook document.
2. Select the **View / Toolbars** Menu and Select **Forms**.
 - The Forms toolbar appears.
3. Create a **Button**
 - In the Forms toolbar click on the "Button" button: the mouse-pointer will become a small cross.
 - On Excel worksheet, use the mouse to draw a rectangle to mark where you want your button to appear.
 - When you release the mouse after drawing the rectangle the "Assign Macro" dialog-box will appear.



4. Type Macro name: **ReadAll** and click **New** button.
 - Microsoft Visual Basic edit window will open. A subroutine called *ReadAll* has been created.
5. Substitute the default text:

```
Sub ReadAll()  
End Sub
```

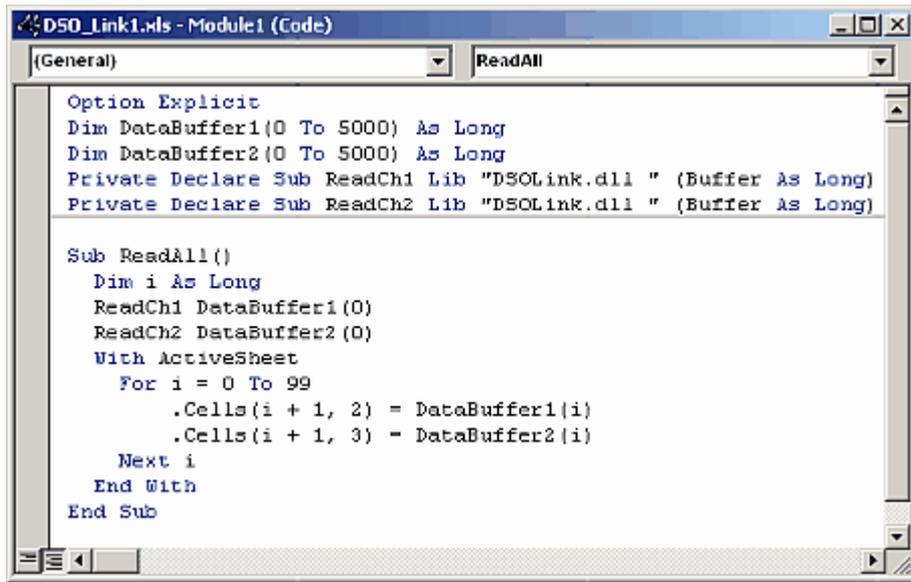
with the following text in the edit window:
(Use Copy and Paste.)

```
Option Explicit  
Dim DataBuffer1(0 To 5000) As Long  
Dim DataBuffer2(0 To 5000) As Long  
Private Declare Sub ReadCh1 Lib "DSOLink.dll" (Buffer As Long)  
Private Declare Sub ReadCh2 Lib "DSOLink.dll" (Buffer As Long)
```

```

Sub ReadAll()
  Dim i As Long
  ReadCh1 DataBuffer1(0)
  ReadCh2 DataBuffer2(0)
  With ActiveSheet
    For i = 0 To 99
      .Cells(i + 1, 2) = DataBuffer1(i)
      .Cells(i + 1, 3) = DataBuffer2(i)
    Next i
  End With
End Sub

```



6. Press **Alt+F11** to return to Excel.
7. Type following texts to column **A**:

Sample rate [Hz]
Full scale [mV]
GND level [counts]
Data 0
Data 1
Data 2
...

8. Start oscilloscope program for PCSU1000, PCS500, PCS100 or K8031 and click **Run** or **Single** button.
9. Click the button on the Excel worksheet. The created macro will execute and the data described in column **A** will appear to the worksheet columns **B** and **C**.
 - Rows 4...4099 contain the acquired data in A/D converter counts (0...255) for PCSU1000 and PCS500.
 - Rows 4...4083 contain the acquired data in A/D converter counts (0...255) for PCS100 and K8031.
 - The triggering point of PCSU1000 and PCS500 is on row 1030 and of PCS100 and K8031 on row 4.

The three first rows contain the scope settings and the rest of the rows contain the raw oscilloscope data in A/D converter counts (0...255).

Using the **Sample rate**, **Full scale** and **GND level** values it is possible to reconstruct the waveform data into engineering units (e.g. Volts and seconds) for further analysis.

